RFBitBanger – Batch 2 January 2024 Kit Builder's Notes

Thanks for your donation in exchange for a kit from batch 2 of the RFBitBanger! This document is to help you get started with the kit, and build it successfully.

Use the Source

RFBitBanger is an open source project. That means that everything about the design, hardware and software, are available to you. It was designed by Daniel Marks, KW4TI, and his GitHub repository is the primary source for all the documentation and source code. Find it at https://github.com/profdc9/RFBitBanger. Be sure and read the RFBitBanger Manual, found in the Docs directory. These kit builder's notes are not a substitute for the Manual.

You'll find several designs in that repo, many of which may be changing from time to time. The specific main board design implemented in the batch 2 kits is found in the RFBitBangSMT directory, as of December 3, 2023. This is a version intended specifically for kit manufacture using many surface-mount (SMT) devices. The circuit board was manufactured and assembled by JLCPCB, and all the SMT parts were chosen from JLCPCB's inventory, by LCSC part numbers. You received the circuit board with all the SMT parts already soldered on.

In the repo, you'll find a PDF version of the schematic for easy reference. You will also find the design files for use with KiCad. Even if you don't plan to change the design, you may find it useful to install KiCad so you can use its features to study the schematic and circuit board layout. KiCad is free and open source, available for macOS, Windows, and Linux, and pretty easy to learn.

In addition to the main board, the RFBitBanger requires a plug-in band filter for each band you wish to operate. The specific filter board design provided with the batch 2 kits is found in the SerialResonantFilter directory, as of May 18, 2023. This is a hybrid board design, in that it may be built with through-hole disc capacitors or SMT capacitors in 1206 or 0805 sizes. At the time batch 2 was kitted, we were unable to find any source of through-hole capacitors appropriate for the filter design, so we had to use SMT capacitors. The kit includes 12 filter boards, and enough SMT capacitors and toroid cores to enable you to build one band filter for each of the nine HF bands from 80m to 10m.

Inspecting the Main Board

We suggest that you start by familiarizing yourself with the main board. Take it out and look at it. You'll see that many of the parts are already installed, but there are many others that are not installed. You'll find unpopulated through-hole parts and also unpopulated SMT parts. In building the kit, you'll populate most of the through-hole parts, but not all. Here's a list of through-hole parts that are not included in the kit.

Through-Hole Parts NOT TO BE INSTALLED on the Main Board		
Ref Des	**Description**	**Reason**
Q5, Q15	2N7000 transistor	Three PA transistors (Q3, Q6, Q7) are enough
Y2, Y4, Y5	7 MHz crystals	We use the Si5351 synthesizer and don't need the crystal oscillator. The cheap parts for the crystal oscillator are installed, though, so you can install crystals here and adjust JP5 and JP8 if you wish to experiment with it.

You'll also find a number of SMT pads that are unpopulated, and don't need to be installed. If you find any unpopulated SMT locations not listed in the following table, that is a manufacturing defect (or else an error in the table).

SMT Parts NOT TO BE INSTALLED on the Main Board		
Ref Des	**Description**	**Reason**
R53	Resistor	Not needed.
R67	Resistor	Not needed.
C32	Capacitor	Not needed.
R56	Resistor	Not needed.
C18	Capacitor	Not needed.
C43	Capacitor	Not needed.
C60	Capacitor	Not needed.
C71	Capacitor	Not needed.

There are also a bunch of through-hole locations for header pins.

Header Pins on the Main Board			
Ref Des	**Pins**	**Install?**	**Notes**
JP1	2	Yes	Normally shorted. Open for full break-in QSK.
JP2	2	Yes	Shorted to disable RF current sample LED.
JP3	2	Optional	Shorted to power the board from ICSP connector. Useful only in special cases like replacing the microcontroller chip.
JP4	2	Optional	Shorted to bypass the band filter. The filter is required for clean (legal) transmissions. Bypassing it is useful only for test or in emergencies when the appropriate band filter is not available. If you don't need that feature, you can omit JP4.

