# 10 MHz Reference for FT-817/818/857/897
By
Ray Cannon W7GLF 11-Jan-2022

## 1. Introduction

Ever since owning the Yaesu FT- 817 I have wanted to add the ability to lock it to a 10 MHz reference. I discovered there was a system for sale by VK3HZ that used the CT1DMK RefLok I. The VK3HZ kits are no longer available and the RefLok 1 boards themselves are no longer available. I have some RefLok I boards but trying to recreate the VK3HZ design would require a lot of research.

As an alternative I decided I wanted to fulfill the following requirements:

- The synthesizer board design would simply replace the existing TCXO-9 board.
- It should detect an external 10 MHz input and switch between its internal TCXO (.5 ppm) and the external reference automatically.
- The board should have its own microcomputer to handle the reference switching.
- Expose the I2C SCK and SDA inputs so the board can be used with an external microcomputer and for troubleshooting.

## 2. Design Decisions

My first hurdle was to learn KiCAD to allow me to design a replacement PC Board. Due to the complexity a 4 layer board design seemed desirable. I could use one of the inner layers to distribute the 3.3 volts and the other to be ground. The KiCAD learning process was greatly helped by having something to design and having a friend (Bob W7PUA) who already had designed a number of boards using KiCAD. Advantages of KiCAD is not only is it free and well maintained but it can generate Gerber files which are a standard for PC Board design. As I worked with it I was impressed with how easy it was to move parts around and having it regenerate the fill areas. OSH Park and OSH Stencils both accept the KiCAD files directly without having to generate the Gerber files which made life very simple.

I started with a board outline that was ½” by 1¼” which is the size of the existing TCXO-9 board. I have had experience with a number of synthesizers and the specifications of the Si5351C seemed to be a natural fit for my purposes. One advantage the Si5351C has over the Si5351A and Si5351B synthesizers is that it has 2 separate reference pins - one for an external TCXO (using the XTAL input) and the other for using an external frequency reference. To switch between the sources all that is required is to reprogram the Si5351C chip. The down side is that the Si5351C chips are currently extremely difficult to find - luckily I bought 8 of them when I saw they were becoming hard to find.

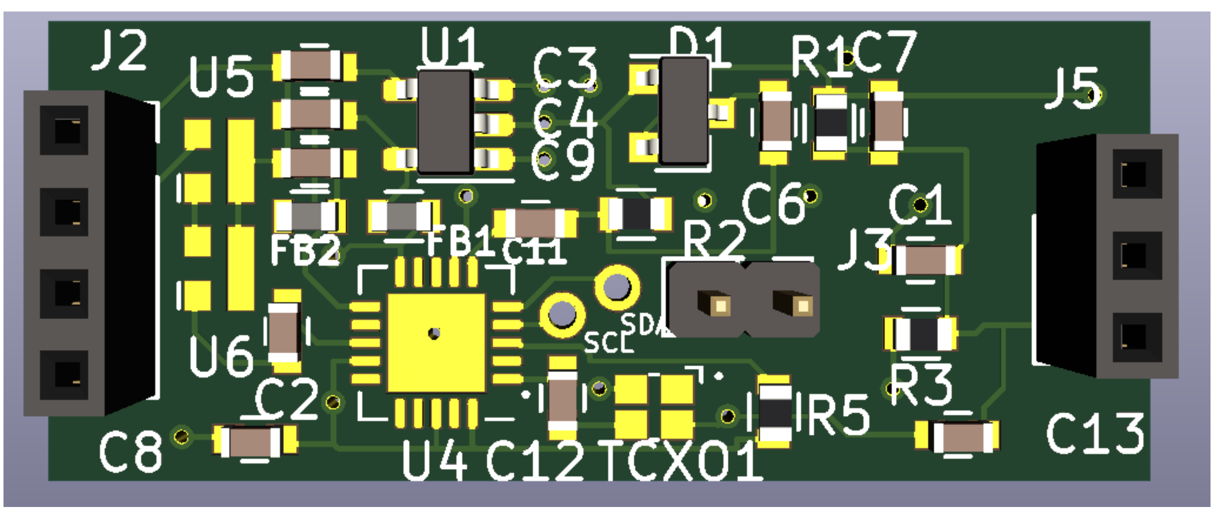
A Si5351A can be used but since it only has one input reference pin the switching between the TCXO and the 10 MHz external reference will need to be done using solid state switches which will require a board redesign.

