Paul Harden, NA5N • na5n@zianet.com

Part 1

The Handyman's Guide to – HOMEBREW CONSTRUCTION PRACTICES

From Copper to Manhattan

Prepared for and published in AmQRP's Homebrewer #6 (Oct. 2005)

What is the difference between a QRPer and a Homebrewer? Not much, from what I have seen over the years. Where you'll find a QRPer, you'll generally find someone who loves to build his own equipment.

When it comes to homebrewing, there are two QRPer's that have set the standard (in my opinion). First is Bill Jones KD7S. Bill's homebrew gear, mostly crafted from ABS plastic, sets the standard that rivals professional equipment. Second is Jim Kortge K8IQY and his now famous 2N2/40 built *Manhattan style*. While this method of construction has been around for years, and many will argue who actually "invented" it, there is no doubt that Jim's 2N2/40 elevated it to a whole new level. The craftsmanship of these two master builders sets the standard many homebrewer's now strive to achieve.

Due to the continuing interest in these "build it from scratch" construction techniques, George Heron and Joe Everhart asked if I could prepare a basic guide for the **Homebrewer** based on some of the gear I have built or seen – which I am pleased to attempt. However, I make no pretenses that this is the complete guide to homebrewing. More precisely, it might be called *"Homebrewing Using Copper"* – and for good reason. Copper clad is readily available at hamfests and from many vendors (I get mine from Electronic Goldmine). Copper clad is very easy to work with, not only for the "circuit board," but for the construction of the enclosure and front and rear panels. It is also the main staple of "Manhattan Style" construction. And, best of all – it's fairly cheap!

I have used copper clad and Manhattan Style for many years myself, both for my QRP homebrewing, and prototyping circuits at work. Examples of both will be presented here.

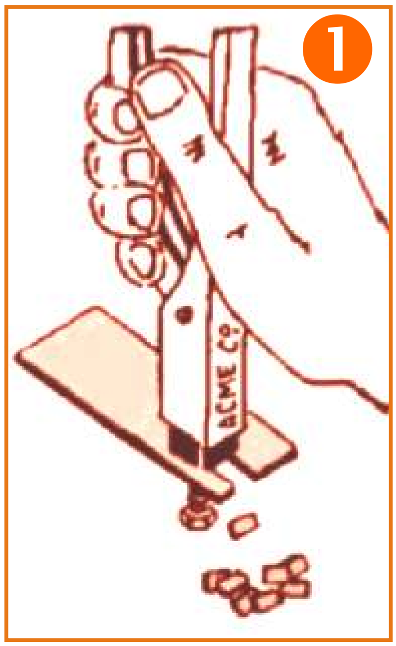
1. Let's Get Started . . .

MANHATTAN STYLE . . . What is it?

Simply put, ***Manhattan Style*** of construction uses small pieces of copper clad (the "pads") glued to the main copper clad circuit board (the "substrate") that serve as component mounting platforms. The electronic components are then mounted and soldered onto these pads. The main "substrate" board serves as the ground plane. Not only is this technique an easy and neat way to build a circuit, it also produces a very quiet circuit due to the solid ground plane.

When Jim Kortge, K8IQY, submitted his 2N2/40 at the FDIM building contest, one of the judges, Chuck Adams, K7QO, commented how the construction technique, with the IC's and electrolytic capacitors in neat rows, looked like an aerial view of Manhattan. Thus, Chuck is credited with dubbing it *Manhattan style* – the term it is well known as today amongst QRPers.

Making the "pads." There are numerous methods to make the pads. The most popular and easiest is using a ***nibbling tool*** to *nibble* out small pieces of copper clad from a larger piece, as shown in **Figure #1**. A nibbling tool costs $20 or less and used for making square cut-outs in 1/8" (max.) aluminum, such as for mounting a meter. The tool easy nibbles through .031" or .062" copper clad. The chards from the nibbling tool forms the pads, about 1/16" x 3/16". (Known as "chads" in Florida!).



1 Using a nibbling tool

Others make round or circular pads with a ***hand-punch*** tool from Harbor Freight or other sources. Dies of various sizes can be purchased for the hand-punch tool, with 3/16" or 1/4" diameter being popular sizes The tool punches-out holes in a piece of copper clad. The punched out material serves as the small, circular pads for Manhattan construction.

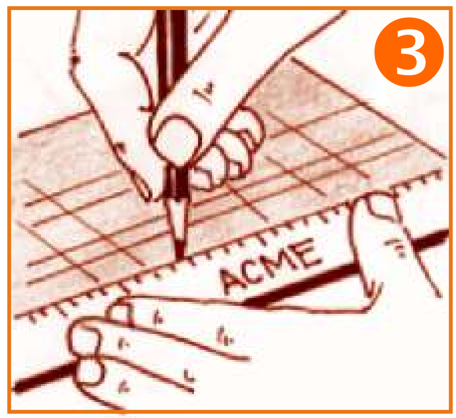
Still another method is to snap-off pads from a piece of ***perforated copper clad board*** as shown in **Figure #2**. The pads are twisted off with a pair of needle nose pliers or cut apart by a hefty pair of wire cutters. These pads are not as "pretty" as those made by a nibbling tool or circular hand punch, but work equally as well. The board can also be cut by following the perforated holes, using a coping or hack saw, to produce long strips, which can be cut-off at the desired length. One advantage of this technique is it allows you to make long strips that can serve as the +Vcc bus or making longer runs without having to connect two smaller pads with a jumper wire.

Using perforated copper clad circuit board for making the pads is another method, requiring no special tools.

Pads can be made from .031" or .062" thick copper clad, single sided or double sided.

Once the pads are made, it's a matter of placing them on the main circuit board for mounting the components. Before gluing on the pads, it is best to plan ahead.

Laying out the circuit. It is recommended to lay-out your circuit on a piece of paper, arranging the components in a logical circuit manner, similar to laying out a printed circuit board with paper and pencil. This will ensure that all of the components will fit on the size of copper clad board you have selected as the circuit board or substrate. One can build Manhattan Style by "building as you go," but problems fitting components, working yourself into a corner, or ending up with long wire runs reaching front panel controls can occur. Planning ahead by laying out the circuit first is by far the best way to ensure the finished product is correct to the circuit, functional, and the final appearance is nice and neat.



Draw footprints of each section and guidelines with pencil on the copper clad board. Planning ahead is important!

Once this is done, transfer the layout to the copper clad board with a ruler and pencil as shown in **Figure #3**. This provides guidelines for gluing down the pads and keeping things straight, square and symmetrical.

Gluing down the pads. Once the circuit has been layed-out, it is time to mount the pads on the main substrate board with small drops of super glue, as shown in **Figure #4**. And, small drops is the secret! Learn to issue a very small drop, smaller than the size of the pad, to keep excess from being squeezed out over the board when you apply the pad. It takes a little practice, but you can learn to apply the right amount with little waste.

There are many opinions as to what type of super glue works the best. Some prefer one brand over another, some prefer the gels. I have tried them all and have found little difference between them other than personal preference. I build most of my Manhattan circuits with the cheapest glue I can find, which is usually Duro-Bond Super Glue, with two tubes per package costing $1.79 or less at Wal-Mart or local hardware stores. The small "snout" on the tube is also relatively easy to keep clean and open.

Pads are "mounted" to the main circuit board with glue – usually with super glue. The secret is learning to administer a small drop of glue. This comes with practice.

The biggest problem I have found with different manufacturers or with the exotic applicators is keeping them clean. They work great – the first time.

But, when you come back to work on the project the following night, that fancy $5 tube has super-glued itself shut. You either can't get the protective cap off, or the tube has turned into a solid brick. Time for a new tube. A couple of cheap tubes of super glue goes a long ways when this happens.

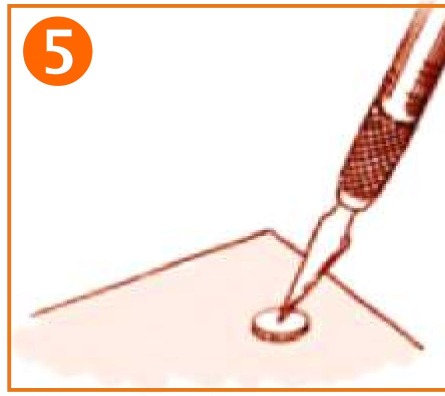
To avoid these problems, I usually do two things when I'm done for the day:

1) Remove the applicator tip and run a resistor lead down the spout to open up the channel from excess glue. The excess may run or drip out the end. This will ensure the applicator tip is "open" when you place the tip back onto the tube. Without doing this, the super glue left in the tip can turn solid and hard, preventing it from being used again.

2) Clean the applicator tip and protective cap with a Q-tip or paper towel soaked in alcohol or acetone. Clean off all access glue, particularly on the threads for the protective cap. Then, clean dry with a piece of paper towel. If the paper doesn't stick – it's clean! This will ensure you'll get the protective cap off the next time you use the tube of glue.

These two simple cleaning steps can keep a tube of super glue useful for a long time. If not, that's why the cheap tubes of super glue should be used.

Positioning the "pads." The pad is placed on the drop of super glue and positioned into exact placement with an Exacto knife or other sharp object, as shown in **Figure #5 and #6**. For the first few seconds, the super glue will be slippery, allowing the pad to be easily positioned. Once in the desired position, push down on the pad against the main board to squish against the glue. It will be solidly glued into place in a few seconds.

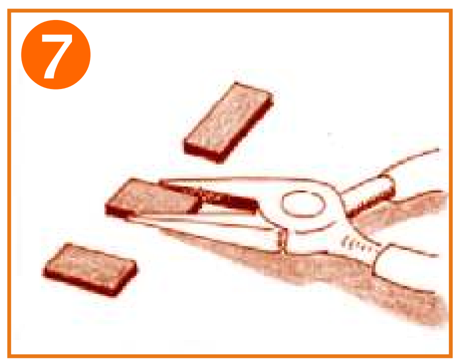


Place the pads onto the drop of glue and position with an Exacto knife, other sharp object or tweezers..

The method I often use is to place the pad into position with a pair of sharp, needle nosed tweezers. Once in position, I push down on the pad with a small screw driver or the wooden shaft of a Q-tip. However, the tweezer method does not work well when using the punched-out circular pads.

After several seconds, the pad should be firmly attached to the substrate board. Some of the circuits I have built years ago using this method have the pads still firmly affixed to the board.

To remove a pad that got positioned in the wrong place, simply "twist it off" the board with a pair of needle-nosed pliers, as shown in **Figure #7**. Any pad that becomes dislodged from the board can be simply re–glued into place with a new drop of glue and holding in place for a few seconds.



Remove a pad by a twist with a pair of needle-nosed pliers.

Cleaning up. Once the excess glue had dried, it can be scraped off the board with a hobby knife or a small flat blade screw driver. It's up to the builder how picky one wishes to be with this. At a minimum, the board and pads should be cleaned with a hobby brush or toothbrush moistened with alcohol or acetate to remove oils, fingerprints and debris. This will make for easier soldering and a nicer appearance.

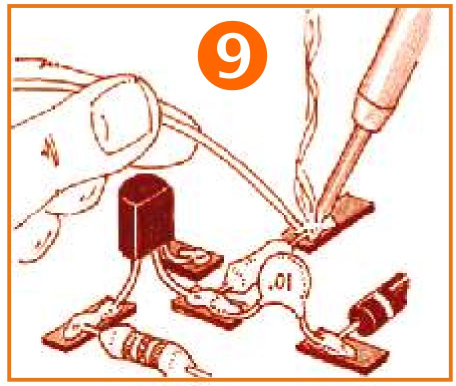
Acetone dissolves dried super glue better than alcohol. It is easily obtainable as fingernail polish remover in many stores. However, most fingernail polish remover sold today is "acetone free." Ensure you get a bottle that contains real acetone. I get a bottle of acetone based fingernail polish remover from Wal-Mart that works quite well. It costs 88 cents for a pint bottle and usually lasts for several projects.

