

# 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.

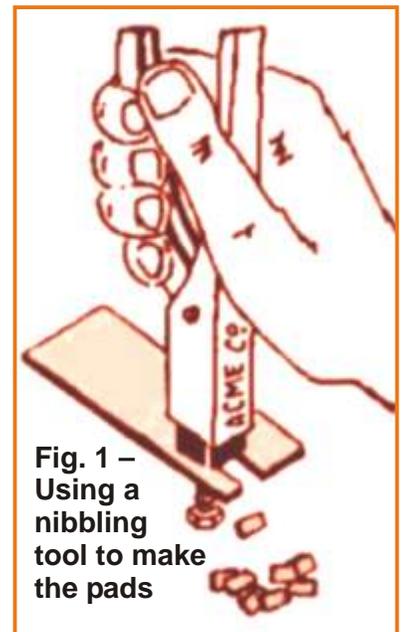


Fig. 1 – Using a nibbling tool to make the pads

**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.

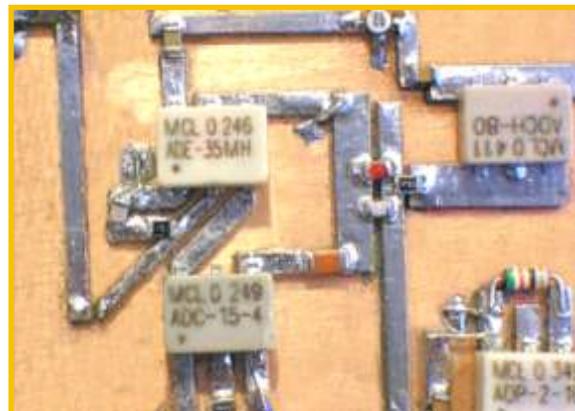
I have developed this technique from building numerous surface mount “Manhattan” circuits, both at home (QRP) and at work over the past couple of years. The combination of using .031” copper clad and the strips works quite well ... *up to around 1GHz*... a technique I’ll refer to as **Manhattan-031**.

### 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. 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.**

**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.

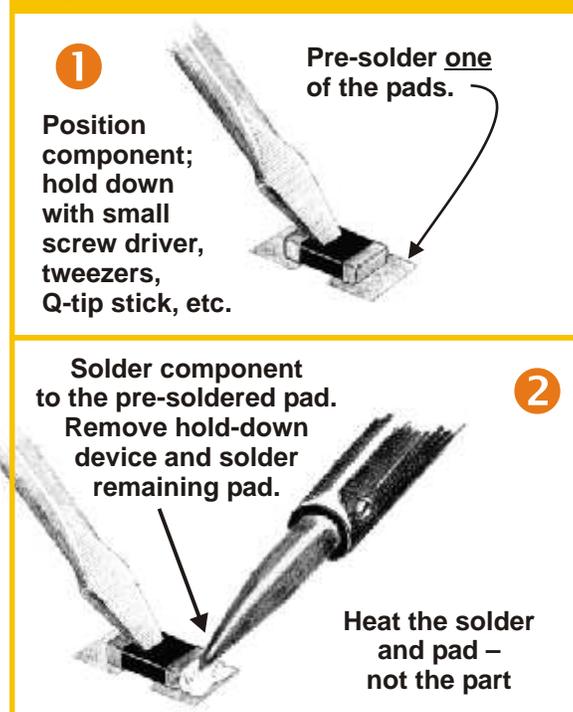
**Do not pre-solder both pads.** This will cause the component to mount at an angle, or noticeable tilt. When soldering the second pad, there is a tendency to push down on the component with the hold-down device or soldering iron to make it lay flat between the two pads. With the first pad already soldered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

