The "Jersey Fireball 40" Transmitter

Technical Manual & Construction Guide

Introduction

Thank you for purchasing the Jersey Fireball 40 transmitter! You should be able to assemble this kit in about 30 minutes using common tools on your workbench: a low power soldering iron, wire cutters and a voltmeter. A well-lighted area and a magnifying glass will also help.

This document describes the operation of the Jersey Fireball 40 (or "FB40" for short!) and then takes you through construction of it step-by-step. We'll also illustrate simple use of the FB40 with a receiver to give you hours of satisfaction operating in the world of very low power communications called "QRPp".

The Jersey Fireball 40 isn't going to get you any trophies in the DX contests, any pelts in the Fox hunts, or any ooh's & aah's in the "bells & whistles" category of equipment in your shack ... but this little gem will go together quickly and provide all sorts of amazing contacts for you. See how many miles-per-watt you can get with just a 9V battery and antenna!

Okay, just what is the "FB40"?

The "Jersey Fireball 40" is a simple, easy-to-build low-power CW transmitter designed to operate in one of several amateur radio bands. It's name is derived from the fact that the project is designed and kitted by the New Jersey QRP Club members, and that the basic RF power output is close to 40 milliwatts.

The FB40 uses a TTL crystal oscillator "can" as the heart of a milliwatt-level CW transmitter. The designer of our club project, Clark Fishman, WA2UNN, chose an oscillator frequency of 28.322 MHz as a starting point, added some simple circuits to divide this frequency down to hit 80m, 40m, 20m as well as 10m. We also put a low pass filter on the design to clean up all the ratty harmonics coming from the square waves of the ICs and to allow multi-band operation.

You get to select what band in which you want to operate! All you need to do is install a jumper on the printed circuit board to select one of the following frequencies: 28.322 MHz, 14.161 MHz, 7.080 MHz, or 3.540 MHz. The kit comes with filter components for 7.080 operation, but we supply a list of filter component values that can be used to put the transmitter on the other frequencies as well … just substitute a couple of parts from your junk box and you'll be able to operate on 10m, 20m and 80m in addition to the 40m band supplied.

We've also provided pads on the printed circuit board for an optional TiCK keyer chip! This is a small IC that's programmed as a fully-featured iambic keyer, including speed control and other options. All you have to do is drop this chip into the board, add a couple of components and you'll be paddling to your heart's content. (Note: the TiCK chip is not supplied in the FB40 kit. See the parts list for ordering instructions.)

And yet another option that should thrill many hams is that we've provided circuit board pads for an optional RF power amplifier! Once you get expert at making contacts at 40 mW levels, you might want to add a few more parts from your junk box to boost the FB40's output power to around 1 watt. The assembly guide and schematic make this a piece of cake to do.

The Jersey Fireball 40 is really a feature-packed little project. All parts are supplied for stock operation on 40m, and a 2" x 3" double-sided printed circuit card making assembly a breeze.

A Little History

The "fireball" transmitter concept has been around for a number of years and has been published by several individuals. All of these designs, including our Jersey version, are based on the use of a pre-packaged oscillator contained in a metal "can" which is able to be plugged into a standard 14-pin IC socket. These cans are typically used in computers, test equipment and other devices as a source of a stable and accurate master frequency.

Oscillator cans come at various factory-prepared frequencies like 4 MHz, 10 MHz, 40 MHz, etc. However, designers are able to order the cans at specific frequencies they might need for their projects. Our FB40 oscillator frequency was chosen because it was readily available and its 28.322 MHz base frequency divided down relatively nicely into the amateur bands.

Prior articles concerning the use of oscillator cans appeared in Nov 1990 of *73*, April 1993 of *QRP Quarterly*, and Nov 1998 of *73*. You could reference these articles for additional background and use.

Circuit Description

Refer to the schematic later shown in figure 3. The heart of the Jersey Fireball 40 QRPp Transmitter is a pre-packaged TTL oscillator can in the form factor of a 14-pin IC. This oscillator operates at 28.322000 MHz and swings about 1.5 Vp-p.

The basic principle of operation is that the oscillator U1 provides a signal to a series of TTL flip-flop stages that successively divide the frequency in half. These signals are routed through a jumper block that connects to a 5-element elliptical filter to provide appropriate filtering and conditioning of the original square wave signal, thus turning it into a relatively clean transmitted signal.

The divider circuits are 74LS74N dual edge-triggered TTL flip-flops, selected for their low price and low power consumption. They are configured as toggle flip-flops - meaning that for every positive edge transition of the input clock, the output changes state. This is ideal for a divide-by-two function desired for the FB40, knocking the 10m frequency down by half each time we add another gate. The chips operate to well over 30 MHz, so there was little problem with response, delays or signal levels.