Clean board and pads with a brush and alcohol or acetone.

Melt solder! Once the pads are in place and cleaned, there is nothing left to do except mount the components onto the pads and solder in place according to the layout drawing. Of course, ground connections are soldered directly to the main substrate board, being the circuit ground plane, as shown in **Figure #9**.

I use a small hobby brush with hair or fiber bristles (not steel) for cleaning the pads. I use the same brush moistened with alcohol or acetone for cleaning the pads after soldering. This removes excess flux and debris, leaving a nice, shiny soldered pad.

Tools. In addition to the obvious — a soldering iron, wire cutters and the small pliers already discussed, several other small tools come in handy:
Tweezers are handy for positioning the pads when gluing to the board, in addition to holding small parts while soldering – particular surface mount components. *(Surface mount techniques will be presented in Part 2).*
Hemostats are another useful small tool for holding resistors or capacitors while soldering. They are locking, making it easy to hold the component with one hand while soldering with the other. Just ensure you don't over-squeeze the component to cause damage. Hemostats often allow a component to be held with a better grip than with tweezers.
Small screwdriver, flat-blade or phillips, is useful for holding down pads while the glue dries, pushing down ill-bent or stubborn component leads while soldering – even a lead bender to ensure smooth bends on component leads and internal wiring.
Q-tips are handy for cleaning or scrubbing around the pads after soldering, where a hobby brush may not often reach. Lightly moistened with alcohol or acetone, they are also useful for cleaning the components. When cut in two, the wooden shafts are also handy for holding down pads during gluing, or components while soldering.



The glued pads become the mounting platforms for the components, soldered in place.

Some of the small tools useful when building Manhattan style.

2. Some Practical Examples

THE ROCKMITE QRP TRANSCEIVER

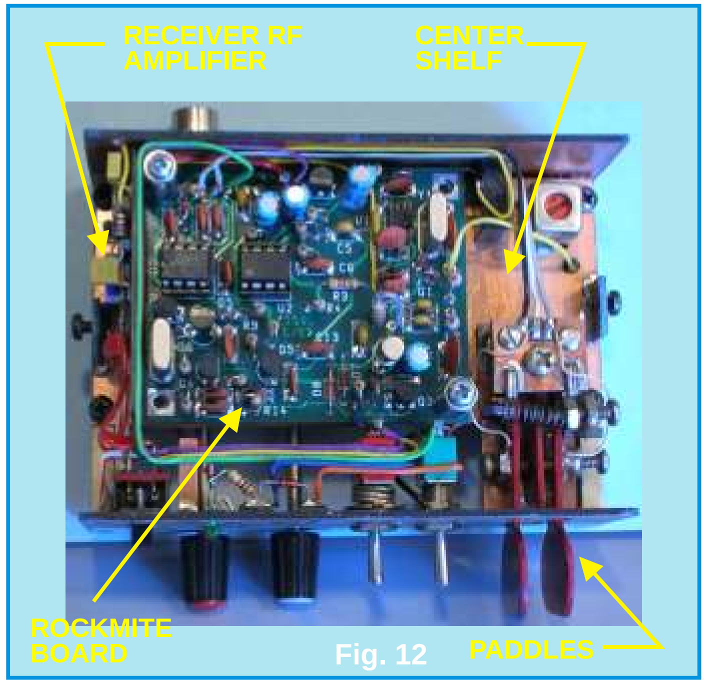
Like hundreds of others QRPers, I built a ***Rockmite*** about two years ago when the kit was first introduced. The Rockmite QRP transceiver is a kit from Small Wonders Lab, furnished with a printed circuit board. I decided to highly modify mine, building it in a custom enclosure with a set of homebrew built–in paddles, something I always wanted to do. Additionally, it served as a test platform for a 5W Class-E PA circuit. The entire rig, including the paddles, was built of copper clad, except for the top cover, made from a scrap piece of perforated aluminum and painted black.

Fig. 11

The front panel, shown in **Figure 11**, was made from a piece of copper clad. Holes were drilled and the "square holes" for the power switch and paddles were filed to shape with a small jewelers file. After drilling, the copper clad was brushed with emory paper to rough up the copper a bit before applying a light coat of gray primer paint. The second coat, applied the following evening (this is a big hint for painting enclosures!) was a coat of light avocado green. The following evening, when fully dry, the light blue trim and boxes for the transmitter drive and receiver RF gain controls were painted by hand using a small brush. The legends were applied using rub-off letters and sealed with a light coat of Krylon Protective Spray – available at many office supply or art stores. Clear enamel can also be used, but always test first on a scrap piece of material to ensure it doesn't "melt" the rub-off letters.

The front and rear panels were soldered to a copper clad "center" shelf, mounted about half the height of the two panels. This shelf serves to mount the Rockmite PCB and the paddles on the top (see **Figure 12**), and the transmitter components on the bottom.

The paddles are made entirely of pieces of copper clad, including the paddle pieces, as shown in the photograph of the top view. A 4-40 bolt and nut, with a spring from a BIC pen, formed the tension on the two paddles, while two other 4-40 machine screws serve as the dit and dah contactor and sets the spacing. It's not exactly a work of art, but they worked well, enough to have around 50 QSOs with this rig.

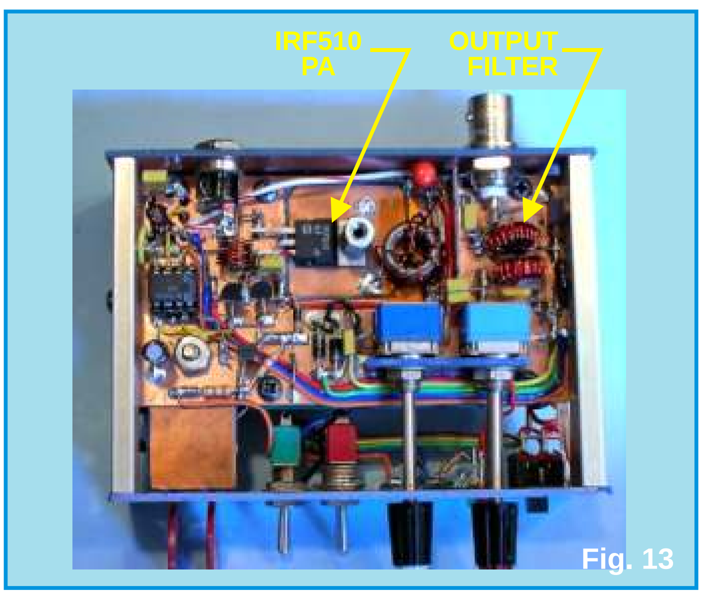


Top view of the Rockmite. The Rockmite PCB and the homebrew paddles are mounted on the top portion of the center shelf.

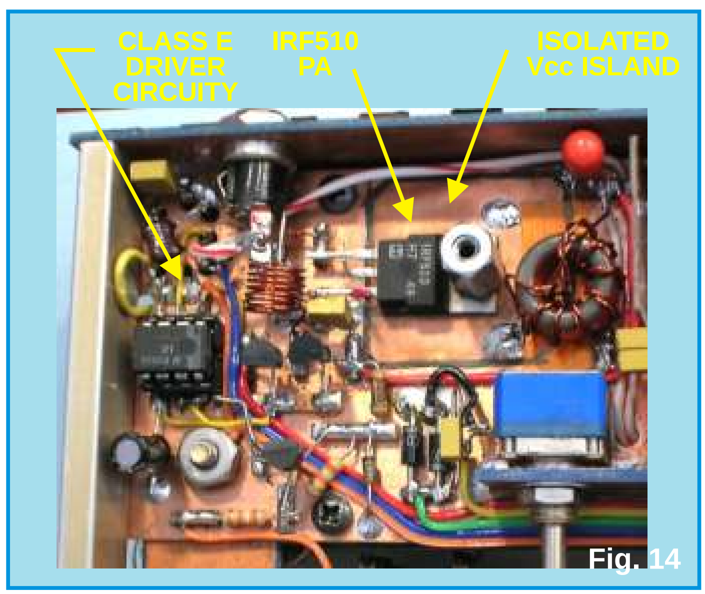
The transmitter was built Manhattan style, with the pads glued directly to the bottom of the copper clad shelf. The IRF510 was mounted to an island cut-out of the copper clad by a Dremel tool. Since the IRF510 tab is the drain, this isolated the +12v on the drain tab from ground. Most of the interconnecting wiring was performed by using flat ribbon cable as shown in **Figures 13 and 14**.

The intent of this particular custom-made Rockmite kit is to show the flexibility of copper clad. It was easy to form the copper pieces into the desired front and rear panels, the center shelf, and even the paddle pieces. Granted, it took a little cutting and filing to form some of the pieces, but far easier than forming the same pieces from aluminum or metal stock. Plus, it can all be easily soldered together.

By applying a light primer coat before the final color of spray paint, copper clad makes an attractive and durable front panel as well.



Bottom view of the Rockmite, showing the home-brew transmitter section built Manhattan Style.



A closer view of the transmitter section, showing the Manhattan style of construction.

MANHATTAN STYLE HOMEBREW TRANSCEIVER

I have built several QRP transceivers on different bands Manhattan style, mostly my own designs. In fact, that is one of the advantages I have found with Manhattan is how adaptable it is for a test platform. Changing components of different values to set proper biasing or gain is relatively easy, as is making circuit changes. Of course, too many circuit changes can get ugly as you try to fit things in you didn't originally plan on. But, it usually works fine. In the circuit shown here, I converted the RF amplifier from fixed gain to an AGC driven stage, moving around a few components from that originally planned.

The schematic (**Fig. 15**) is the "front end" portion of a 40M receiver I built, where T1 and T2 are Mouser 42IF124 IF cans. Q1 is a common base amplifier with the bias via R2 from the AGC line. C1 and C2 are the tuning capacitors to resonate T1 and T2 at the desired frequency (T1,T2 are 4.5uH nom. with no internal tuning capacitor). T1 is made resonant at the RF frequency and T2 at the IF frequency. Built Manhattan style, this RF amplifier and mixer scheme was fairly sensitive with a good noise figure.

The **Figure 16** photograph shows the "front end" portion of the receiver, based on the above schematic. With a little layout on paper first, the RF amplifier, receive mixer and 1st IF amplifier fits in an area about 1 x 2.5 inches. Interestingly, I also built a surface mount version of this same receiver, using the same IF transformers, and it took only about 1/4" less space! I used SOT-23 SMC 2N3904 transistors, which are about the same width as the TO-39 plastic versions. As a result, little space savings was noted – at least using this layout configuration.

The IF transformers are mounted on the main board in standard Manhattan style. See **Figure 17**. The only caution is to ensure the IF "can" is soldered to the main board with either the mounting tabs (if they reach) or with a piece of solid bus wire or a scrap resistor lead folded in two. Solder on two adjacent sides of the IF can for a firm mechanical connection.

Likewise, ground the desired pads(s) by soldering a wire or resistor lead to the main board for grounding. All transformer pins should go to a Manhattan pad to keep the IF transformer "level." Soldering the wire to ground the pad(s) to the main board also helps with the mechanical mounting without depending solely on the super glue. Otherwise, with a "stiff" IF can, you can twist the pads off the board while adjusting the center slug if not soldered directly to the board.