**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.

**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.

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

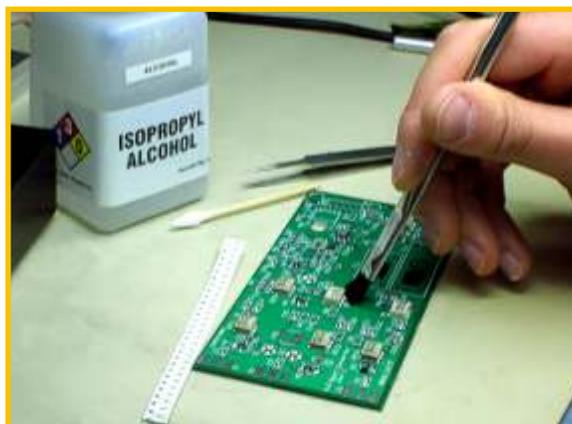
Believe it or not, there is not much difference in how SMC circuits are built by the homebrewer vs. the professionals. For large scale operations, such as building hundreds or thousands of cell phone boards, machines are used for placing and re-flow soldering the components. For building dozens of the 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!



**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.**

### 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!



**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.**

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. 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

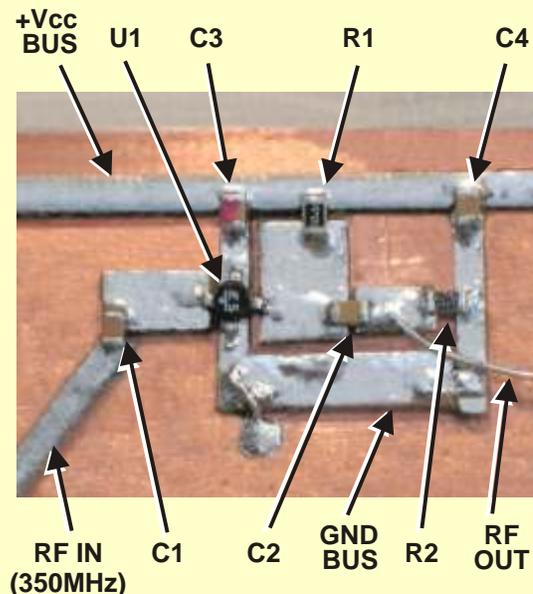
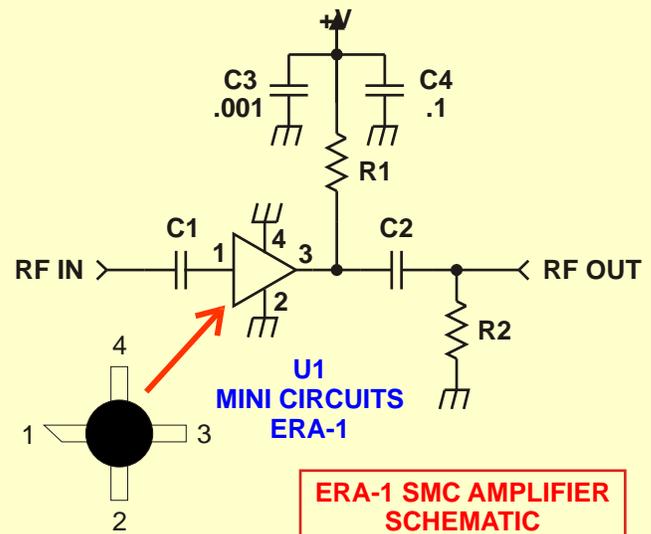
The schematic diagram of a very basic wideband amplifier, using the MiniCircuits ERA series of amplifiers, is shown in **Fig. 11**. The circuit was built entirely of SMC components, using 0805 resistors and capacitors Manhattan-031 style. The ERA is a series of surface mount amplifiers commonly called 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 53.7  $\Omega$ , a 1% precision resistor value. A 5% 51  $\Omega$  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**