JP5	3	No	Selects either the Si5351 synthesizer or the crystal oscillator. Since we have only the synthesizer, permanently install a wire between the pin marked PLL and the center pin, unless you plan to experiment with the crystal oscillator.
JP6	2	Yes	Shorted when a microphone is connected to J11, open when a line-level audio input is connected to J11
JP7	3	Optional	Short center pin to BYP pin to bypass the band filter; short center pin to FLT pin to enable the band filter. The filter is required for clean (legal) transmissions. Bypassing it is useful only for test or in emergencies when the appropriate band filter is not available. If you don't need that feature, you can omit JP7 and solder a wire from the center pin to the FLT pin.
JP8	2	No	Permanently install a wire to disable the unused crystal oscillator, unless you plan to experiment with it.
JP9	2	Yes	Shorted when a microphone is connected to J11, open when a line-level audio input is connected to J11
JP10	2	Optional	Access to measure the power amplifier bias voltage. Used during tune-up.
J5	4	Yes	Serial port for programming microcontroller (Arduino style) or for terminal emulation in digital modes
J9	6	Optional	ICSP (in-circuit serial programming) connector. Required only in special cases.
J10	7	No	Used to connect a Si5351 synthesizer on a separate module board. We have the Si5351 directly on the main board, so we don't need this (and can't use it without removing some of the pre-installed SMT components).
J17	4	Yes	Input to the band filter.
J18	4	Yes	Output from the band filter.
J22	2	Optional	Connect an external speaker here. Parallel with J8.
None marked on the circuit board	1 each	Optional	There are three single-pin header locations on the board. Two of them are GND (the one marked GND and the one next to D15 in the upper right corner of the board) and the third is a test point for the signal named AUDIOFILT (next to R24, just below the display). It's useful to have at least one of the GND pins installed. Note that the six mounting holes are also GND (J3, J4, J6, J7, J20, and J21 on the schematic), as are the two heatsink areas on the board (J13 and J23 on the schematic) and most other bits of exposed metal on the components. We recommend not installing a header in the AUDIOFILT location (J14 on the schematic) unless you need it.

That leaves a whole bunch of through-hole parts that you will be installing on the main board. Please refer to the separate [Batch 2 Kit Inventory] document for a complete list.

Unless I've made a mistake in this document, every part location on the main board should appear in one of the tables above or in the Batch 2 Kit Inventory.

Inventory the Parts

Carefully open up the bag of parts and identify them using the photos in the separate [Batch 2 Kit Inventory] document for reference. If you're missing any, let us know on the rfbitbanger-talk channel on [ORI's Slack] and we'll make it right.

Two DC power connectors are provided, J1 and J2. You can install both, or either one, or you can install neither and solder wires directly to the board. We recommend installing both. Use the coaxial receptacle J2 for convenience, but if you're caught out in the field without a matching power supply, you can always connect bare wires to the screw terminals on J1.

The BNC antenna connector, J12, has four pins. The two small, flimsy ones carry the signal and ground and must be soldered to the board. The two thick, robust pins are not connected to any signal. They are just for strength. There are no pads on the circuit board for the thick pins, just holes. You can still use them by soldering a nice round blob of solder to each of the thick pins. This will help keep the connector from flexing the signal pins. Like this (the PCB is green because this photo was taken with a prototype board):

You'll notice in the photo above that there are two extra pads for the BNC connector. They are for compatibility with a different version of the connector. Similarly, there are also extra pads for the volume and gain potentiometers.

Don't confuse R41 (a 2-watt film resistor) and L10 (an axial lead inductor). They look somewhat similar.

Think About Your Build Strategy

By and large, you can solder in the through-hole component in any order you like. Most people find it easiest to install the shorter components first, building up to the taller components. That order would be something like this:

1. Zener diodes, D1 and D6
2. Fixed resistor R41 and inductor L10
3. Pushbuttons, SW1 through SW6
4. Transistors Q3, Q6, and Q7 (all 2N7000) and Q8 (SS8550) – (these all lay flat against the heatsink areas on the board)
5. 3.5mm TRS jacks J8, J11, and J19
6. Transistor Q9 (the fourth 2N7000) – (this one stands upright)
7. Trimmer potentiometers RV3
8. Electrolytic capacitors C50 and C52
9. Pin headers (see table above and discussion below)
10. Trimmer potentiometer RV4
11. Volume and Gain potentiometers RV1 and RV2
12. Electrolytic capacitors C6, C19, C40, and C49
13. Relay and connectors
14. Electrolytic capacitors C3 and C53
15. Toroids (see separate document on Winding the Toroids)
16. LCD screen and band filter (see discussion below)
17. Heatsink

When you solder in pin headers, it can be a little tricky to get them in straight. The usual suggested procedure is to solder one pin at one end of the header. Then put a finger on the top of the header at the opposite end (don't touch that soldered pin!) and reheat the solder joint from below. Move the header into the right position and then let it cool before letting go. This will usually get you close enough for jumpers and standalone headers. However, it isn't easy to get the pin headers perfectly vertical by eyeball.