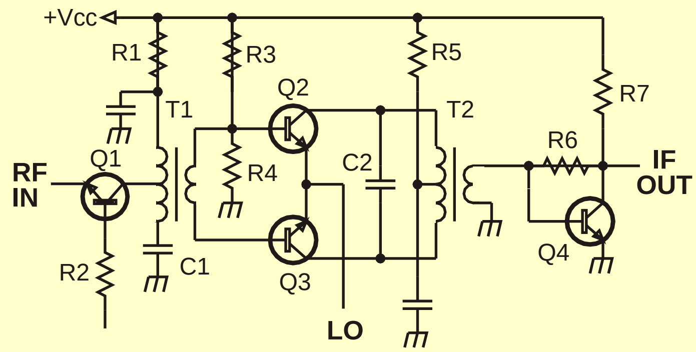


Fig. 15
Functional Schematic of the 40M "front–end"

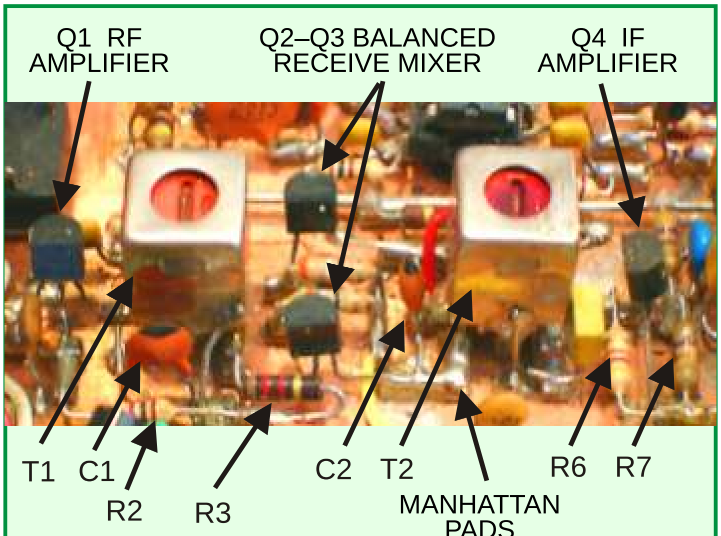
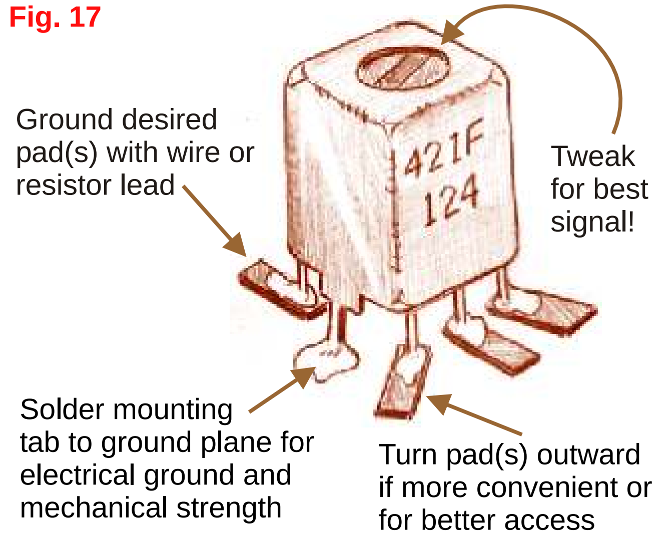


Fig. 16 – The "front-end" of a 40M QRP receiver built Manhattan style using IF "cans."

Fig. 17



Mounting IF Transformers Manhattan style

Following the 1st IF amplifier is the crystal ladder filter, as shown in **Figure 18**. Four of the crystals are for the IF filter, the one on the far left is actually the crystal for the transmit oscillator. The transistor in the upper left of the photograph is Q4, the 1st IF amplifier in **Figure 16** on the previous page. The wire soldered along the tops of the crystals serve two purposes: 1) to ground the cans to the main board, and 2) provide mechanical rigidity. Without this ground wire, I find myself constantly bending over the crystals while I'm building and poking around in the circuit.

Figure 18

The IF Crystal Filter

Figure 19 shows in a bit closer detail how the crystals and shunt capacitors are mounted to the Manhattan pads.

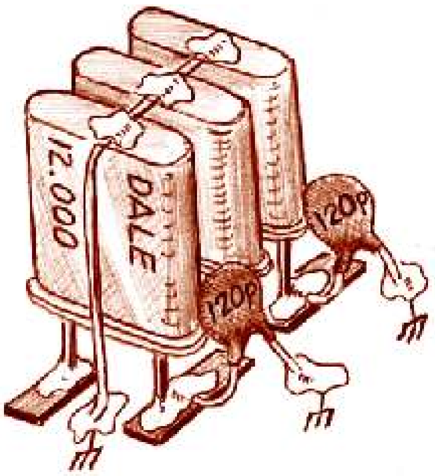


Figure 19

In mounting the crystals on standard Manhattan pads, the crytal leads need to be bent to fit. This is a case where cutting small strips of copper clad to length makes for a neater and more accessible assembly.

Figure 20 shows the LM386 audio output amplifier I.C.. This is such a simple, yet effective audio amplifier, it has become the benchmark amplifier in most QRP rigs.

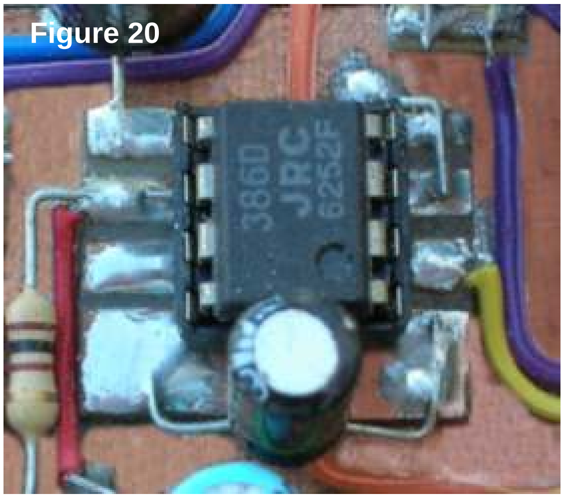


Figure 20

The LM386 Audio Output Amplifier IC

I mount ICs either on individual Manhattan pads, or build a single pad as shown in the photo of **Figure 20**. This pad is made from a single piece of copper clad, cut into the pads as shown by sawing away the copper between the pins with a hack saw, coping saw, or a Dremel tool with a cutting disk. Then, obviously, another cut down the length of the IC to separate pins 1–4 from 5–8. Either method takes about the same amount of time, though the single Manhattan IC pad does look nicer, in my opinion.

If you make a rig out of copper clad, including the front and rear panels, don't forget the copper on these surfaces can be used as well. **Figure 21** shows one rig I built with the PA output filter mounted on the inside of the rear panel, next to the Antenna BNC connector. In this particular case, I etched away the unwanted copper with a Dremel tool, though Manhattan pads could just as easily be used. To the left of the filter (not shown) is the TO-220 PA transistor – also mounted on the inside rear panel. This allows the rear panel to serve as a large heat sink.

Figure 21

The PA Output Filter mountedon the inside of the rear panel saves space

A MANHATTAN BUILDING JIG

One of the difficulties I've experienced building small circuits is the copper clad board, weighing only a couple of ounces, moves all over the workbench surface as you work on it.

Shown in the photograph is a Manhattan Building Jig (MBJ) I built for holding down a circuit board while it is being built and tested. In this case, I used a piece of aluminum and milled out several slots. In these slots ride the screw heads for the threaded standoffs. On the top of the standoffs, the washers and nuts secure the circuit board. Once attached, the screws on the bottom of the plate are tightened to hold everything rigidly in place. This allows different sizes of circuit boards to be mounted onto the jig. I have found this simple jig to really ease construction and testing. Particularly testing. Once you get a couple of cables and wires connected to the board, the weight of the cables alone will pull the board right off the bench! A jig with a little weight and larger footprint will keep this from happening.

Figure 22

A building jig for holding down the circuit board for Manhattan style of construction.

On the far right hand side of the jig, under the circuit board and hardly noticeable, is the TO-220 voltage regulator used for the circuit. This places the voltage regulator close to the circuit and the base acts as a heat sink.

Of course, a jig of this nature could be built out of plywood or even a piece of 2x4. In this case, the board is held down to the jig with wood screws or other fastening scheme.

MANHATTAN – VHF STYLE

One of the "modules" I am responsible for at the VLA observatory is called the "4/P Converter." This converts our low-band receivers, being 74, 196 and 308–348 MHz, to an IF of about 1.1–1.4 GHz (L-band), then upconverted again to our 8–12 GHz X-band IF. In order to checkout this upconverter, I would need 4 signal generators, one for the three receivers and one for the 1024 MHz LO. I'd get killed by my co-workers for sucking up 4 of the lab signal generators everytime I needed to work on this converter! And, I've got 28 more of them to build over the next 3 years. So, I designed and built a test set that simulates the three receivers and the 1024 MHz LO. Additionally, it contains a sweep generator for "sweeping" the bandpass shape of the RF and IF filters on a spectrum analyzer.

Fig. 23 – A Manhattan-built Test Set used to simulate three VHF receivers and contains a 1024 MHz Local Oscillator

This was a fun "ham radio" project at work. It was built largely from copper clad and Manhattan style. Most of the parts were ordered from Mouser, All Electronics, Electronics Goldmine and MiniCircuits. The overall "4/P Band Test Set" is shown in **Figure 23**. The copper clad circuit assemblies are mounted vertically with the push-button switches and potentiometer controls protruding through the front panel. Details of two of these assemblies, the 74MHz oscillator and the 200-400MHz sweep oscillator, are shown on the next page. The meters indicate the output power level, normally set to –35dBm to simulate the receivers. While this is not a ham radio QRP project, it does contain many construction techniques that can be applied to any HF or VHF project. (Although, it *was* built by a QRPer!). When I built this, I was a bit concerned at how the copper clad and Manhattan pads would behave at the VHF frequencies..It turns out, it works quite well. Wideband sweeps reveal only minor gain "suck-outs" between 600-1500 MHz. Rumors that Manhattan style should not be used above about 20MHz are thus unfounded, as proven with this project.

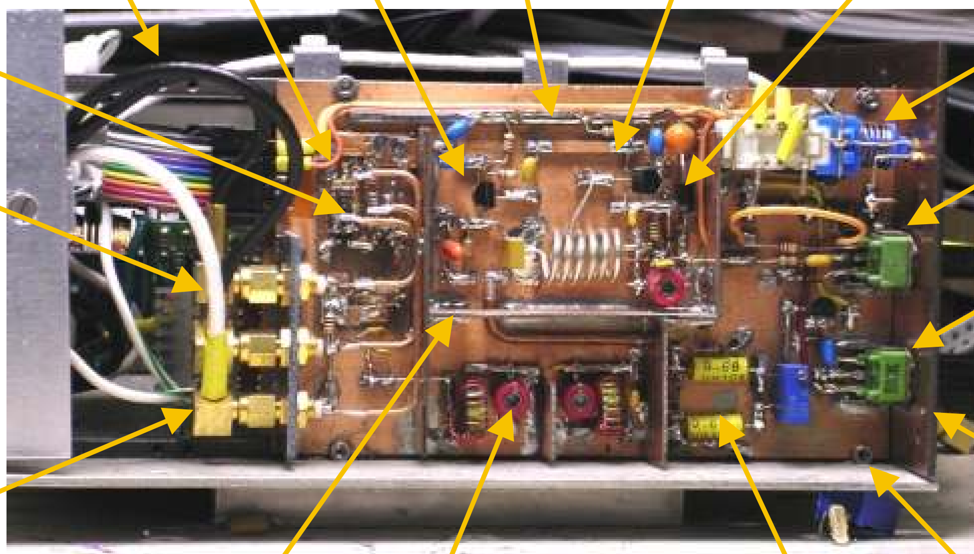


FIG. 24 — 74MHz OSCILLATOR (74MHz RECEIVER SIMULATOR)