My next step was to look at the technical manual for the FT-817 to find that the reference level should be 90 mV rms (or -8 dBm). The Si5351C puts out quite a bit more than that (spec'ed to be .4 * Vdd volts PP – which works out to be .4 * 3.3 = 1.32 Vpp or about +6 dBm). I decided to add two 8dB T pad attenuators (being conservative) to the reference output to the radio which is about -10 dBm.

I discovered I also needed to minimize the power drawn by the new reference board. I changed my source code to only enable the first pair of outputs (the Si5351C has a total of eight outputs which can be enabled/disabled in pairs). I also designed the code to sleep after programming the synthesizer until it detects that a 10 MHz external reference was either plugged in or disconnected. At that point it reprograms the synth and goes to sleep again. With these3 changes I got the current draw down to 29 mA. I tested with my 817 and I noticed no thermal heating on its 5 volt 1 A regulator when the synthesizer was plugged in. The 5 volt bus went from 4.97 to 4.95 volts which did not seem excessive. I have used the new board in several 817s running for hours with no heating or other ill effects noticed.

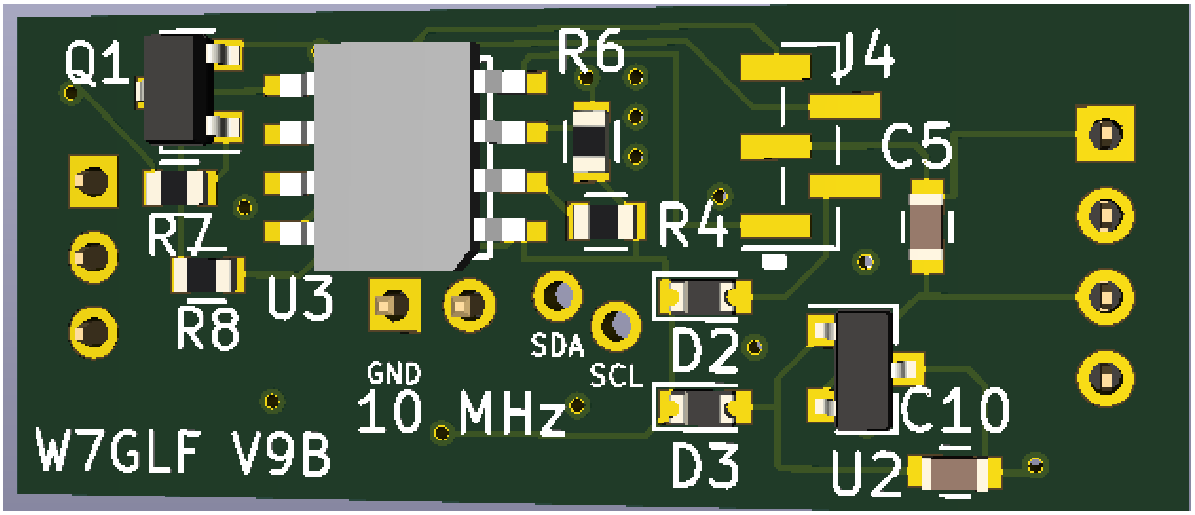
Initially I spent quite a bit of time pushing various choices for parts around on the ½" by 1¼" board with KiCAD. I decided I wanted the RF on one side of the board and the computer and control on the other side. As a result my assumptions about needing a 4 layer board were quickly confirmed.