A possibly better procedure, which also works for single pin headers, is to use a spare piece of the female header as a handle. Use it to hold the pin header in place while you solder one pin. Then inspect carefully before soldering the rest of the pins. Don't try to remove the female header until you've soldered all the pins, though. If you do, you might pull some of the header pins out of their plastic carrier.

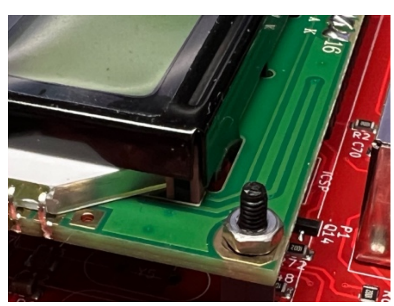
There are two places where the header orientation is more critical: on the connector for the LCD screen, and on the two connectors for the band filter.

To get the LCD headers on right, we recommend this procedure:

1. Install the four standoffs on the bottom of the LCD board using the four screws.

2. Cut a 16-pin length of male header pins. Cut at the notch between the pins.
3. Cut a 16-pin length of female header. To do this, you must cut right on top of the 17th pin. Don't try to cut between pins. You have to waste a pin.
4. Insert the longer pins on the male header strip into the female header strip. Don't push them all the way in, just far enough so they don't fall out easily.
5. Place the joined header strips loosely into the holes on the main board. The shorter ends of the male pins should be in the main board, and the pins of the female header should be up.
6. Maneuver the LCD board assembly with standoffs into place. The pins of the female header have to go through the holes on the LCD board, and the threaded ends of the standoffs need to go through the mounting holes on the main board.
7. Secure each of the four standoffs with a nut on the bottom of the main board, and tighten. Be careful with the nut on the upper right standoff: there's a trace right there. See photo below.
8. Make sure the male and female header pins are flat against the main board and the LCD board, respectively. They should still be fully engaged with each other.
9. Now, and only now, solder the header pins on the top of the LCD board and on the bottom of the main board. Go ahead and solder all the pins. Don't try the usual trick of soldering just the end pins and then removing the mating connector before soldering the rest. If you do that, you'll pull pins out of the header strips.

You will now notice that the threaded part of the standoff sticks out pretty far on the bottom of the board. If this is a problem for you, you have two choices. You can snip the nylon threads off close to the nut, at some risk of making it difficult to ever get the nut off and back on the threads. Or you can install the standoffs the other way around, with the screw on the bottom and the nut on top. That puts the ugly part on top, which may or may not bother you. If you're using a case, you may not have much choice.



To get the filter board headers right, we recommend this procedure:

1. Cut two 4-pin lengths of male header pins. Cut at the notch between the pins.
2. Cut two 4-pin lengths of female header. To do this, you must cut right on top of the 5th pin. Don't try to cut between pins.
3. Insert the longer pins on the male header strip into the female header strip. Don't push them all the way in, just far enough so they don't fall out easily.
4. Place the joined header strips loosely into the holes on the main board. The shorter ends of the male pins should be in the main board, and the pins of the female header should be up.
5. Separate one of the filter boards from the panel of twelve. To do this:
    a. Place the panel at the edge of a square-edged board or table, so that one row of filter boards hangs over the edge.
    b. Line up the V-groove with the edge.
    c. While holding the panel in place, gently press down on the overhanging row of filter boards until the V-groove at the edge gives way.
    d. Wiggle the row of filter boards up and down until the V-groove separates.
    e. Repeat the above steps to separate a single filter board from the row of filter boards.
6. Optionally, install the capacitors and toroids on this filter board, according to your strategy for which band filter to build. This is optional because you can always install the parts later, but it's a bit easier to install the parts on top of the board *before* you install the connectors on the bottom.
7. Place the joined header strips loosely into the holes on the main board. The shorter ends of the male pins should be in the main board, and the pins of the female header should be up.
8. Place the filter board onto the pins at the tops of the joined header strips.
9. Hold the filter board in place so that the headers stay in their holes and stand up as close to vertical as you can manage.
10. Now, and only now, solder the header pins on the top of the filter board and on the bottom of the main board. Go ahead and solder all the pins. Don't try the usual trick of soldering just the end pins and then removing the mating connector before soldering the rest. If you do that, you'll pull pins out of the header strips.
11. When you build additional filter boards, use a similar procedure, using the headers already installed on the main board as an alignment jig for the female headers on the bottom of the filter board. That way your filter boards will all fit your RFBitBanger, even if none of the connectors are perfectly perpendicular to the boards.