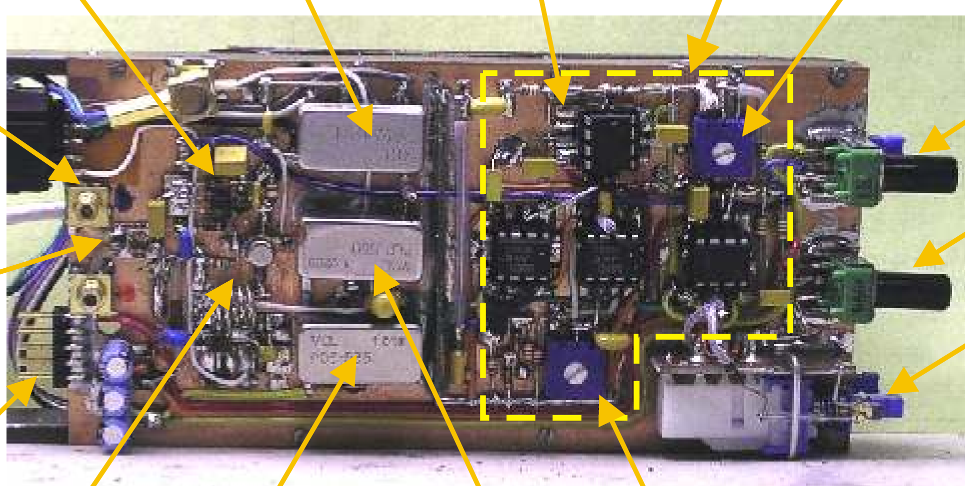


FIG. 25 — 200–400MHz SWEEP OSCILATOR (P-BAND RECEIVER SIMULATOR)

3. "Ugly" or Manhattan?

While the majority of this article focuses on Manhattan style of construction, it is not the only means to build a circuit. Since the dawn of radio, hams have built equipment "ugly style." Ugly has a charm of it's own.

Ugly began in the earliest days of vacuum tubes, where a circuit was built on a piece of smooth wood, mounting components between nails or screws – often just twisting the wires together, not soldered. Since a cheap piece of attractive wood in the early 1900s was a breadboard used by bakers, the term for this style of construction was called "breadboarding" – the genesis of the term still used today for building a one-of-a-kind circuit. Building on a breadboard could be anything from beautiful (**Fig. 26**)– to outright ugly.

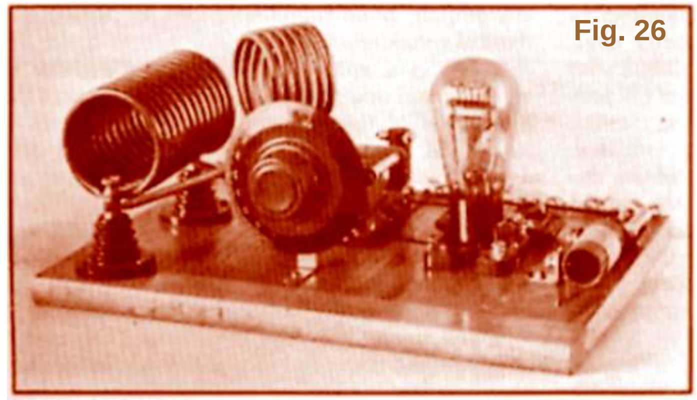
Basically, building something ugly means "just throw it together" with little regard to appearance. Ugly also tends to imply building it cheaply as well, a common attribute of most hams – yesterday as well as today.

Today's breadboard tends to be a piece of copper clad. Component leads that are grounded are soldered to the copper clad surface as in Manhattan style. Everything else just gets soldered together, often with the components hanging in mid–air. The only concern is to make sure the component leads to do not touch ground or other things they shouldn't – often by bending or routing leads and wiring in a precarious manner. An example of this is the 7 MHz VFO built "ugly" as shown in the photograph of **Figure 27**.

A variation of the ugly circuit is called "dead bug." This technique is where the integrated circuits (the "bugs") are glued (or not) to a surface, face down, with the IC pins sticking up in the air for easy access. Wiring and components are soldered directly to these pins.

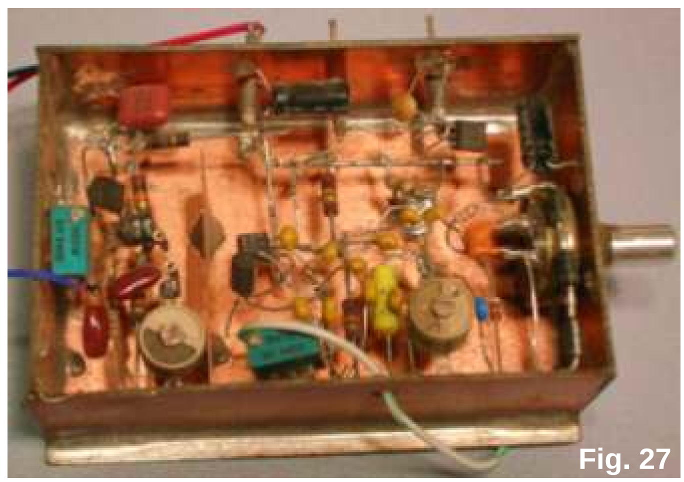
Regardless of the ugly method used, the circuits usually perform quite well. The biggest problem is stray capacitance from the often long component leads and wiring hanging in mid-air and in close proximity to each other. However, once the circuit is "tuned" to account for the stray capacitance, the circuit will work reliably – as long as you don't move or rearrange anything!

This becomes one of the biggest problem in ugly construction – duplicating the circuit. It often works fine for the first person building it, but when the circuit is built by someone else, results may vary. This is why early QRP publications seldom detailed how the circuit was built, as it was difficult to document to show exactly how the circuit was built.

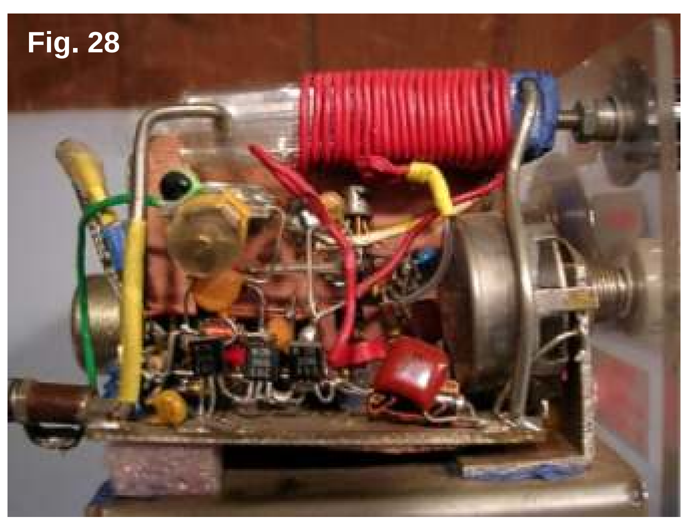


From 1933 ARRL Radio Amateur's Handbook

Early ham equipment was often built on a standard 10 x 12.5 inch bread board, such as this 1930s "7000 kc low-power transmitter."



The modern "breadboard" is often a piece of copper clad. This 7.0 MHz VFO was built "ugly style" in a copper clad "box."

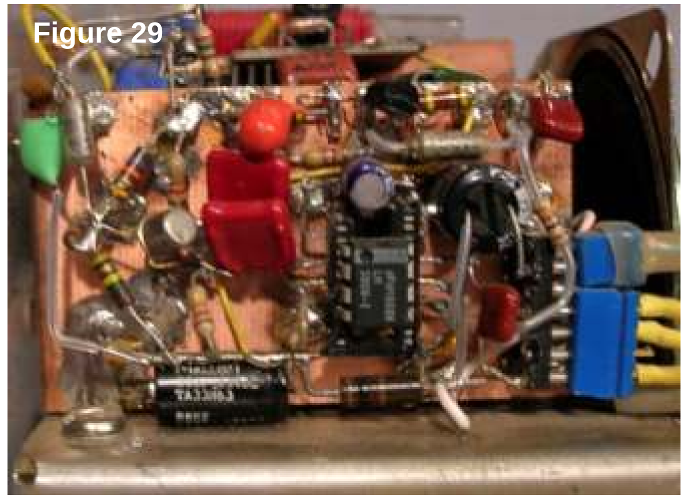


A regenerative receiver built ugly style on a piece of copper clad. The coil is wound on an IC shipping tube.

In my opinion, this is the strongest advantage of Manhattan style for QRPers ... it is easy to document. Photographs or drawings illustrate exactly where each component goes and how it is built, ensuring consistency in construction amongst the various builders. This consistency also ensures the performance of the circuit will be about the same from unit-to-unit. This is why those people building Jim Kortge K8IQY's 2N2/40 were so satisfied with the results. Those who built it from the detailed drawings in the original QRPp article, the book, or on Jim's website, all ended up with a hot 40M transceiver with very similar performance to Jim's original. Had the 2N2/40s been built "ugly," this consistency in performance could not have been guaranteed. How would you document with any degree of accuracy the "ugly" circuit shown in **Figure 29**?

This is why Manhattan style has become so popular with QRP homebrewers. The designer can *clearly* illustrate *exactly* how to build the circuit to guarantee the expected results. The builder has *precise instructions* to follow and can build the circuit with the confidence it will work. This is true with the seasoned builder as well as the beginner. This is why Manhattan style has become the biggest boost to building a circuit "from scratch" by QRPers. Circuits designed and built Manhattan become excellent construction articles, since the step-by-step instructions lie mostly in the illustrations or photographs.

This is not to say building a circuit "ugly" style is inferior. As already mentioned, problems can occur in attempting to duplicate the circuit. However, for building a one-of-a-kind circuit, ugly can be a quick, cheap, dirty way to get it built and get it on the air. Over the years, I have had many QSOs with homebrew rigs built ugly. A couple were really ugly! The classic "Ugly Weekender" 40M receiver by Wes Hayward W7ZOI and Roger Hayward KA7EXM is a good example of a very nice performing rig built ugly style. It was featured in the 1992 Radio Amateur's Handbook and in the ARRL's book "QRP Power."



The AM detector and audio amplifier portion of a shortwave receiver built "ugly" style. The unused end-pins of the IC socket are soldered to the copper clad. The remaining socket pins are bent outward to make the connections.

There are few rules in building ugly. You simply "do your own thing" and get it working.

4. Conclusion

As most homebrewer's will tell you, there is nothing like the feeling of building a QRP rig and the thrill of having that first QSO with it. Whether you build ugly, a kit, or Manhattan style, QRPers will always be building their own equipment. This is why some of the QRP clubs and various vendors provide kits for building your own QRP transceiver. And, for those wishing to build a rig from scratch, this is why the QRP journals like the ***Homebrewer*** present as many construction articles as they can on the subject of homebrewing.

This article is intended for both the experienced builder and the new comer. If you've never built anything from scratch before, build a simple circuit using these techniques to "get your feet wet." AmQRP is committed to homebrewing. There will continue to be construction projects of different skill levels in future issues of the ***Homebrewer***.

In Part 2 **–** we'll continue with some of the construction practices employed in building circuits from scratch, including an emphasis on building with surface mount components, some various "hints and kinks," and a photo gallery of what others have built.

I am not a master builder of Manhattan. I never dreamed some of the stuff I've built would be featured in an article – or else I would have built them a little nicer! If you've built something from scratch, ugly or Manhattan, feel free to send me a photo or two to include in Part 2 to show what others have built, and how they built them. Likewise, if you have a construction hint or kink, send it to me and I'll gladly illustrate it for the next issue.