Here is a picture from KiCAD of the current board design (note it does not have 3-D layouts for the some of the parts):




RF Side

Here is the other side of the board (V9B is a slightly improved version of V9A with markings for diode direction that are under the diodes).  Note cathode marks on diodes (on the left side).




Computer Control Side

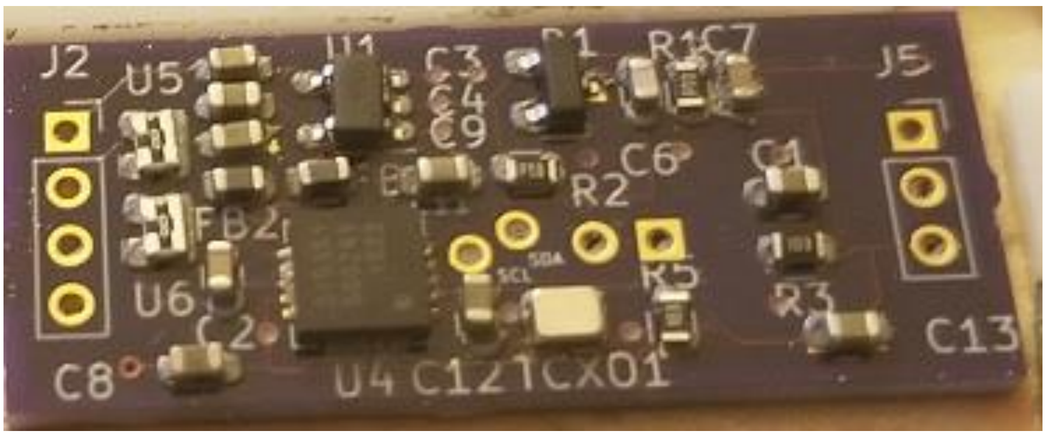
## 3. Building the Board

Several of my parts dictated that I needed to reflow solder at least the RF side of the board.  The TCXO I chose has no legs and the Si5351C has a pad in the center of the bottom of the chip for heat sinking.  There is a nomenclature for these parts.  QFN stands for Quad (leads on 4 sides) Flat No Leads.  I learned something very interesting about parts that have a pad on the center of the bottom.  Many (including the Si5351C) have two hidden contacts on the corners of each side that are connected to the bottom pad.  Thus a chip that says it is a 20 pin QFN chip with 5 contacts on each side might actually have 28 contacts with 7 contacts on each side.  These extra contacts are in the side of the package and do not go to the bottom of the chip and thus they cannot be soldered.  They are there so a machine (or a human) can verify that the bottom pad is actually soldered to the board.  I have tested this and found by taking an ohmmeter and measuring the resistance between ground and one of these “hidden” contacts in the side of the package one can indeed verify the bottom pad has been soldered.

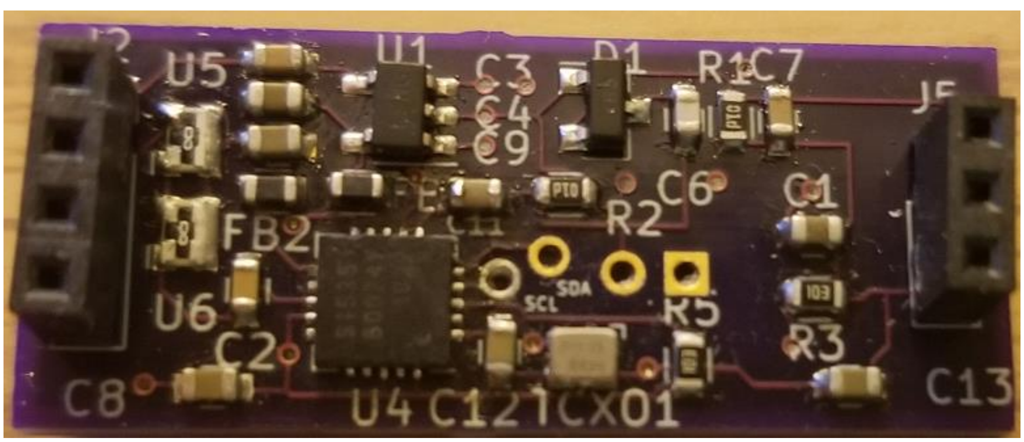
I did a lot of reading and considered using a hot plate but once I placed the parts on the board I did not want to try to lift the board to move it on top of a hot plate.  Another issue is a hot plate takes a long time to cool down so it is desirable to lift the board off the plate soon after the reflow takes place.  With my tiny circuit board I was not clear on how to lift it securely and not spill parts from the hot plate.  I also thought about using a toaster oven but it would be hard to see the board in the oven and I would still have the issue of moving the board into the oven after placing the parts. I finally decided on using a hot air rework station.