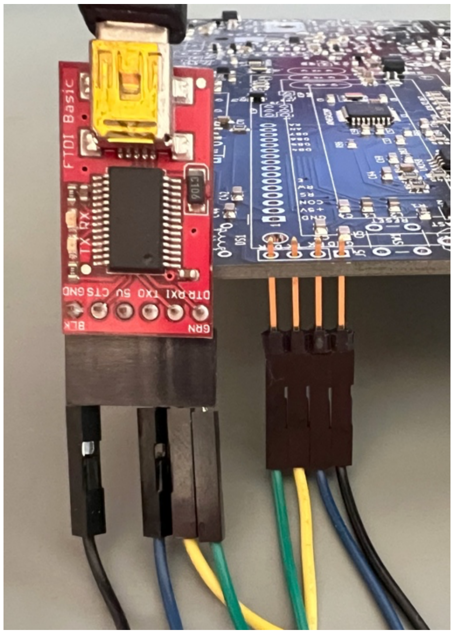
Programming the Microcontroller

The brains of the RFBitBanger is an ATMEGA328P microcontroller. Essentially, this is identical to the popular classic Arduino Uno board, and software development for the RFBitBanger is done with the standard [Arduino] environment.

Arduino boards are delivered with the ATMEGA328P pre-programmed with a boot loader. Normally, this boot loader is permanently installed in the device at the factory and never needs to be reloaded. User programs, called *sketches* in the Arduino world, can then be loaded into the device through a serial port. Most Arduino boards provide a built-in USB serial port adapter, so all the user needs to do is connect a USB cable between their computer and their Arduino board. In the case of some bare-bones Arduino-like systems, there is no USB adapter built-in, and the user has to provide one externally, and also arrange for the board to receive power. This serial port needs just three signals: transmit data, receive data, and the DTR signal, which is repurposed to reset the ATMEGA328P so that the boot loader runs. The RFBitBanger follows the same bare-bones pattern.

When the boards are received from the factory, the ATMEGA328P device is soldered down but not programmed. We then program the bootloader using the ICSP (in-circuit serial programming) port at J9 (located under the display board in a fully assembled RFBitBanger). This is done with a special programming device. We use a [USBtinyISP] from Adafruit, but many other suitable programmers are available. We set the jumper JP3 on the USBtinyISP so that it provides power to the ICSP port, and we temporarily short JP3 on the RFBitBanger board so that ICSP power is allowed to power up the board. Then in the standard Arduino development environment, we select *Burn Bootloader* from the Tools menu and wait a few seconds for it to finish. Ideally, you should never have to worry about the bootloader again. If the bootloader gets overwritten somehow, or if you need to replace the ATMEGA328P for some other reason, you'll need to reload the bootloader using a similar technique.

We then go ahead and program the RFBitBanger sketch into the ATMEGA328P through the serial port, J5. We use a USB to 5V TTL serial adapter, specifically a [5V FTDI Basic] from Sparkfun, though again many suitable adapters are available, many at lower prices. Since we're programming the bootloader and the sketch at the same time, we let the USBtinyISP provide power to the board during both programming stages. The six pins of the FTDI Basic do not line up with the four pins of J5, so we use a simple adapter made out of a 4-position pin header with extra-long pins and four jumper wires. Following the labels on the silk screens, RXI on the FTDI Basic is connected to TX on J5, TXO to RX on J5, DTR to DTR, and GND to GND. Like so (photo shows a black Batch 1 board):



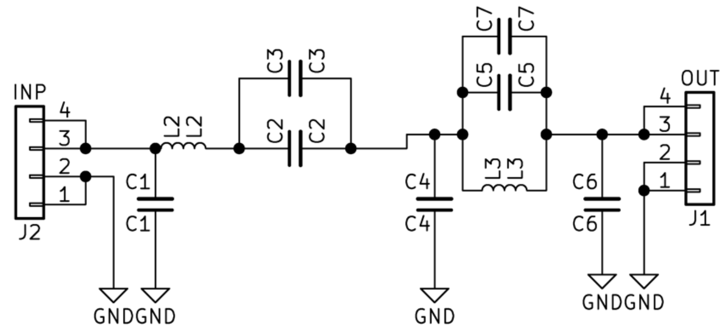
The extra-long pins on the adapter are placed without soldering into the circuit board holes for J5, from below. A tiny bit of downward pressure on the circuit board is enough to make secure temporary connections between the adapter and J5.