72, Paul Harden, NA5N
na5n@zianet.com

Paul Harden, NA5N • na5n@zianet.com

Part 2

The Handyman's Guide to – HOMEBREW CONSTRUCTION PRACTICES

Surface Mount — Manhattan Style

Prepared for and published in AmQRP's Homebrewer #7 (June 2006)

*In **Homebrewer #6**, Part 1 looked at techniques for "building from scratch," based largely on thru-hole components and building circuits "ugly" to Manhattan style of construction.*

*Part 2 focuses primarily on techniques for homebrewing with **surface mount components** (SMC), for which the least amount of documentation exists. Unlike thru-hole components, SMC is not well suited for ugly-style of construction. A variation of Manhattan style is shown that makes building from scratch using SMC a viable approach. Even if you build an SMC circuit from a kit, you might find some of the following information useful.*

1. A Quick Review . . .

MANHATTAN STYLE . . . What is it?

Simply put, ***Manhattan Style*** of construction uses small pieces of copper clad (the "pads") glued to a main copper clad circuit board (the "substrate") that serve as component mounting platforms. The electronic components are then mounted and soldered onto these pads. The main "substrate" board serves as the ground plane. Not only is this technique an easy and neat way to build a circuit, it also produces a very quiet circuit due to the solid ground plane.

Making the "pads." One popular and easy method for making the pads is using a ***nibbling tool*** to *nibble* out small pieces of copper clad from a larger piece, as shown in **Fig. 1**. Others use round pads from a punch or cut the pads out of the main board with a Dremel tool and a small cutting disc.

Building the circuit. The pads are then glued onto the main board for mounting the components. Super glue is usually used for affixing the pads to the main board. The pads are positioned more-or-less in circuit order, similar to laying out a printed circuit board (PCB). A little forethought of layout goes a long ways. After the layout is decided, it is best to "build as you go along" ... that is, glue down a few pads, solder the components, then move to the next few pads, to keep from working yourself into a corner or running out of room.

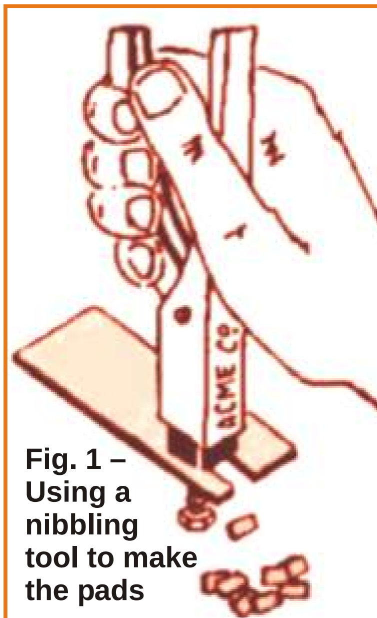


Fig. 1 – Using a nibbling tool to make the pads

FIG. 2 – The famous "Iowa 10," a 10M QRP rig designed and built by Mike Fitzgibbon, NØMF. It is an excellent example of building a rig from scratch using thru-hole components and Manhattan Style of construction. Note the use of "vertical" boards - one contains the VFO and receiver, the 2nd board the transmitter circuitry.

2. Techniques for Surface Mount Components (SMC)

With thru-hole components, there is great flexibility in how one builds a circuit. With surface mount, there are few options. Using printed circuit boards intended for prototyping SMC circuits are expensive. Trying to build "ugly" or "dead bug" is nearly impractical. It turns out, Manhattan style can be a nice way to prototype or build circuits using SMCs.

For SMC MANHATTAN CONSTRUCTION, I recommend using the thinner .031" thick copper clad for the pads. The main board "substrate" can be either .031" or .062" as desired.

With thru-hole parts, you have long leads to fit components between the pads. With SMC, you don't have this luxury. The small SMC parts must be mounted between two very closely spaced pads, requiring a host of jumpers or hookup wires to connect to the other component pads in the circuit. This additional interconnecting wiring gets very tedious (and ugly).

Therefore, in addition to using the .031" copper clad for the ***pads***, ***I recommend using the .031" for forming circuit strips*** as shown in **Fig. 3**. The strips can be cut on a PCB shear, paper cutter, or with scissors. They should be cut to about the width of a nibbled pad (0.065"), or other widths as desired. When cutting the strips with scissors, they will have a tendency to "curl." However, they are easily straightened.

The strips are used for interconnecting the circuits by cutting to length, and positioning as if forming traces on a PCB. **Fig 4**. shows the strip technique for mounting several MiniCircuit SMC devices, including an ADE-35MH, identical to the ADE-1 mixer, and an ERA amplifier, both used in many QRP circuits. The SMC caps and resistors are size 0805.

I have developed this technique from building numerous surface mount "Manhattan" circuits, both at home (QRP) and at work over the past couple of years. The combination of using .031" copper clad and the strips works quite well ... *up to around 1GHz* ... a technique I'll refer to as ***Manhattan-031***.

Fig. 3 – The basics for *Manhattan-031*. Strips of .031 copper clad are cut into narrow strips with a PCB shear or regular scissors; pads were made with a nibbling tool; super glue is used to affix the pads and strips to the main copper clad board.

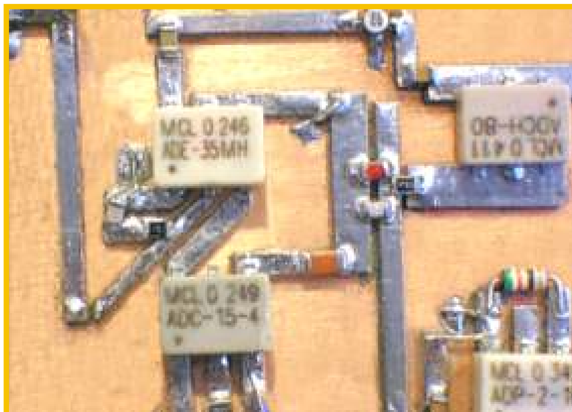


Fig. 4 –The strips form "traces" similar to a PCB for orderly mounting of the SM components. Often, diagonal runs will be more convenient, as shown here.

TOOLS TO USE

Fig. 5 shows the tools you should have for surface mount construction. **Tweezers** are used for positioning the smaller components (such as 1206 or 0805 sized resistors) into place. **Hemostats** are often easier to manipulate than small pliers for positioning and holding odd shaped components (such as SOT-23 packages or round components). A small **screw driver** can be used as a hold-down device, particularly for gluing the pads, islands or strips into place. **Q-tip** shafts can also be used as a hold-down device – and the cotton swab end for cleaning. Q-tips with the soft paper or plastic shafts do not work as well as those with the wooden shafts.

Fig. 5 – Tools of the trade: hemostats, tweezers, small screw driver and Q-tips.

SOLDERING (”PICK AND PLACE” METHOD)

Soldering surface mount components to the pads is relatively easy. The various exotic schemes of building hold-down devices and jigs is not necessary and actually quite time consuming to use.

Fig. 6 illustrates the simple steps of soldering a surface mount component, such as a 1206/0805 resistor or capacitor. Lightly solder *one* of the pads. Position the component into place with tweezers and hold in place. Heat the pre-soldered pad until the *solder flows onto the component*. It does not have to be a good solder joint at this point – just enough to hold the component in place. Remove the tweezers or hold-down device and solder the remaining pad. This should be a good, proper solder joint, though be frugal with the solder (no big blobs or excessive heat). Return to the first pad and apply a bit more solder if required. It is helpful to perform this assembly and soldering with plenty of light and under some sort of magnifying lens (particularly if you are older than say, 50!).

Do not make contact with the component when soldering – heat only the pad and solder, letting it “flow” to the component.

Do not pre-solder both pads. This will cause the component to mount at an angle, or noticeable tilt. When soldering the second pad, there is a tendency to push down on the component with the hold-down device or soldering iron to make it lay flat between the two pads. With the first pad already soldered and cold, applying pressure to make the part lay flat can crack or break the component.

With a little practice, this “pick and place” method becomes quite easy and quick. It’s how the professionals do it.

Fig. 6 – How to solder surface mount parts.

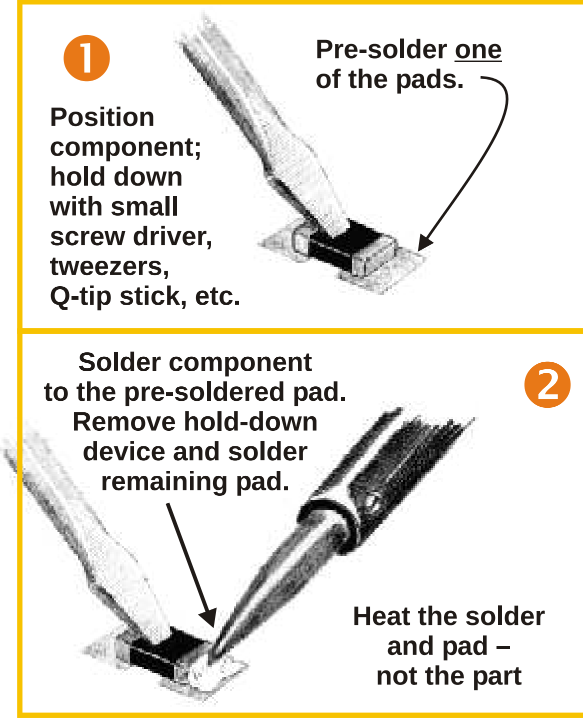


Fig. 7 – A Quick Primer on 60/40 Solder

60/40 is the standard rosin-core solder used by most amateurs and QRPers.

60/40 melts at 370°F (188°C). This is the ***plastic state*** and makes a poor connection, dull in appearance. Also called a “cold” joint.

60/40 liquifies at 500°F (260°C). This is the ideal soldering state, allowing the solder to easily “flow” to the component. Leaves a smooth, shiny connection.

60/40 rosin (flux) **vaporizes** around 700°F (370°C), leaving a dull, uneven connection.

THE SOLDERING IRON and SOLDERING TEMPERATURE

The soldering iron should have a fairly sharp tip (See **Fig. 6–2**). Keep it clean and well tinned. SM components should not be exposed to excessive heat, both in terms of temperature (>800°F) or duration (>5 seconds) of direct heating. *Temperatures in the **500°–600°F (260°C–320°C)** range are recommended*. Below 500°F, it takes longer to melt the solder, heating the SMC part longer than 5 seconds. Above 800°F can damage the part, plus, all the flux is boiled off, leaving a dull solder finish. Note that these temperatures are those recommended by the manufacturers of the standard 60/40 solder, as shown in **Fig. 7**.

“But, I don’t know what temperature my soldering iron is at!” If your soldering iron takes several seconds to melt the solder and it dries in a lumpy, dull finish, it is not hot enough (<500°F). If it quickly melts the solder, but dries in a dull finish, it is running too hot (>700°F). If the solder quickly melts, flows rather smoothly, and cools with a “shine,” the temperature is correct.

Use as little solder and heat as possible for SMC. Excessive heat can alter the value of the component, cause the metal solder tab to separate from the component, or cause the body to crack. Remember, a large solder blob will fuel heat into the component long after you remove the soldering iron.

THOSE EXOTIC SMC SOLDERS

There are various solders developed for SMC, such as special low melt solder, organic rosins, 2-10% silver, and no lead (Pb) varieties. These are becoming more popular in industry due to environmental and health concerns. However, they are also expensive. The standard 60/40 resin-core solder still works great for most all soldering needs, including SMC, following the temperature guidelines in **Fig 7**. To reduce health hazards and eye-irritation, solder in a well ventilated area. Of course, the choice to use organic rosins or no-lead solders for hobby use is a personal one.