The first stage flip-flop (U2a) after the oscillator divides the 28.322 MHz signal in half to yield a 14.161 MHz signal. The second stage U2b divides the 14.161 MHz signal down to 7.080 MHz. And the third stage (U3a) divides the 7.080 MHz signal in half to 3.540 MHz. Admittedly, the specific frequencies are not necessarily hot beds of CW activity (except perhaps on 80m), but they are not too far off the beaten path. Additionally, other oscillators can easily be substituted to achieve better/different frequencies through the divider chain.

The outputs of the flip-flop stages, and the original 28.322 MHz signal itself, are all fed to a jumper block where the builder/user can select which signal is presented onward to the next stage. The basic kit as-shipped needs to have the jumper in the third position, allowing the 40m signal to pass on to the output filter.

The output filter is a 5-element elliptical filter with components determined by using Wes Hayward's popular RF Analysis software programs. A table of values is shown below:

	C4	L1	C6	L2	C5
80m	1700pF	2.17uH	2400pF	2.17uH	1700pF
40m	820pF	1.1uH	1000pF	1.1uH	820pF
10m	230pF	0.3uH	330pF	0.3uH	230pF

Table 1: Output Filter Component Values

In order to further reduce the cost of the project, the filter inductors are constructed by winding toroids to achieve the desired values. The standard formula for the number of turns was pulled from Paul Harden's, NA5N "Data Book for Homebrewers and QRPers" (see reference at end of document):

$$N = 100 * SQRT (Ldesired / AL)$$

So with 1000 pF being the desired inductance, we need about 16 turns for 40m operation. If you'd like to operate the FB40 on 80m, 20m or 10m (which would be really nice in the coming sunspot peak!), just substitute the appropriate caps and inductance (# turns) per Table 1 above.

The output of the FB40 was measured using an HP spectrum analyzer. The second harmonic of the fundamental 40m signal was seen at 45 dB, and the third was seen at 52 dB … not to shabby for a TTL oscillator can, some divider chips and a simple filter. And certainly good enough for safe operation at these power levels.

The FB40 is keyed by bringing the ground pin of the TTL oscillator to ground at the KEY connector pad at the left side of the board. If undesired levels of key clicks or hash are found to be in the RF output when keying, it may be necessary to place a 0.01uF capacitor across the KEY-GND pads.

The output of the FB40 was measured at 10 dBm, which corresponds to 10 milliwatts into a 50 ohm load. This varied from unit to unit and tended to be a function of how strong the TTL totem pole outputs were in the 74LS74 chips. (They aren't optimally made to be looking at 50 ohms, so the interface to the output filter isn't quite ideal.) But even so, with almost no insertion loss, the filter takes the signal and presents it effectively to a 50 ohm load, such as a tuned antenna feedline or an ATU.

The power source input to the FB40 board can be any voltage in the range of 9-14 volts. The LM78L05Z three terminal regulator can supply 5V at 100ma safely, and this circuit design operates well within that limit.

The circuits are minimally bypassed with 0.01uF capacitors. We could've enhanced this filtering to better ensure stability, but this is meant to be a pretty simple project and we tried to keep the parts count.

The TiCK keyer was made a construction option by providing the circuit pads on the pc board for the device. This cool little iambic keyer chip is an 8-pin IC provided by Embedded Research (see reference at end) that enables you to simply connect a paddle to the DIT and DAH input pins, and key the transmitter through a 2N2222A driver transistor. The TiCK can be programmed by grounding the PGM connector pad at the bottom of the pc board, per the instructions provided by the vendor. Speed, memory, weighting are all controllable parameters for this chip.

In order to provide the builder with an easy option of boosting the RF output power, we decided to include a broadband small-signal linear amplifier. Designed to take the 40mW QRPp signal from the first output filter, the 2N2222a single-stage RF amplifier circuit boosts the flea signal up to almost 1 watt, providing about 13 dB of gain. This provides the user with just a little more respect on the air!

Shunt feedback is provided between the base and collector of Q2, and degenerative feedback is also provided by the unbypassed emitter resistor. The value of the bypassed emitter resistor R7 was chosen to establish the quiescent collector current of the transistor.

A transformer is used to step the amplifier stage's impedance down from 200 ohms to 50 ohms. A pi-filter (same Hayward calculations) was used to filter the output of the power amp to provide a clean signal.

Building the Kit

This section describes how to put your kit together. As you progress through each step, be sure to put a checkmark in the boxes to help you keep track of things during the interruptions (phone calls, kids pulling at the sleeves, dog biting your slippers, rare DX coming from the rig's speaker, etc.)