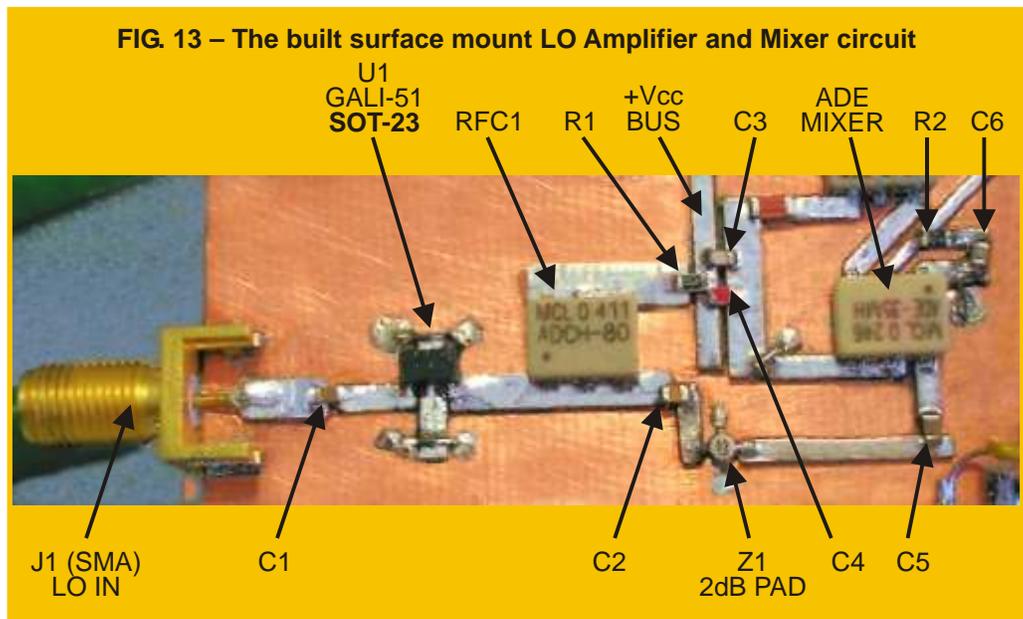
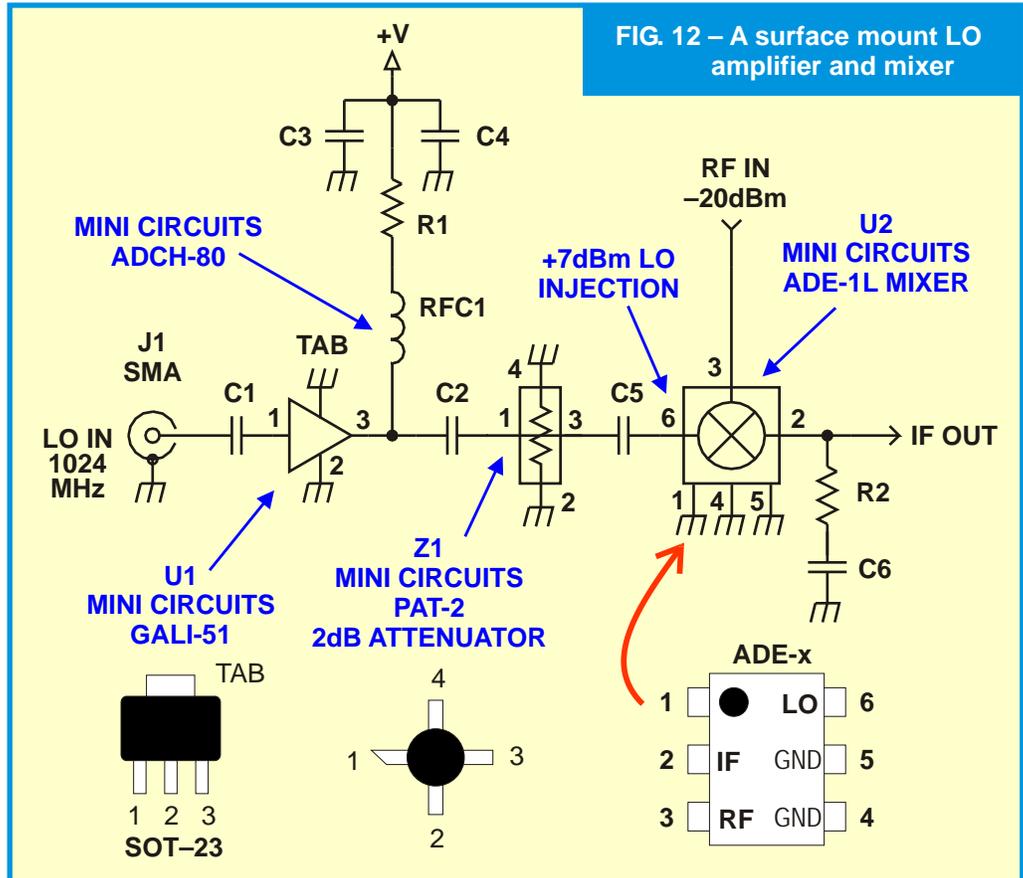
**Fig. 12** is the circuit for a 1024MHz LO amplifier and mixer. This scheme could be used for any HF or VHF frequency.