AMATEUR vs. PROFESSIONAL METHODS

Believe it or not, there is not much difference in how SMC circuits are built by the homebrewer vs. the professionals. For large scale operations, such as building hundreds or thousands of cell phone boards, machines are used for placing and re-flow soldering the components. For building dozens of the same board, hand assembly is still more cost effective. At the Very Large Array (VLA) radio telescope (where I work), we are completely replacing the original 1970s electronics in the antennas with newly designed electronics. Most all circuits are now surface mount, which are built "in-house" by several professional assemblers, such as Connie Angel shown in **Fig 8**. She is shown building an SMC board by hand – using good old tweezers and a soldering iron. The only advantage she has is the assembly station microscope, especially useful for mounting those 0603 or 0402 parts!

CLEAN–UP

After soldering several components, whether Manhattan-031 or a PCB, clean the solder, components and pads with a Q-tip or hobby brush moistened with rubbing alcohol as shown in **Fig. 9**. This will remove the excess rosin (the brown stuff) before it has time to solidify for a clean, shiny solder job. Removing the flux, even organic, can prevent corrosion problems months or years later. At the observatory, (as at other facilities) final cleaning is done in an ultrasonic sink using 91% alcohol as the solvent. Critical boards are then checked for loose components on a "shaker table." Dropping your project on the floor a time or two serves the same purpose!

I have cleaned my Manhattan-031 projects in the ultrasonic cleaner with no problems with the pads vibrating loose. This is only a luxury if you have access to an ultrasonic sink.

ASSEMBLY JIG

One building problem is a small board has a tendency to "walk around" on the work bench as you attempt to solder and built it. On a printed circuit board, this can scratch and nick the circuit (bottom) side of the board. **Fig. 10** shows an assembly jig I made for building prototype boards. I have used this both at work and at home. It is a simple aluminum base with 4-40 (or 6-32) threaded stand-offs for mounting the board being built. The standoffs are mounted to the base along slots for adjusting to different board sizes. Small rubber feet assist in keeping it from sliding around on the work bench.

Fig. 8 – Connie Angel, one of the professional assemblers at the VLA observatory, solders surface mount components onto a board with tweezers and a soldering iron. The board is part of a new radio telescope (called ALMA) being built in Chile.

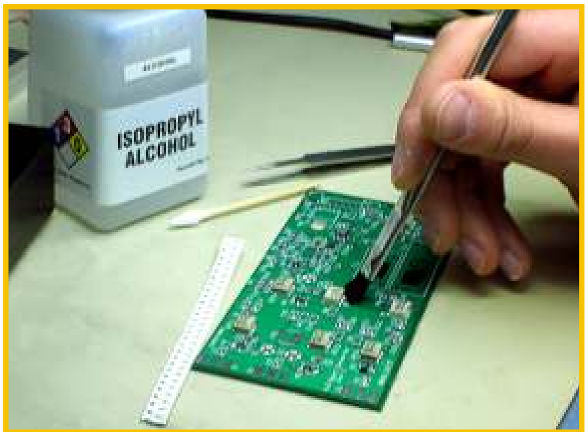


Fig. 9 – Clean solder connections with a hobby brush and alcohol. This keeps the board clean and free of corrosive rosin, whether building Manhattan or a PCB.

Fig. 10 – An assembly jig for holding the board being built can be made from an aluminum plate or piece of wood. It gives the board some weight and stability during construction and testing.

A wood base can also be used for the assembly jig as well. It is simply to give it some weight, and to keep the board from sliding around. The jig is also helpful when oscilloscope, power and signal generator leads are connected to the circuit during testing.

3. Manhattan-031 — Let's Build Something SMC

AN ERA-1 SMC RF AMPLIFIER

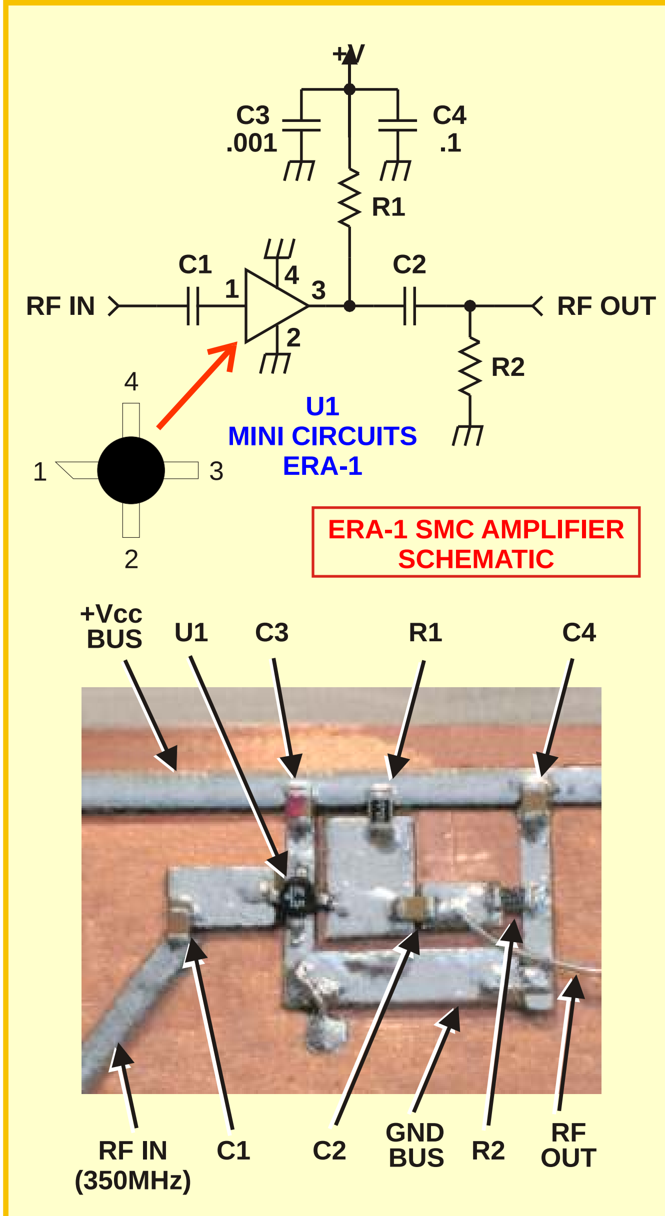
The schematic diagram of a very basic wideband amplifier, using the MiniCircuits ERA series of amplifiers, is shown in **Fig. 11**. The circuit was built entirely of SMC components, using 0805 resistors and capacitors Manhattan-031 style. The ERA is a series of surface mount amplifiers commonly called a MMIC device, short for "Monolithic Millimeter-wave Integrated Circuit," often pronounced "mimmic."

The value of R1 is determined from the data sheets, which establishes the bias and operating current for a given +Vcc. For the +8v used in this circuit, the value of R1 is 53.7Ω., a 1% precision resistor value. A 5% 51Ω standard value could also be used.

Two circuits were built Manhattan style. The first (not shown) used .062" pads cut from a nibbling tool, #22 hookup wire for the +8v, and #26 wire for interconnecting the pads. Analysis of this circuit on a scalar network analyzer (HP 8657) revealed reduced gain in the 10-500 MHz range due to oscillations in the 4-6 GHz region. An 18dB gain "suck-out" around 200MHz was also present. The circuit was rebuilt as shown to the right, using .031" copper clad for the pads, islands and strips to replace the hookup wiring and the +8Vcc bus. The larger "islands" on the input and output terminals of the ERA (ERA-5 "E5" used in the version in the photograph) added sufficient inductance to prevent interactive feedback. The circuit was used for amplifying a 60–400MHz signal. The gain sweep of this circuit was much flatter, due to removing the inductance and resonances caused by the stranded hookup wire at the higher frequencies. This method proved to be a stable circuit at both HF and VHF (no oscillations or major gain suck-outs).

These ERA MMIC amplifiers are wideband, from DC-4GHz. Even if the circuit is used for a mere 10 MHz, the wideband nature can cause oscillations at VHF and degrade HF circuit performance. The Manhattan-031 style of using islands and strips should still be employed at the lower frequencies (<30MHz). If the gain of the amplifier circuit is several dB below what the device is rated for, chances are, you have an oscillation at a very high frequency.

Fig. 11 – A surface mount RF amplifier using a MiniCircuits ERA-1 MMIC amplifier.



The pads, islands and strips were cut from .031" copper clad. The main board, in this case, was also .031, although either .031" or .062" could be used. The pads and strips were affixed with super glue.

0805 resistors and caps were used. It is always advisable to check your soldering through a magnifying glass to ensure you have a good solder bond to all SMC components.

The circuits that follow are part of a low frequency (74-350 MHz) upconverter I was assigned to design for the VLA. The prototype was built using Manhattan-031 to test the circuit and differences between components (such as the mixers and power splitters for phase stability). Once the prototype was finished and tested, it became the model for laying out the 4-layer PCB for the project. This upconverter is now in several VLA antennas doing science.

LO AMP AND MIXER

Fig. 12 is the circuit for a 1024MHz LO amplifier and mixer. This scheme could be used for any HF or VHF frequency.

The LO amplifier is a MiniCircuits GALI-51 in a SOT-23 package. It is shown here since the SOT-23 is also used for SMC transistors, such as the 2N3904, and would be mounted identical to U1 in **Fig. 13**.

The mixer is a MiniCircuits ADE-35MH – identical to the popular $1 ADE-1 mixer used in some QRP kits and homebrew circuits. It is easy to mount Manhattan-031 style as shown. Pins 1, 4 and 5 are grounded via .031 strips, rather than bending the small pins for soldering to the ground plane. This circuit is one way to drive an ADE-1 50Ω DBM mixer with the required +7dBm (or +13dBm) LO power.

The Z1 2dB pad is the same package used by the ERA series of amplifiers. In this case, the ground pins are soldered to the main board as they are for the ERA-1.

This circuit up-converts the outputs from 74, 196 and 327 MHz receivers to 1.1–1.5GHz L-band IF with a 1024 MHz LO. Therefore, it should work on 20M!