❑ Step 1: Set up

The first thing you should do is find a clear, well-lighted spot on the bench or table. You'll need a fine fine-tipped 30W (or so) soldering iron, wire cutters/strippers, and a voltmeter or DVM. (It will be helpful if the DVM is able to read milliamps too.) If you have difficulty reading small print up close, you might want a magnifying glass to help out in some areas of the construction. If you have an oscilloscope, you'll be able to see waveforms of the RF signals as you build the board.

❑ Step 2: Parts Inventory

Carefully empty the contents of the parts bag onto the table and identify all the supplied parts. This section will help you do that. Put a small checkmark in the box once you've located the parts.

- ❑ Printed Circuit Board (qty 1). This is a light-green, 2" x 3" board with traces on one side (the top, or component side) and a full ground plane on the other (bottom). You will carefully place all the components into the little holes of this board and solder the connections.
- ❑ Oscillator, 28.322000 MHz (qty 1). This is the heart of the project. It's in a metal can about the size of a 14-pin IC. It has four pins on the bottom that will be soldered into the circuit board.
- ❑ 74LS74A integrated circuit (qty 2). This is a 14-pin IC that has some other markings on it, but you'll definitely see "74LS74" someplace on it.
- ❑ LM78L05Z voltage regulator (qty 1). This is a 3-terminal regulator that looks like a transistor, with a flat side on which the part numbers are printed.
- ❑ 0.01 uF capacitors (qty 5). These small orange disc caps are connected together on a tape. The marking on the caps show "103Z", which indicates the .01uF value.
- ❑ 820 pF capacitors (qty 2). These light tan caps also have their values marked on the side: "821M"
- ❑ 1000 pF capacitor (qty 1). This light tan cap has its value marked the side: "102"
- ❑ Toroid cores (qty 2). These T37-2 cores look like little donuts that are .37" in diameter. They are painted red on one side and gray on the other.
- ❑ Magnet wire (about 24" coiled up). This thin #24 wire with a red coating will be wound upon the toroid cores to make the low pass filter inductors.
- ❑ Hookup wire (about 6"). This white insulated solid #20 hookup wire will be cut up and used as jumpers on the PCB.

If you are missing any parts, please contact the NJ-QRP Club at the address listed at the end of these notes so we can replace them.

❑ Step 3: PC Board Orientation

Okay, now that you've got a lay of the land, let's get to work. The first order of business is to inspect the printed circuit board (PCB) and understand how you will mount the components.

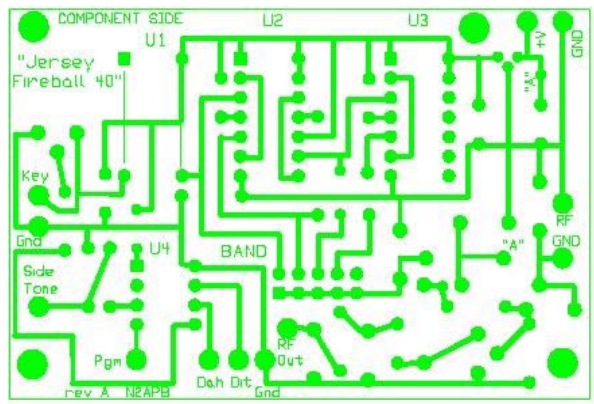


Figure 1: PCB (Component Side)

The first thing you'll note is that the board has "copper" on both sides. The top (component) side has all the signal lines and many of the labels (like U1, U2, BAND, etc.). You'll also notice that there are two pads located along the right side having black marks on them. The labels indicate RF and GND, but they are actually reversed from what they should be. We draw your attention to this reversal by marking over the labels.

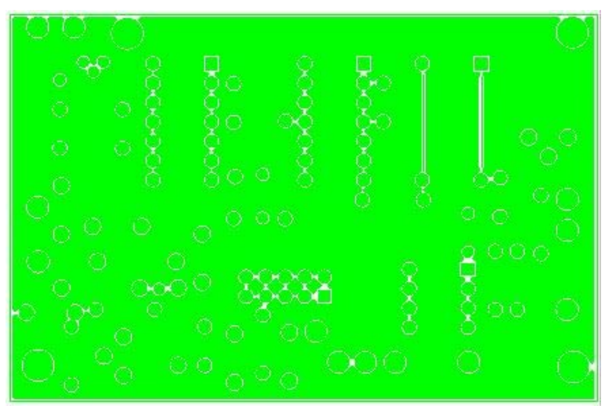


Figure 2: PCB (Bottom Side)

If you turn the board now over to look at the bottom side, you'll see little isolated pads surrounded by the ground plane. Such a large ground plane is used to help shield the weak signals from interference. When you solder the components, you will be doing it on this bottom side. Because of the close spacing of the ground plane to the isolated pads, you will need to be very careful not to use too much solder and bridge across the pad to ground. We'll check your work at various points to help ensure that there are no shorts.