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 50 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!



## A SURFACE MOUNT POWER DETECTOR

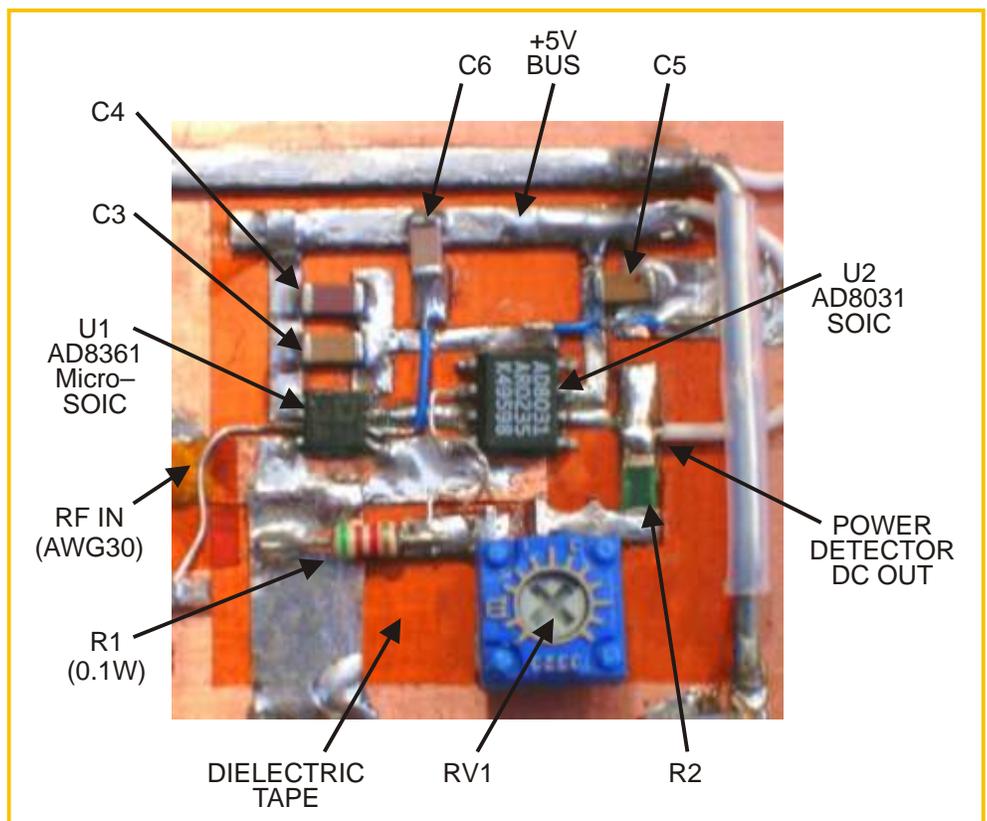
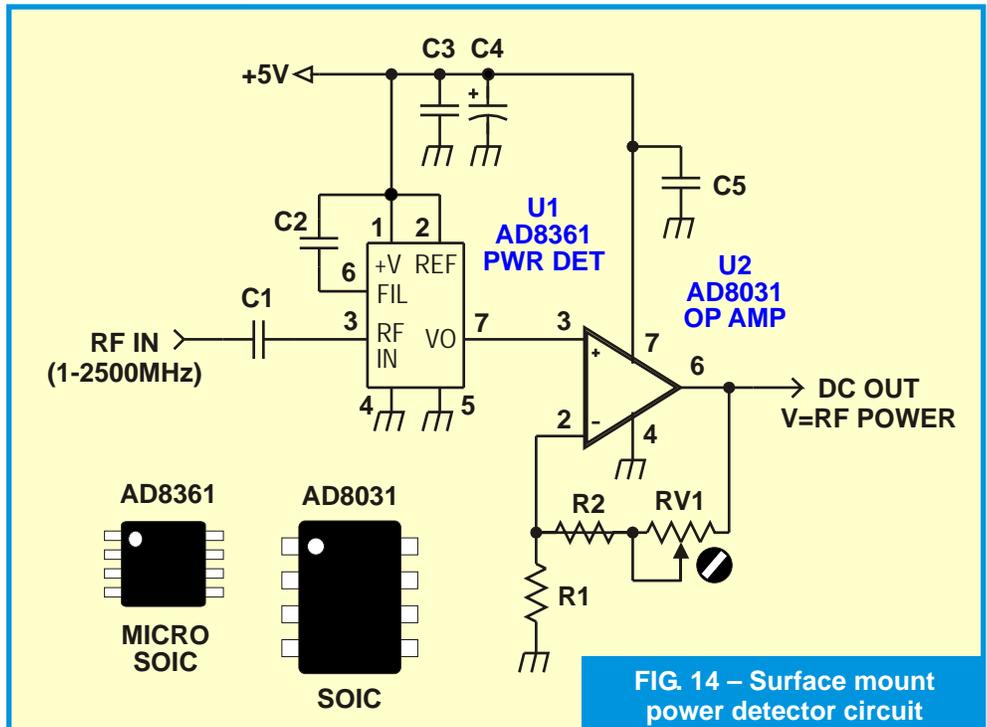
This is another circuit built at work to test out the new (at the time) Analog Devices power detector IC, the AD8361. This would make an excellent QRP project as either a bench power meter (−30 to 0dBm), or with an external attenuator, a power/SWR meter. Except for one thing. The AD8361 is an 8-pin IC 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  $R2 + RV1$  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!