FIG. 12 – A surface mount LO amplifier and mixer

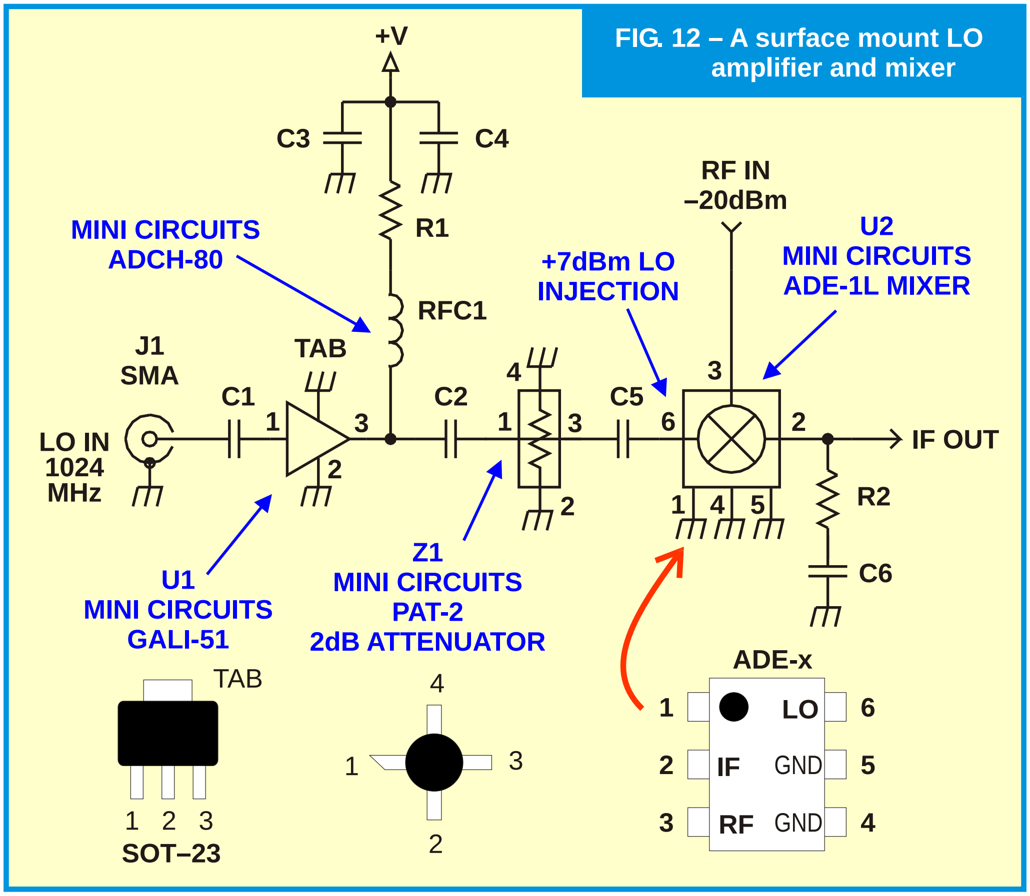
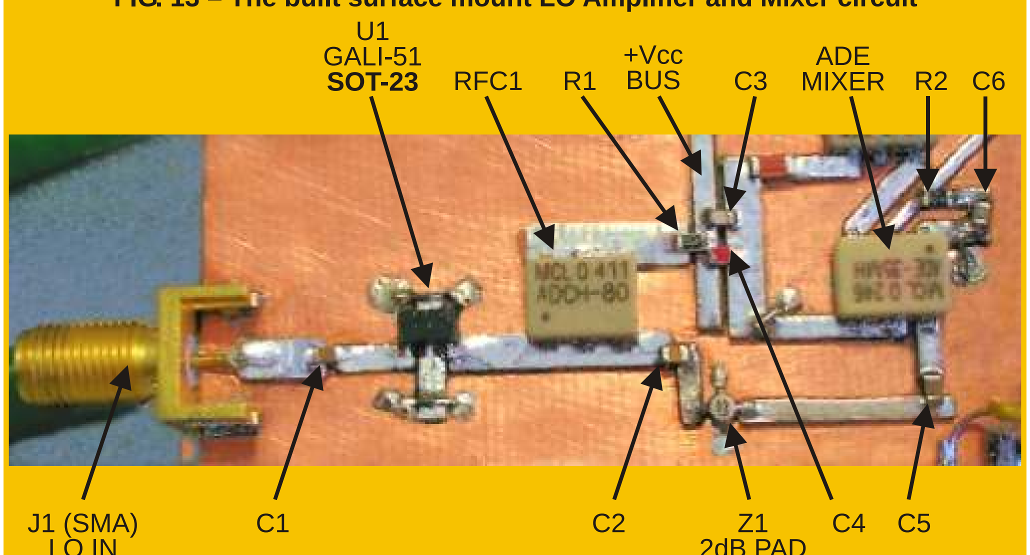


FIG. 13 – The built surface mount LO Amplifier and Mixer circuit



A SURFACE MOUNT POWER DETECTOR

This is another circuit built at work to test out the new (at the time) Analog Devices power detector IC, the AD8361. This would make an excellent QRP project as either a bench power meter (–30 to 0dBm), or with an external attenuator, a power/SWR meter. Except for one thing. The AD8361 is an 8-pin IC in the Micro-SOIC package, or about ten times larger than a grain of salt. It's difficult enough soldering it to a PCB, but I about died trying to build it ugly/Manhattan style, as shown in **Fig. 15**.

While the purpose of this article is construction techniques, a little description of the circuit may be appreciated by some.

The AD8361 produces about a 0-2v output voltage related to the input power. U2 AD8031 op amp amplifies this voltage by about 2.5 to drive a 0–5v analog to digital converter. The gain of the non-inverting op amp is determined by the ratio of RV1+R2 to R1. RV1 is thus included to precisely set an output voltage to correspond to a certain input power (in this case, +4v=–20dBm).

I built the prototype of the circuit as shown in **Fig. 15**. First, I covered the copper clad with a dielectric tape (the reddish stuff). Most of the "pads" around the AD8361 are actually strips of copper tape, requiring insulation from the main board ground. I did this because of the extremely small size of the AD8361, making pads that small impossible. Note the input wire labeled "RF IN." This is a piece of fairly small #30 solid wire wrap wire. Yet, it is about the same diameter as the pins on the Micro-SOIC AD8361. Anything larger could not be soldered to the pins. I had to solder the connections to U1 using our surface mount station at work under the microscope. The magnifying lens I normally use was not sufficient to see what I was doing while soldering.

Building the circuit with the Micro-SOIC chip was fun – *once*. I doubt I will try it again!

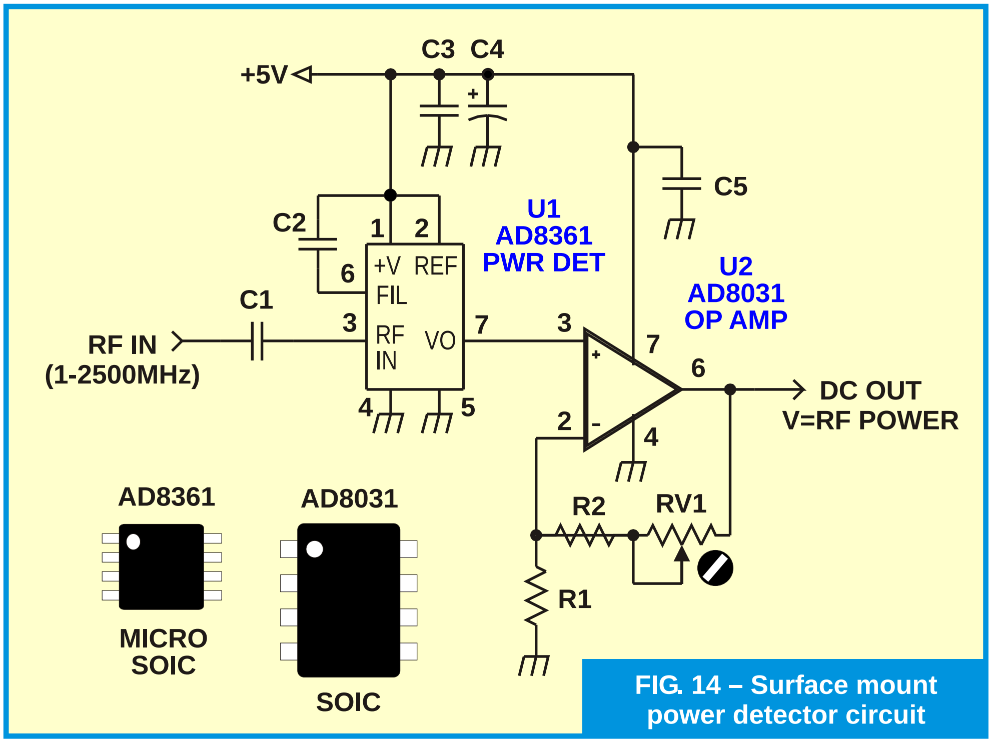


FIG. 14 – Surface mount power detector circuit

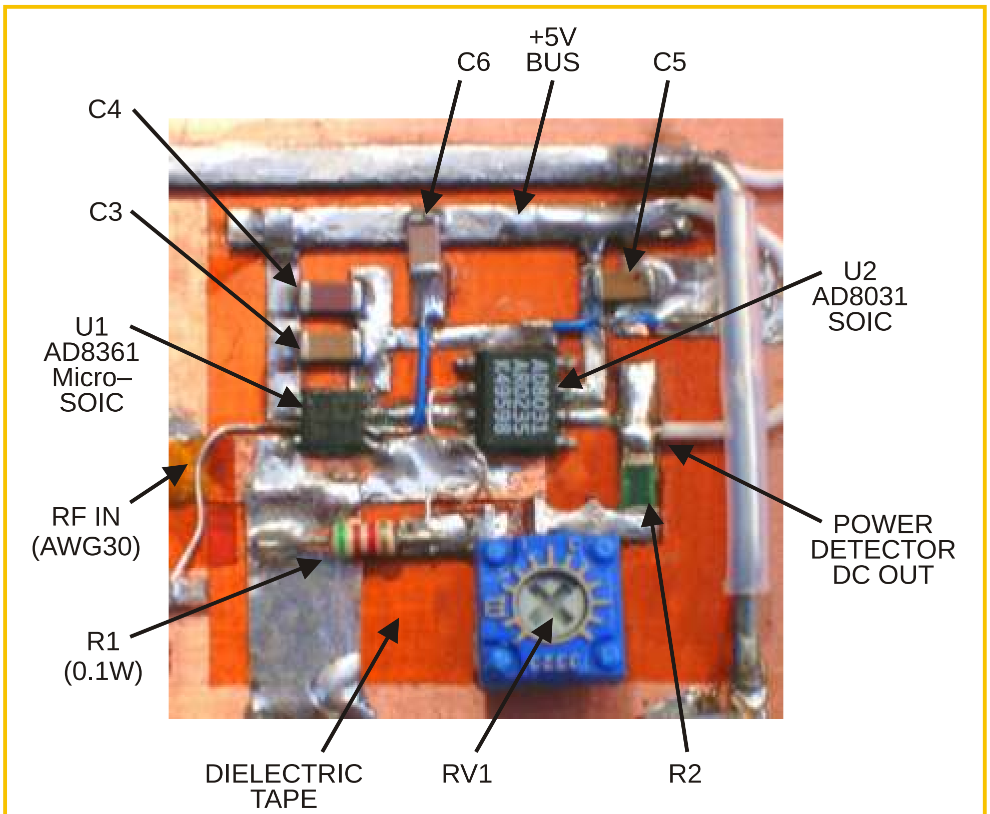


FIG. 15 – THE Prototype Build of the SMC AD8361 Power Detector

The Micro-SOIC package is just too small to properly build on a prototype level without a microscope. The other surface mount IC, the U2 AD8031, in the "normal" SOIC package, was relatively painless.

The rest of the circuit was built in a fairly straight-forward way – mounting the SMC components between .031" Manhattan pads.

4. Some More SMC Applications

A SMC "ADD ON CIRCUIT"

One advantage of the small surface mount components and building a circuit Manhattan style is if a small circuit is needed to modify an existing circuit or a PCB. This was the situation I found myself in with the upconverter project. After the PCB version was built, it was not working properly – there was hardly any image rejection. The problem turned out to be the commercial bandpass filters were incorrectly built by the filter manufacturer (the 3dB point somehow ended up being about 800 MHz instead of 1090 MHz as ordered). This was critical, since the LO was 1024 MHz. Therefore, 400 MHz of image was blowing through the filter below 1090 MHz.

FIG. 16 – A 1090 MHz High Pass Filter made from SMC capacitors and "hand made" inductors Manhattan-031 style.

The manufacturer agreed to replace the filters, but the turn-around time was estimated at 12 to 16 weeks. We couldn't afford to have several VLA antennas down for that long. I decided to try and build my own 1090 MHz high pass filters (HPF) and add them on to the PCB ahead of the bad filter on the board until the commercial filters arrived.

The final 1090 MHz HPF is shown in **Fig. 16**. The board is about 0.5 x 0.75 inches. This is pushing what one can do with "lumped elements," that is, with discrete components. The values to establish the –3dB frequency of 1090 MHz was C=1.7 and 2.2pF, and the inductors about 4nH (.004uH). If nothing else, this project verified that .031" Manhattan pads have about 1.0pF of capacitance to ground! This value had to be subtracted from the capacitance needed to select the capacitors. The coils were wound as shown in the photographs, and squeezed and twisted to tune the filter. **Figure 14** shows the board added to the bottom of the PCB. The filter did knock down the unwanted images about -25dB, or just barely enough for the upconverter to be used. 16-weeks later, the commercial filters arrived, providing –60dB of image rejection.