By the way, you'll probably note that it is quite possible to solder all the components from the top side, thus reducing the chance of solder shorts to ground … you might consider doing this if your solder iron tip is not fine enough, or if your hand isn't steady enough. All of our prototype assemblies went together fine by soldering on the bottom, so you shouldn't have any problems.

The PCB has a 1/8" hole in near each of the four corners that can be used for mounting the board in an enclosure when assembly is completed.

❑ Step 4: Install the Voltage Regulator

Locate the LM78L05Z three terminal voltage regulator and carefully insert its pins into the location shown as "VR1" in the upper right hand corner of the Component Layout (figure 4). The flat portion of this TO-92 package should be oriented as shown. Carefully solder the three pads on the bottom side of the board and clip off the excess lead lengths.

Make sure your battery is disconnected from its clip, or that your power supply is turned off and insert the wire of the positive lead of your voltage source through the hole in the +V pad at the board edge. Do the same for the negative lead in the GND pad. Solder both in place from the bottom side.

IMPORTANT: When you solder the negative lead, bend the wire over on the bottom of the board and also solder it to the ground plane. That is, intentionally bridge over from the pad to the ground plane. This will ensure connection to the system ground.

Measurement #1: Voltage Regulator

The first test measurement could be done now to give yourself the confidence of having soldered successfully. Turn on the power supply (or connect your battery) and immediately feel the case of the VR1 device. If it gets warm, immediately turn off power to the board and inspect the three VR1 solder pads for shorts to the ground plane.

If the voltage regulator did not get warm, measure the voltage on the +5V bus running along the top of the board (referenced to the GND pad) … you should read something very close to 5V. If not, ensure that you have 9V or more coming into the board, and inspect the VR1 pads for shorts.

❑ Step 5: Install the Jumpers

You now need to install 7 jumpers on the PC board. Refer to the Component Layout (fig 4) for the location of these jumpers, indicated by a short thick lines around U1, U2, U3 and the pads labeled BAND. Cut the appropriate length of white hookup wire provided, insert the wire into the board holes from the top, and solder to the pads on the bottom side. You might find it convenient to strip off all the insulation for these lengths and just use bare wire. Make sure the jumpers don't make contact with any traces underneath them.

Measurement #2: Jumper Continuity

With your DVM set to the "ohm" setting, measure between each jumper and the GND pad on the power input to ensure the all but one of the jumpers are not shorted to ground. (The horizontal jumper between and below U1 and U2 is ground, so you should get a "zero ohm" reading.) If any of the other jumpers show a short to ground, carefully inspect their pads on the bottom for solder bridges.

❑ Step 6: Install the Oscillator

Locate the metal can oscillator and ensure that its 4 pins are straight and perpendicular to the bottom of the device. Orient the oscillator package with pin 1 in the upper left corner as shown in the Component Layout (fig 4). Insert the device into the PCB at U1 with pin 1 going into the square hole pad. Solder the device in place on the bottom of the board and clip off excess lead length.

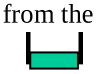
NOTE*: If you have 14-pin IC sockets available, you might find it convenient to use sockets for the oscillator and ICs. Doing this will make it much easier to find and fix problems that might occur downstream. Also, if you later want to change out the oscillator for that of another frequency, you'll be able to do so much more easily by using sockets.*

❑ Step 7: Install Bypass Capacitors

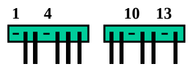
Locate the 0.01 uF capacitors (orange) and insert them through the holes at C1, C2 and C3 on the Component Layout (fig 4). Solder the pads for these components, making certain not to create a solder bridge. Clip off the excess lead length.

❑ Step 8: Install U2 & U3 "74LS74" ICs

You will need to prepare the 74LS74 integrated circuit chips prior to insertion and soldering. First ensure that all leads on the chips are perpendicular to the body of the device. You can bend all the leads against the table such that the packages look like a square "U" when viewed from the end, like below:



Next, you must break off pins 1, 4, 10 and 13 on both of the chips. (With your needle nose pliers, bend the specified pins back and forth until they break off from the chip's plastic body.) This action floats the enable pins on the internal gates thus allowing the gates to operate properly. (Note: This is the not the most elegant solution, but it is certainly the easiest, and the circuit will operate just fine. The board will be changed in the next artwork revision to better facilitate this correction.) The sides of both chips should look as shown below:



Now the chips may be inserted into the positions for U2 and U4. Refer to the Component Layout (fig 4) to ensure proper orientation of the package. Solder the pins of the chips on the bottom side of the board. Again, be very careful to avoid solder bridges to the ground plane.

Measurement #3: Check for U1-U3 pin shorts

Again with your DVM set to the "ohm" setting, measure the resistance from ground to each pin on U1, U2 and U3. If any of the measurements indicate zero ohms (other than the normally grounded pin 7 on each chip), there is a solder short. Carefully inspect the suspect pads on the bottom for solder bridges and resolve the problem before continuing.