I ordered the circuit boards from OSH PARK (only $6 for 3 boards!) and a 3 mil stencil for the RF side of the board from OSH Stencils. I used Kester EP256 solder paste that comes in a syringe for $30.

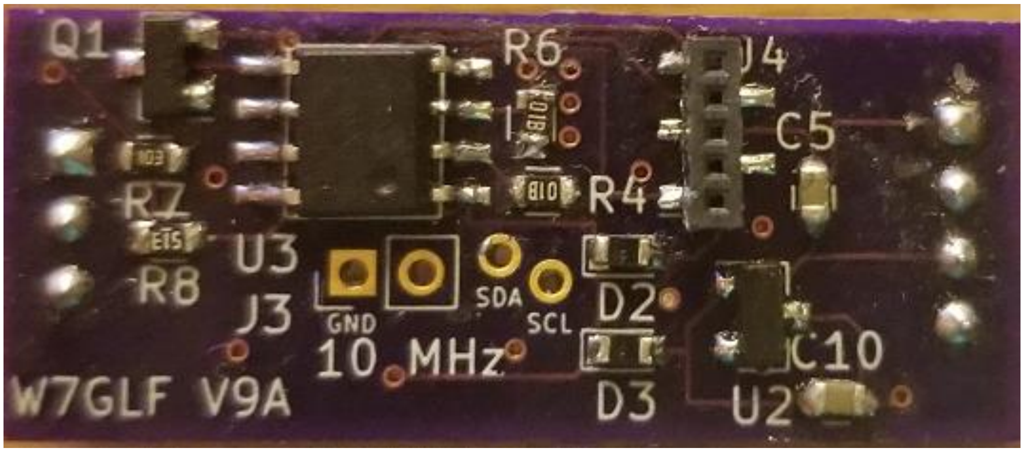
Here is a picture of the one side of the board after parts placement but before solder reflow (note the gray color of the solder paste). I only did reflow soldering on the RF side. I manually soldered the parts to the control side.




Here is a picture of the same side of the board after solder reflow (note how the parts lined up – they really do that):