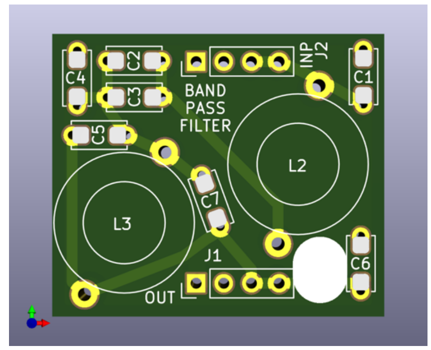
We've programmed the board with the version of RFBitBanger code that was current at the time, so you can get started quickly, but there will be updated versions available in the future. Watch the [RFBitBanger repo] for updates. You will want to be able to update your board with the latest sketch. In order to do that, you'll need to have a USB to 5V TTL serial adapter and a way to hook it up to J5. You may want to solder header pins into J5 and hook the jumper wires to them, instead of building a temporary adapter as we have. It's up to you.

You will need to provide power to the board while reprogramming it. Once the board is fully built up, this is no problem: just use the same power hookup you use to operate the radio.

Band Filters

The RFBitBanger requires a band filter for each band you wish to operate. We've provided you with a panel of twelve band filter boards. Each board requires five to seven capacitors and two toroidal inductors. We've provided the capacitors, toroidal cores, and magnet wire to construct one filter board for each of the nine HF bands from 80m to 10m, plus a very few spares.





The photo shows a green prototype board. Batch 2 boards are red.

C2 and C3 are in parallel, so that their capacitances add together to effectively make a single capacitance in series with L2. Likewise, C5 and C7 are in parallel, making a single capacitance in parallel with L3. The idea here is that you can combine two available standard capacitor values to obtain other capacitance values, and tune the filter more accurately. So, there are effectively five capacitance values on the filter board: C1, C2+C3, C4, C5+C7, and C6. Values for these five capacitances, together with the inductances for L2 and L3, must be chosen to optimize the design of the band filter for a particular frequency.

The designed part values for the band filters are detailed in the RFBitBanger Manual. In these designs, C4 and C6 are always identical, so there are just four capacitance values specified. The Manual leaves it up to you to decide whether to use one capacitor or two for each of the parallel pairs of capacitors, and how to divide up the capacitance if you use two. Besides the target inductance for each of L2 and L3, the Manual gives two sets of winding instructions. One set uses a toroidal ferrite core of a specific size and mix, and the other uses an air core. The circuit board is designed for toroids, and we have provided toroids, so you should ignore the air core directions in the Manual.

Choosing Your Band Filters

You have a strategy decision to make. If you're interested in experimenting with the design of these serial resonant band filters, you may want to build up filters for only one band, or a few favorite bands, and use the rest of the provided capacitors as an inventory of spare parts you can draw on to modify those filters. You'll have a bunch of leftover toroid cores, too. Or, if you just want to build the kit as designed and get going on as many bands as possible, you can follow our plan and build up nine HF band filters. You'll still have three extra filter boards available to experiment with, but you'll need to buy your own capacitors (make sure they are NP0/C0G type) and additional toroid cores to populate them.

Our plan for building up 9 HF band filters is shown below, and is also available as a live Excel spreadsheet for you to work with. The spreadsheet is designed to facilitate planning to build all the desired filters from a limited pool of available components. The table below is a summary of the results of using this spreadsheet to fit all nine HF bands into the supplied parts kit. We've also repeated the toroid winding details from the Manual. Note that these values may not be the best, perfect values, but they are good starting points.