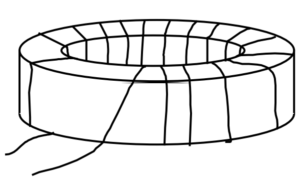
Step 9: Install Filter Caps C4, C5, C6

Locate the light tan colored capacitors and install them as per the Component Layout (fig 4). Be sure to place the 820 pF caps ("821M") into positions for C4 and C5, and the 1000 pF cap ("102) into the position for C6. Solder the pads and (IMPORTANT) bend the excess component leads going into the ground pad and purposely create a solder bridge on those ground pads. In this way we create a good ground path for the RF in the filter. And while you're at it, bridge over the ground connector pad on the right side of the board for the same reason; but be sure you do this to the upper pad (mislabeled RF). Trim the excess leads and check elsewhere for solder shorts.

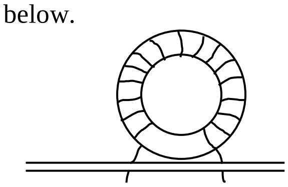
❑ Step 10: Wind the Filter Inductors

Now comes the real fun part of the project … the dreaded ***toroid winding exercise***! You need to create two inductors, L1 and L2, for the low pass filter. It's really not that tough. Uncoil the red magnet wire and cut in half - you should end up with two pieces each about 12" long.

Both inductors will be constructed exactly the same way by wrapping 16 turns of the magnet wire around a toroid core. Count one turn each time the wire is passed through the core. Refer to the figure below for guidance.



The heat strippable magnetic wire being used requires no scraping to clear the red insulation off the leads being soldered to the PCB pads. Once the wires of each inductor are trimmed to the right length (determined by temporarily inserting them on-end into position L1 and L2 on the board), tin the ends of the wires by doing the following. Using a good hot soldering iron, place the tip under the end of the wire to be tinned and add a little solder so that there is a small pool of molten solder and flux on top of the iron with the wire in the pool. After several seconds, the insulation will melt away and the wire will be tinned where it is in contact with the iron. Continue moving the iron slowly toward the toroid core adding solder as you go, until the wire is tinned within 1/16 inch or so of the core. Repeat the procedure for the other leads and brush off any carbon residue from the ends of the wires before you insert L1 and L2 into position on the circuit board as shown below.



Tug the wires gently from the bottom of the board to ensure that the toroids are securely in place and then solder the wires to the pads.

Assembly is now complete! Go have a cup of coffee as a reward.

Putting the FB40 on the Air

Connect a 50 ohm 40 meter antenna to the RF and GND connector pads on the right side of the board. (Remember that the RF and GND labels are labeled in reverse on the board.)

Do not connect the FB40 transmitter directly to the same antenna as is feeding the receiver (or transceiver in receive mode). You will likely damage the receiver. If you only have one antenna to use for both transmit and receive, you should put a SPDT toggle switch in to switch the antenna between the transmitter and receiver, or use some kind of automatic antenna switchover scheme (like PIN diodes for QSK mode.)

Connect a hand key (or a temporary jumper) between the KEY and GND connector pads on the left side of the board.

Apply power and go through some preliminary checks to ensure that there are no shorts. Check for 5V to the IC's and ensure that the voltage regulator VR1 isn't getting too warm. If either condition is not right, go back and look for the solder bridges and proper placement of components. The total current being supplied to the board should be significantly less than 100ma.

With a receiver tuned close to 7.080 MHz, you should hear the Fireball 40 signal. The tone should be steady and free of any hash-like interference. If you can key the transmitter, you should hear very little clicks at the edges of the envelope (i.e., at turn on and turn off times.) If you do hear some gudge in there, try adding one of the left-over 0.01 uF capacitors (orange body, "103Z") across the KEY-GND pads on the board. This should help.

That's it! You can mount your Jersey Fireball 40 in a suitable enclosure, or just have it sitting out on the desk. Several of us in the NJ-QRP Club have put the FB40 into Altoids tins …. It's amazing how the size of the PCB worked out just right for that.

Go forth and communicate!

Optional Installation of a TiCK Keyer

Installing the TiCK keyer chip from Embedded Research makes operating a CW transmitter so much easier. Costing only about $5, this little 8-pin IC will enable you to use your Bencher, NorCal, or whatever kind of paddles you might happen to own.

Insert the TiCK into the PCB at the spot indicated for U4, being careful to align the device with pin 1 in the upper left corner as shown in Figure 5.

Insert a 2N2222A transistor at Q1 and place a 4.7K ohm resistor at R3. Also be sure to add a jumper just above U4 pin 1, as this supplies power to the TiCK.