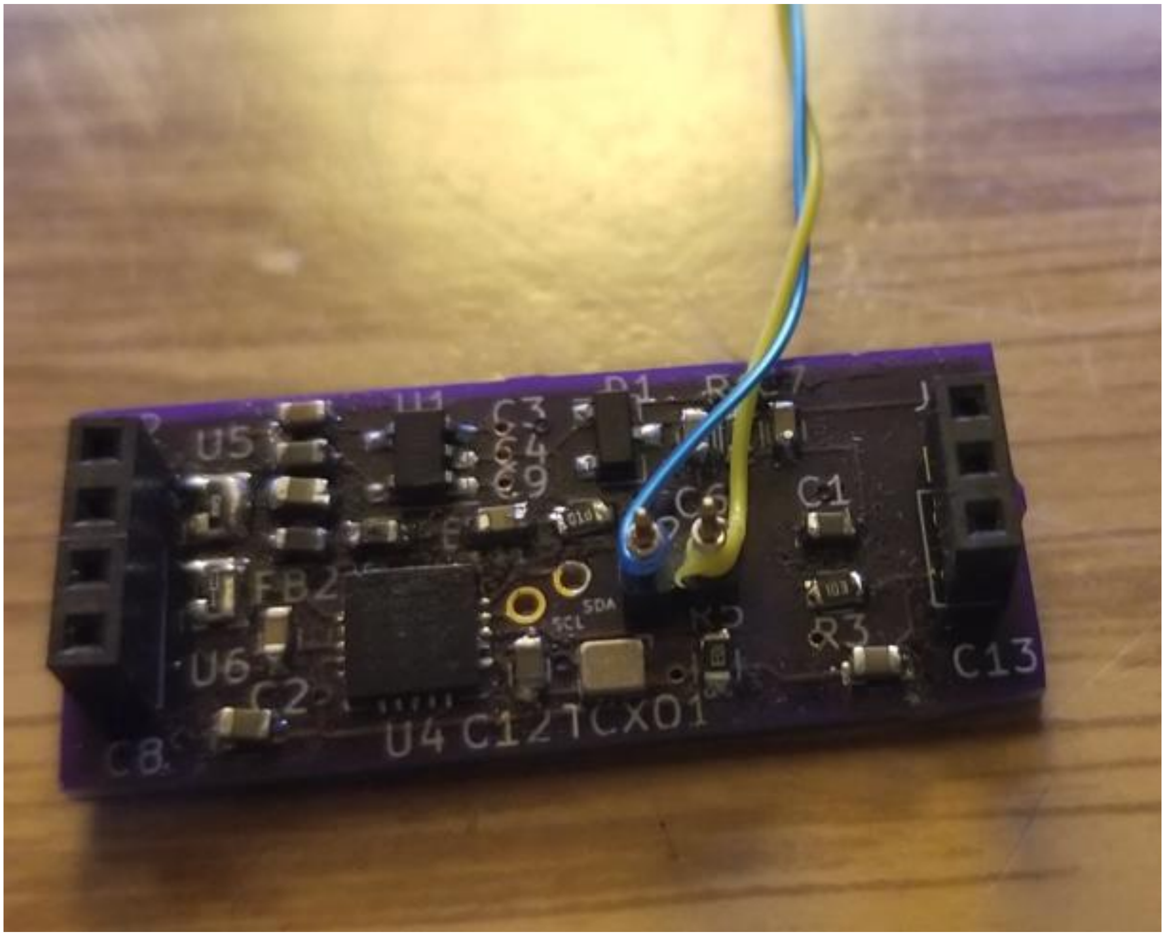

Here is a picture of the rear (control) side of the board -the connector is for ICSP (In-Circuit Serial Programming) for the 12LF1552 microcomputer (chosen for its size):