FIG. 17 – The temporary 1090 MHz High Pass Filter added to the PCB.

The point of this is how a small SMC Manhattan style board can be used to build a small "low profile" circuit board for modifying a PCB. Published mods to a QRP kit, or your own modification, can be easily implemented in this manner. Adding a CW filter, keyer, or an AGC are other examples of circuits that can be built with SMC and added to an existing rig or kit, requiring a minimum of space.

HYBRID CIRCUITS

For those contemplating building something with SMC Manhattan-031 style, it is unlikely one would build a QRP rig or project "from scratch" entirely surface mount the first time. Most of us build something from as many junk box items as possible. Therefore, it would likely be a combination of thru-hole and SMC. In industry terms, this is called a "hybrid circuit." **Fig. 18** shows a hybrid circuit, a 40M QRP transceiver built with both technologies. I used both 0805 SMC and 1/8W thru-hole resistors, though the .031" pads are still high enough to mount standard 1/4W resistors. The surface mount ADEX-10L mixer is a low-power (LO= 0dBm) version of the ADE-1.

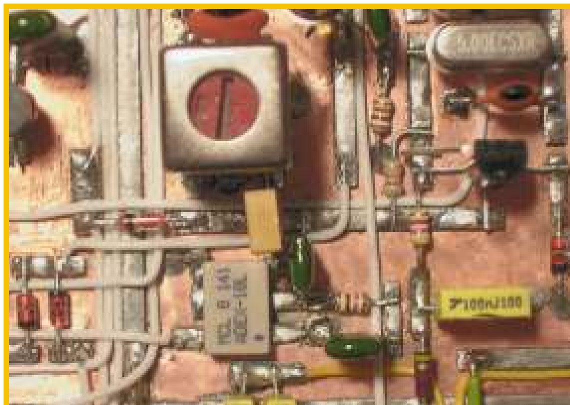


FIG. 18 – A "hybrid circuit" – a combination of thru-hole and SMC using Manhattan-031.

A hybrid circuit is an excellent way to get your "hands dirty" with SMC. On your next project, build it Manhattan style with traditional thru-hole components, but use some SM parts to "get the hang of it," such as a few SOT-23 2N3904s, a MMIC mixer (such as the ADE-1) or amplifier (ERA or GALI series), or a handful of 0.1uF 1206 or 0805 caps for all the bypass caps.

QRP is a hobby, and many of us love to build things. There's no law that says you can't mix thru-hole and SMC. *Just build!!!*

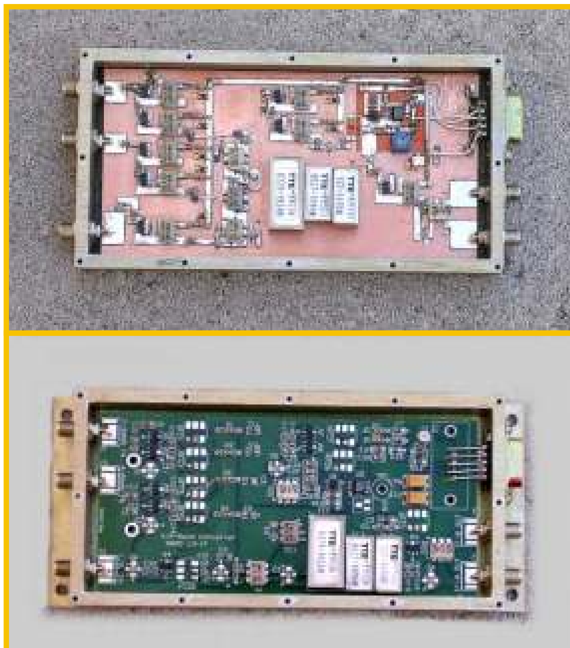


FIG. 19 – The Manhattan-031 prototype VHF upconverter (top) became the model for the PCB (bottom) with few differences, and mounted inside RFI enclosures.

FROM PROTOTYPE TO PCB

Another advantage of Manhattan-031 (thru-hole or SMC) are those situations where it is to serve as a prototype for PC board production. While most QRPers build a circuit on a one-time basis, there are situations where the circuit may be intended for a kit or product. The orderly arrangement of components of Manhattan construction serves as a close model for the PCB layout.

FIG. 20 – The finished VHF upconverter ready for installation in an antenna. There are two converters in each module, one for left-hand and right-hand polarization.

As previously mentioned, some of the circuits shown were prototypes for a VHF upconverter module. **Fig. 18, 19** and **20** shows the evolution of a Manhattan-031 built prototype to the finished product. The PCB shown in **Fig. 19** followed the layout of the Manhattan-031 version almost exactly. The major difference is the PCB uses 50Ω controlled impedance traces, while there was little attempt to do this with the .031" strips. (Theoretically, .031" copper clad .05" wide would be about 50Ω).

5. More Professional Techniques

Hopefully, this article has demonstrated that the way we homebrewer's build SMC circuits with a pair of tweezers and a soldering iron is not indifferent from the professionals. However, there are other methods being used in industry to make SMC construction easier and faster. A few are presented here for information purposes only – to show what a few of the "buzz words" you may have heard are all about – and not neccessarily an endorsement for the hobbyist to attempt.

PASTING TECHNIQUES

There are several forms of "paste" that are used to mount surface mount components to the PCB before soldering. This allows all of the components to be mounted on the board as the first step, then soldering all of the components to the pads in a second step. This saves time over "pick and place" method of mounting and soldering components one-at-a-time.

Hand Soldering Paste. In this method, small drops of paste are placed where the components are to be mounted. The components are then positioned with tweezers on the drops of paste, which remain tacky, much like bees wax, holding the components in place. The components are then soldered in place manually with a soldering iron. Following soldering, the boards are cleaned in water or alcohol to dissolve and remove the paste from the board.

Some amateur's use a similar technique by using glue. *Super glue should not be used*, as when heated, it produces toxic, eye-watering fumes that should be avoided. Others claim success with model airplane glue to Elmer's glue, though I have not personally tried these. These are fast drying glues, such that the components must be mounted immediately, making one wonder, "what's the point?"

Reflow Solder Paste. **Fig. 21** and **Fig. 22** shows solder paste being applied to a PCB. The pasting machine is set for the desired size of the drops, deposited on the board with air pressure to produce drops of uniform size. This type of paste requires refrigeration when not being used. After the solder paste has been applied, the SMC parts are positioned on the board manually with tweezers as shown in **Fig. 23**.

Reflow soldering. Once all the parts have been positioned on the PCB, the board is placed in an oven, usually set for around 350°F, the melting point of the paste and the thin coat of low-melt solder already on the pads. The heat liquifies the paste, pulling the components tight against the pads. The liquified paste also flows onto the pads, acting as a flux, smoothly soldering the SMC pins to the pads. This process takes less heat and solder over manual soldering, plus all components are soldered at the same time. Since the solder was previously ***flowed*** onto the board, this is called ***re-flow*** soldering.

Fig. 21 – Professional assembler Mary Ellen Chavez applies solder paste to a PCB preparatory to reflow soldering.

Fig. 22 – A closer view of the solder paste syringe. Air pressure deposits extremely small drops of paste of uniform size.

Fig. 23 –After manually applying the paste, Mary Ellen Chavez places the SM components on the board with tweezers and microscope prior to reflow soldering.

REFLOW SOLDERING AT HOME

There are various reflow soldering techniques used by the homebrewer. This includes applying the proper solder paste and heating the circuit boards in a convection oven, electric cooking ware, to pans heated on an electric or gas cooking stove. Small pencil-sized heat guns are also used.

These methods are not recommended for the general homebrewer, nor does the ***AmQRP Homebrewer*** endorse methods that includes heating boards over open flames or using pans and convection ovens used for food preparation.

Excessive heat can cause the PCB to warp, the solder masking and screening to literally ignite, setting the board on fire. It can take a fair amount of experimentation to get it right. Although, certainly a "hats off" to those who have mastered it.

If you are interested in this, there are several websites describing these methods for the homebrewer. It is my opinion that for building a one-of-a-kind SMC project, tweezers and a solder iron is by far the most efficient and safe.

STENCIL PASTING

While applying paste and reflow soldering does save time over manual soldering, it is still very time consuming if numerous identical boards need to be built. For high volume PCB work, the paste is applied by a stencil machine, as shown in **Fig 24**. The stencil is a thin piece of anodized aluminum with holes cut out for the exact size of the area to receive the paste. The stencils are generally made by an outside company and cost $100-200, depending upon the size and complexity.

Once the stencil is mounted in the machine, the PCB is placed underneath and properly aligned. The paste is heated and forced through the stencil onto the PCB ... in a very similar fashion to the screen printing process for making custom t-shirts.

The stencil mounted in the machine shown in **Fig. 24** is used for pasting the board shown in **Fig 25**

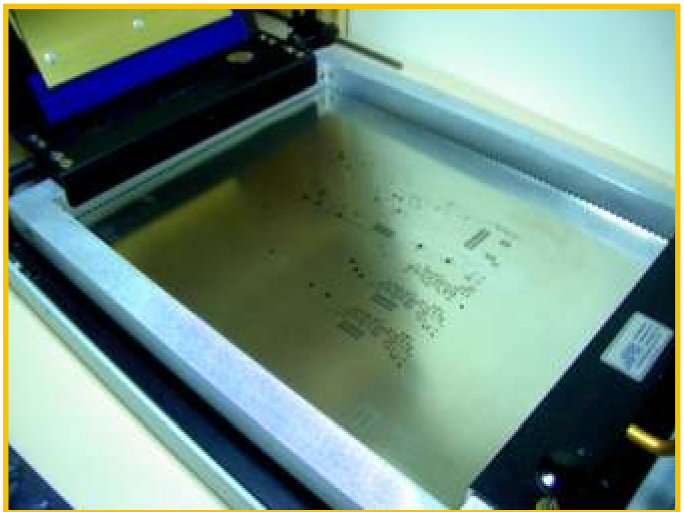


FIG. 24 – A stencil mounted, ready for pasting

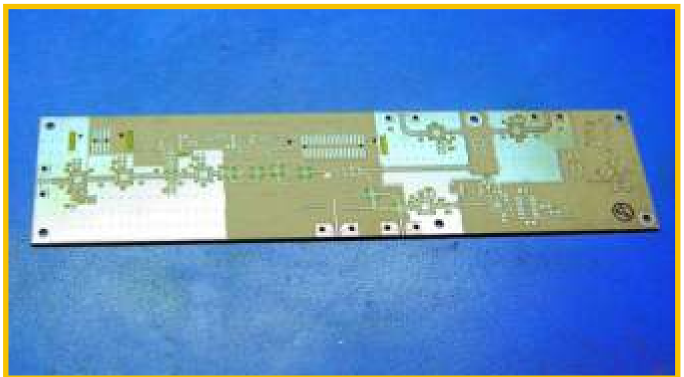


FIG. 25 – The PCB "pasted" with the above stencil. This board down-converts the VLA 8–12 GHz IF to 0–4 GHz baseband, then digitally sampled at 8 GHz.

CONCLUSION

There are numerous methods to build electronic equipment using both thru-hole and SMC. The Manhattan-031 technique presented here has been used by the author and others for building thru-hole, SMC and hybrid circuits with good results. Practice makes perfect. Therefore, try it – and build something. You'll be surprised how easy SMC can be. There is no shortage of thru-hole parts, inspite the claims of some, but SMC, and some of the newer ICs available only in SMC packages, have their advantage and worthy of experimentation by the QRPer.

72, Paul NA5N
.