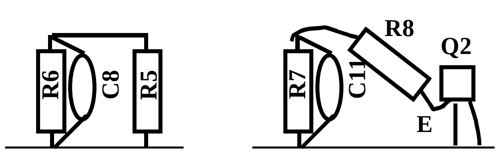
If sidetone is desired, add R1 and R2 next to the TiCK. The sidetone output of the board can be used to feed an audio amplifier or a small speaker, providing a tone whenever the DIT or DAH paddle input is grounded.

A pushbutton may be connected from the PGM connector pad to ground, allowing the TiCK to be programmed in the manner described in its data sheets. The paddle connections are made to the connector pads DIT, DAH and GND.

Optional Installation of an RF Power Amp

If you'd like to operate at more than the 40 milliwatt level, you can add parts for a relatively common one-transistor power amplifier and boost your RF output to almost 1 watt.

Add the components indicated in figure 6 to produce a single stage 2N2222A with a pi-filter on the output. The only tricky part in populating the amplifier part of the circuit board is that we ran out of room on this rev of the board. There are a couple of components that need to be paired up and stood on end in order for everything to fit. Refer to the diagrams below, and the schematic in order to assemble the R6/C8/R5 and R7/C11/R8/Q2 component groups.



The transformer T1 is wound on a ferrite toroid for broadband operation. The primary is 16 turns and is connected to the PCB in the same way as described for L1 and L2. The secondary winding (wound right on top of the primary) is 8 turns, and comes out to the PCB pads as shown in the layout in figure 6.

The output filter inductor, L3, is constructed in the same manner as L1 and L2 in the elliptical filter described above.

Contact Information

1. The "Jersey Fireball 40 QRPp Transmitter" is copyright 1998 by C. Fishman and G. Heron. All rights reserved.
2. Clark Fishman, WA2UNN, PO Box 150, Andover, NJ 07821. E-mail: cfishman@pica.army.mil
3. NJ-QRP Club is selling the basic 40m kit of parts and pc board for $10 postpaid. Write to NJ-QRP Club, George Heron, N2APB, 45 Fieldstone Trail, Sparta, NJ 07871. Website: http://www.njqrp.org E-mail n2apb@amsat.org
4. Embedded Research (supplier of the TiCK keyer chip), PO Box 92492, Rochester, NY 14692.
   E-mail: http://www.frontiernet.net/~embres/
5. Paul Harden's, NA5N, excellent reference book is called "Data Book for Homebrewers and QRPers", ISBN 0-913945-57-9, and costs about $20. Contact Paul at na5n@rt66.com