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 straightforward 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.

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  $-3\text{dB}$  frequency of 1090 MHz was  $C=1.7$  and  $2.2\text{pF}$ , and the inductors about  $4\text{nH}$  ( $.004\mu\text{H}$ ). If nothing else, this project verified that .031” Manhattan pads have about  $1.0\text{pF}$  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  $-25\text{dB}$ , or just barely enough for the upconverter to be used. 16-weeks later, the commercial filters arrived, providing  $-60\text{dB}$  of image rejection.

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.



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



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

**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.”

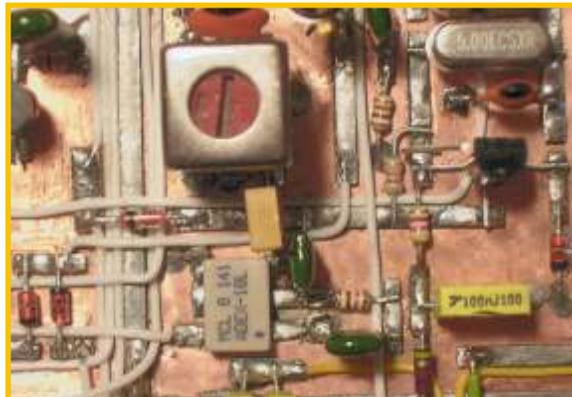
**Fig. 18** shows a hybrid circuit, a 40M QRP transceiver built with both technologies. I used both 0805 SMC and 1/8W thru-hole resistors, though the .031” pads are still high enough to mount standard 1/4W resistors. The surface mount ADEX-10L mixer is a low-power (LO= 0dBm) version of the ADE-1.

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.