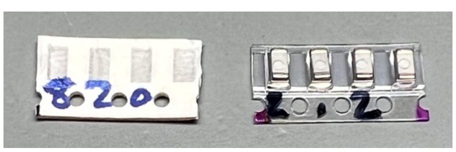
Band	**C1**	**C2**	**C3**	**C4**	**C5**	**C6**	**C7**	**L2 turns**	**L3 turns**
80m	1000	2200	150	2200	680	2200	None	19	13
60m	680	820	820	1500	470	1500	None	16	8 to 9
40m	470	1000	None	1000	330	1000	150	12	8
30m	330	560	220	680	220	680	None	11	7
20m	100	330	150	470	150	470	None	9	5
17m	100	390	None	390	68	390	33	8	5
15m	100	330	None	330	68	330	33	8	5
12m	47	270	68	330	68	330	33	7	4
10m	47	150	100	220	47	220	33	7	4

Building the Filters

Once you've decided which filters to build and with what component values, it only remains to build them. We recommend that you solder the capacitors first, then the toroids, then the female headers to mate with the main board.

Soldering the Capacitors

If this is your first time soldering SMT devices, please don't panic. It's easy.



The capacitors are provided in multiples of four in sections of "cut tape". The devices come from the factory on a long tape mounted on a reel, for automatic dispensing in a "pick and place machine". We've just cut this into lengths of four or eight capacitors for this kit. The tape may be paper based or plastic. Either way, there's a thin film sealing the device within a well in the tape. To get at the capacitor, *carefully* peel back the thin film just far enough to release the number of capacitors you need, and keep the rest sealed up. This might actually be the hardest part of dealing with SMT devices. If you lose control of the tape, the loose capacitor can go flying across the room, and very likely you'll never see it again.

The value is not marked on the capacitor. We've handwritten the value on each little strip of cut tape, in pF for values below 1000 pF (1 nF) and in nF for 1, 1.5, and 2.2 nF values. Once the capacitor gets separated from the tape, it's up to you to keep track of its value. We suggest handling them one at a time. Get it out of the tape, and solder it down before moving on to the next one.

To solder these down, you just need your regular soldering iron, some relatively thin rosin-core solder (what you have is probably ok), and a pair of tweezers. Use whatever magnification you need for fine work, and good lighting. Apply a little mound of solder to one of the flat pads on the filter board. Carefully place the capacitor down in the general vicinity of where it needs to be. Grab the capacitor gently with the tweezers, and put it exactly where it needs to go. One end should be close to or touching the mound of solder, and the other end should be close to or touching the other flat pad on the filter board. While still holding the capacitor in place with the tweezers, touch the mound of solder and the near end of the capacitor with the tip of the soldering iron. This should form a nice smooth curved solder joint between the capacitor and the pad. Remove the soldering iron and wait for the solder to solidify. Now you can let go with the tweezers and set them aside. Inspect the capacitor's placement. If the unsoldered end is still close to the other flat pad, good enough. If not, grab it with the tweezers, reheat the joint, and adjust until it's right. Finally, solder the other pad normally. Congratulations, you've installed an SMT capacitor.

If you have more sophisticated SMT soldering equipment and skills, feel free. But this method works fine.

Winding the Inductors

The inductors L2 and L3 are simple coils, wound on one half of a toroid core, which lays flat on the filter board. We have two detailed sets of instructions for winding the toroids: a PDF document Winding the Toroids and a YouTube video. We recommend you look at both.

Installing the Headers

Please see the section earlier in this document about installing pin headers accurately. There is specific advice for getting the filter board connectors right.

Making a Case

This kit does not include a case for the RFBitBanger, but there is a 3D-printable case design in the CAD directory of the RFBitBanger repo. It consists of a top plate, a bottom plate, and five button extensions. Use the files with "SMT" in the name. You can print the STL files to duplicate the design as is, or you can use the .FCStd files with FreeCAD to customize the design.

This case is not a full box with mounting points for all the connectors. The RFBitBanger board is not really designed to work in such a case. This case is just a top plate and bottom plate, which capture and protect the circuit board, but the edges of the circuit board and all the connectors near the edge of the circuit board are exposed between the plates.

And obviously, a case that has been 3D printed out of plastic is not a shielded enclosure. The RFBitBanger board is designed with a nearly complete ground plane on the back side, minimizing the need for external shielding.

Initial Bring-Up

When you've completed assembly, including installing the toroids, you can apply 12VDC power to either of the power connectors. The LCD backlight should immediately illuminate. If it does not, disconnect power and troubleshoot.

The LCD may display the default frequency, which is `07100000 Frq`, or it may appear blank. Adjust the contrast trim potentiometer, located on the main board below the left edge of the LCD board, for the best appearance on the display. If you can't get it to show the default frequency clearly, disconnect power and troubleshoot.

Operating the RFBitBanger

Please see the RFBitBanger Manual for operational details.