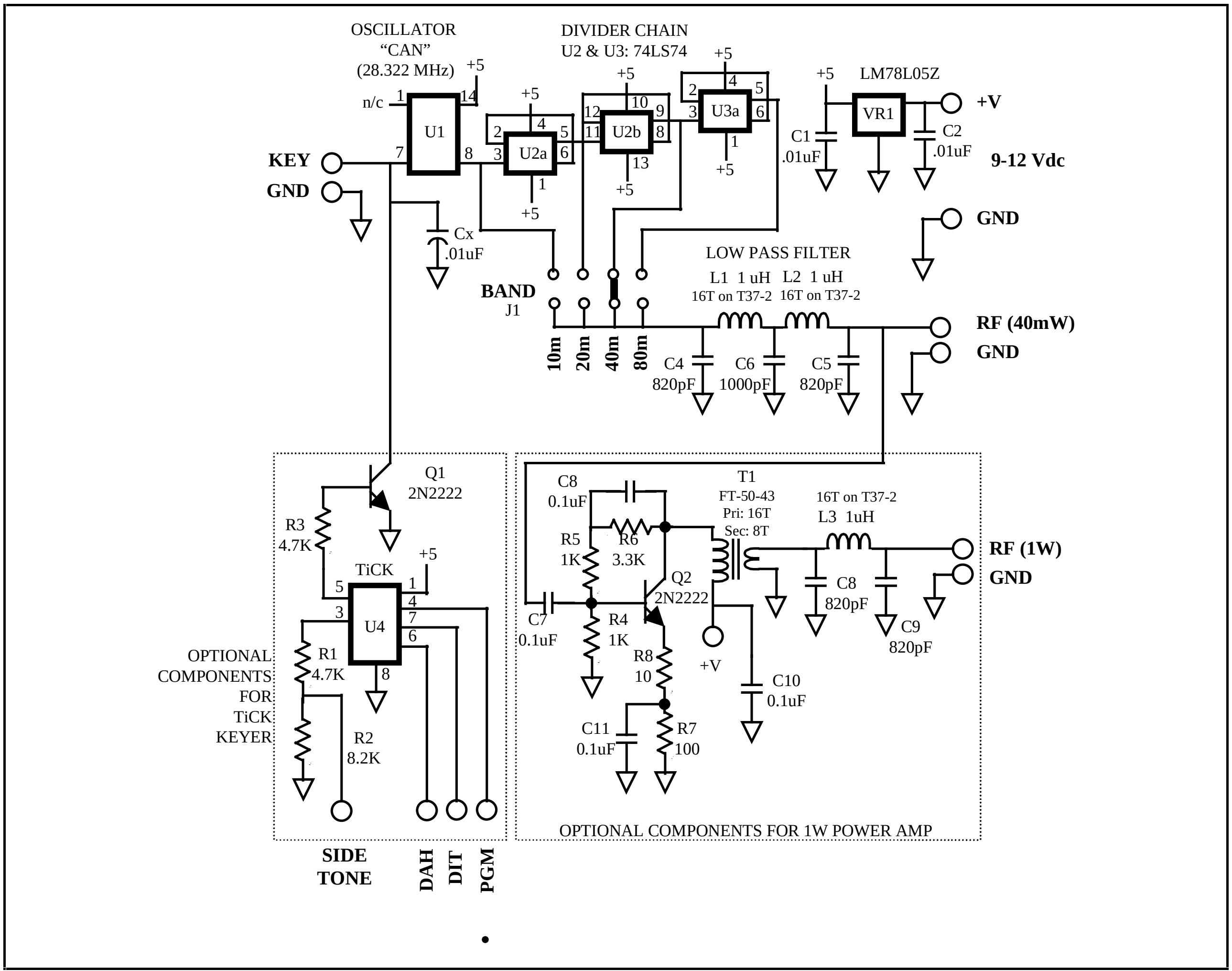


Figure 3: <u>Schematic for the JERSEY FIREBALL 40</u>

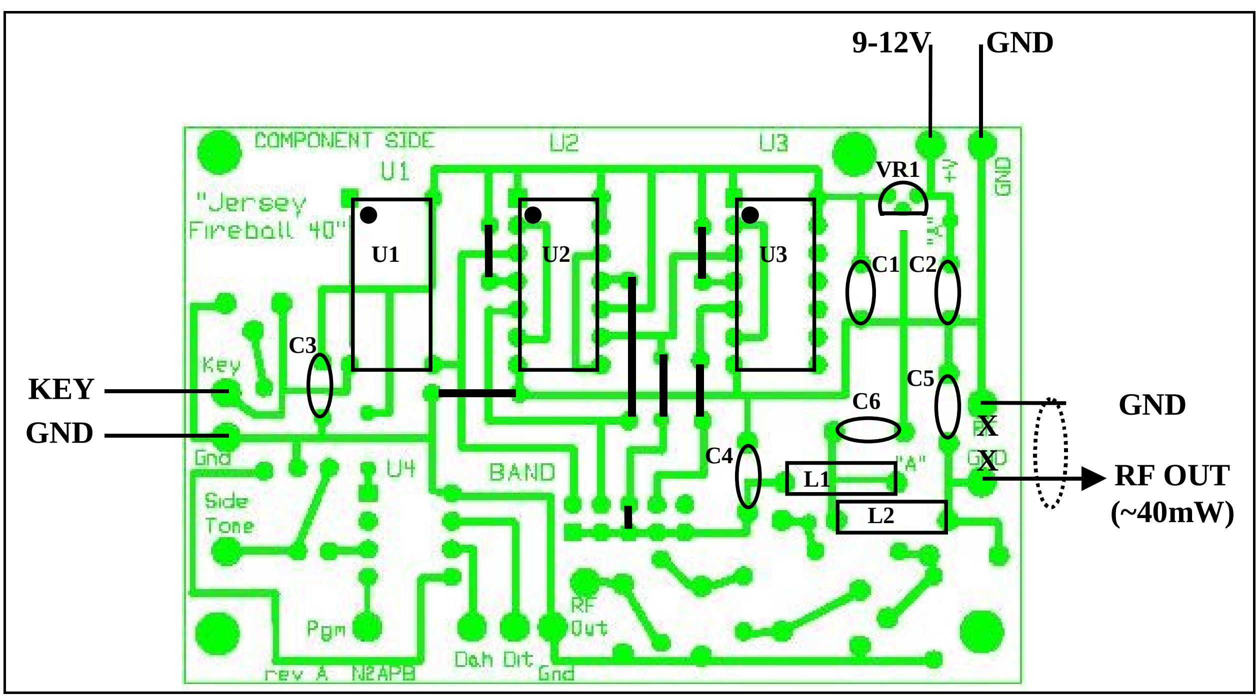


Figure 4: <u>Component Layout (for stock 40m operation)</u>