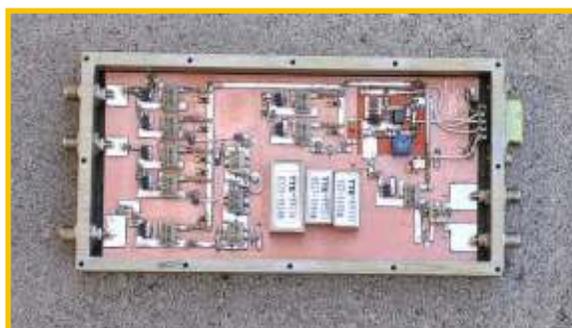
QRP is a hobby, and many of us love to build things. There’s no law that says you can’t mix thru-hole and SMC. *Just build!!!*

**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



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



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



**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.**

Manhattan construction serves as a close model for the PCB layout.

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 50 controlled impedance traces, while there was little attempt to do this with the .031” strips. (Theoretically, .031” copper clad .05” wide would be about 50 ).

## 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 necessarily 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.

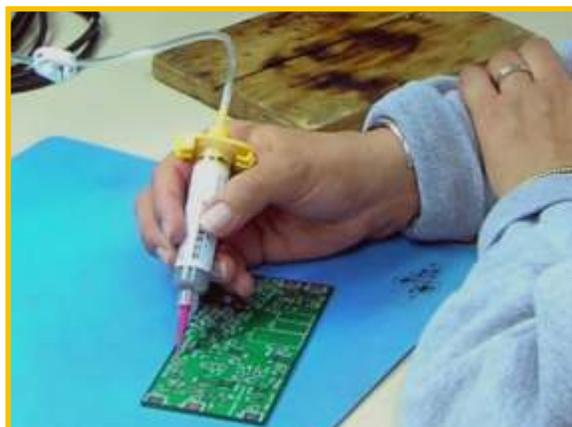


Fig. 22 – A closer view of the solder paste syringe. Air pressure deposits extremely small drops of paste of uniform size.



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.

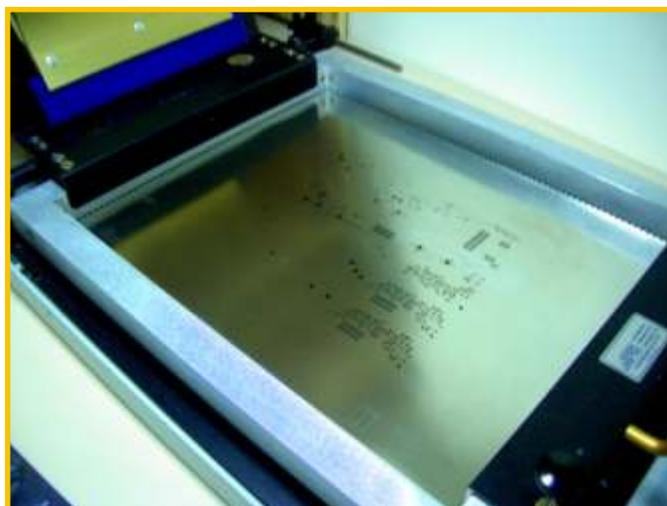
Once the stencil is mounted in the machine, the PCB is placed underneath and properly aligned. The paste is heated and forced through the stencil onto the PCB ... in a very similar fashion to the screen printing process for making custom t-shirts.

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

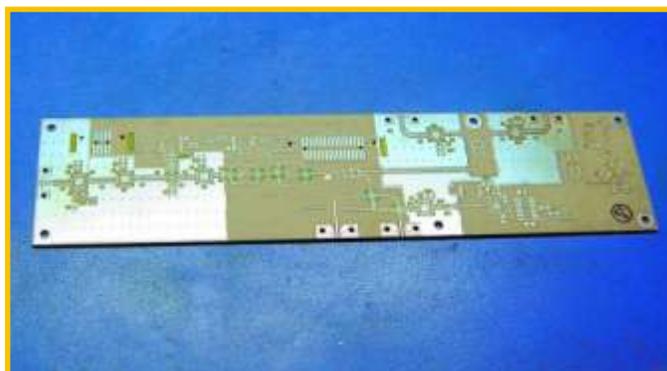
## CONCLUSION

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



**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.**