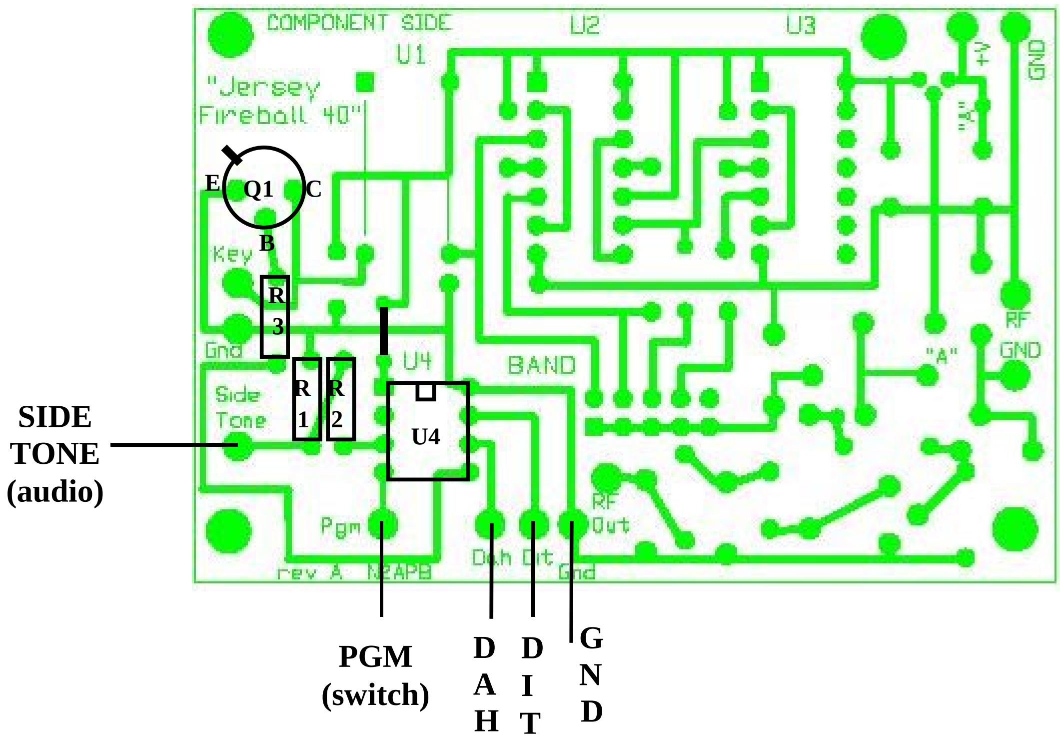


Figure 5: __OPTIONAL TiCK KEYER__

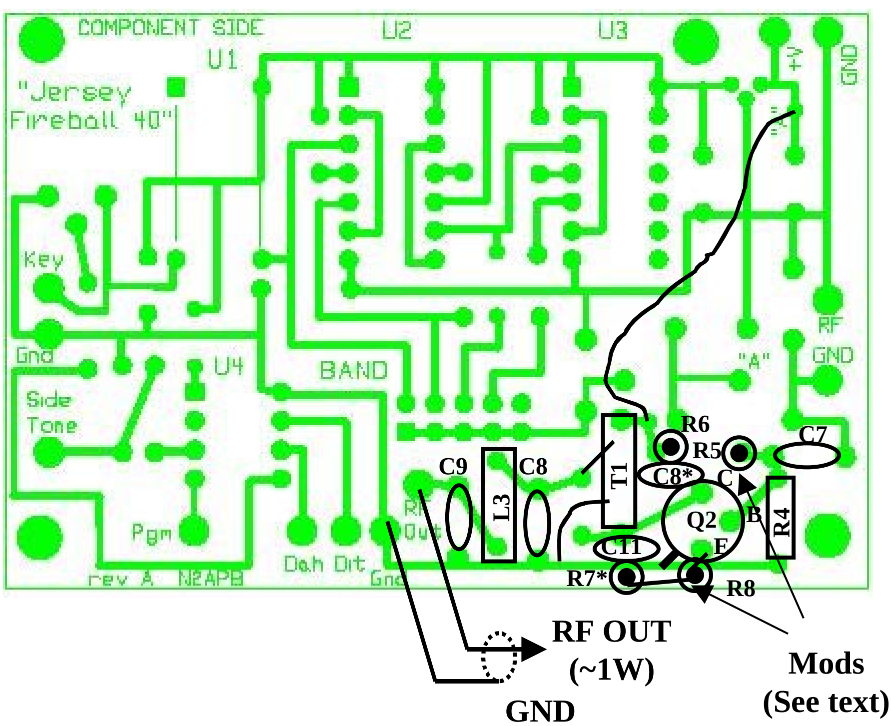


Figure 6: __OPTIONAL RF POWER AMP__