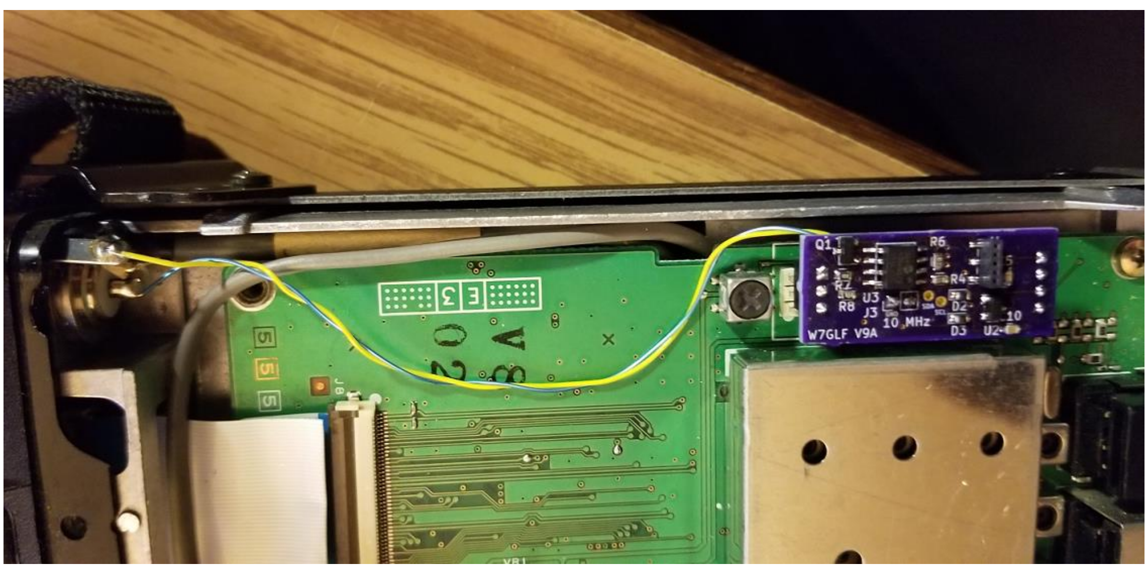

## 4. Mounting the Board

Since I never use the FT 817 front BNC connector I decided to use it for my 10 MHz input connector so I would not have to drill any holes. It was a bit tricky unsoldering the existing connection – I needed to remove the screws for the PC Board so I could lift it enough to get at the front BNC connector. To make the task easier I removed the tape holding the existing coaxial cable.

I was having problems finding coax flexible enough to use so I decided to use a twisted pair of wire wrap wire to connect the board to the front jack. So far wire wrapping the twisted pair of wires seems fine.

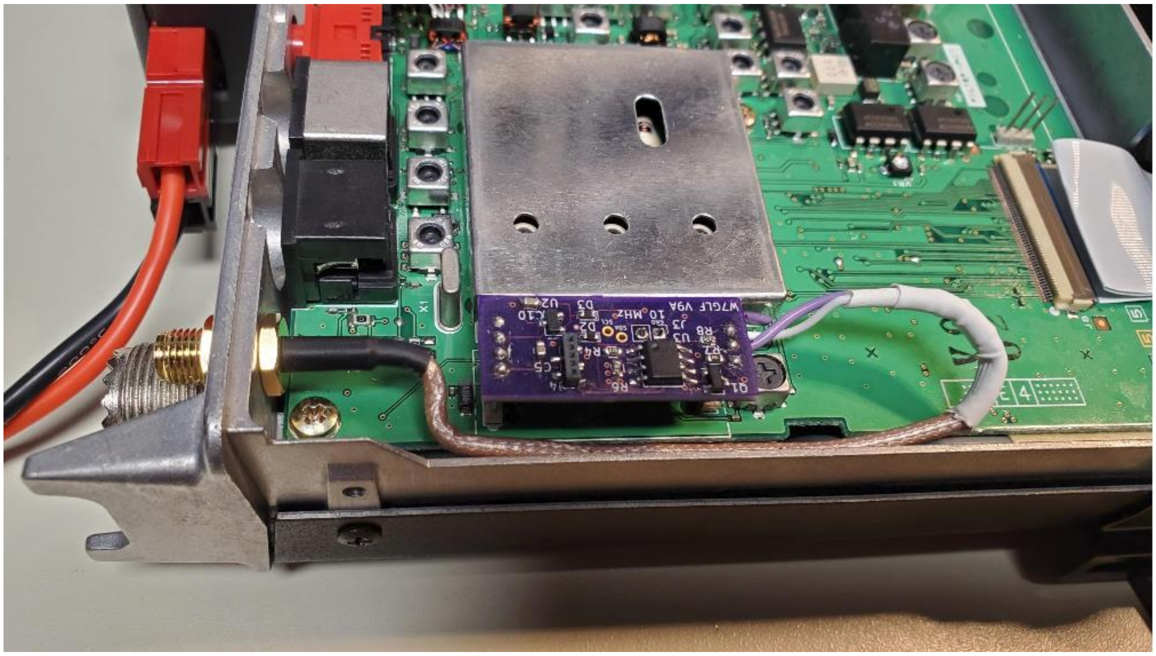


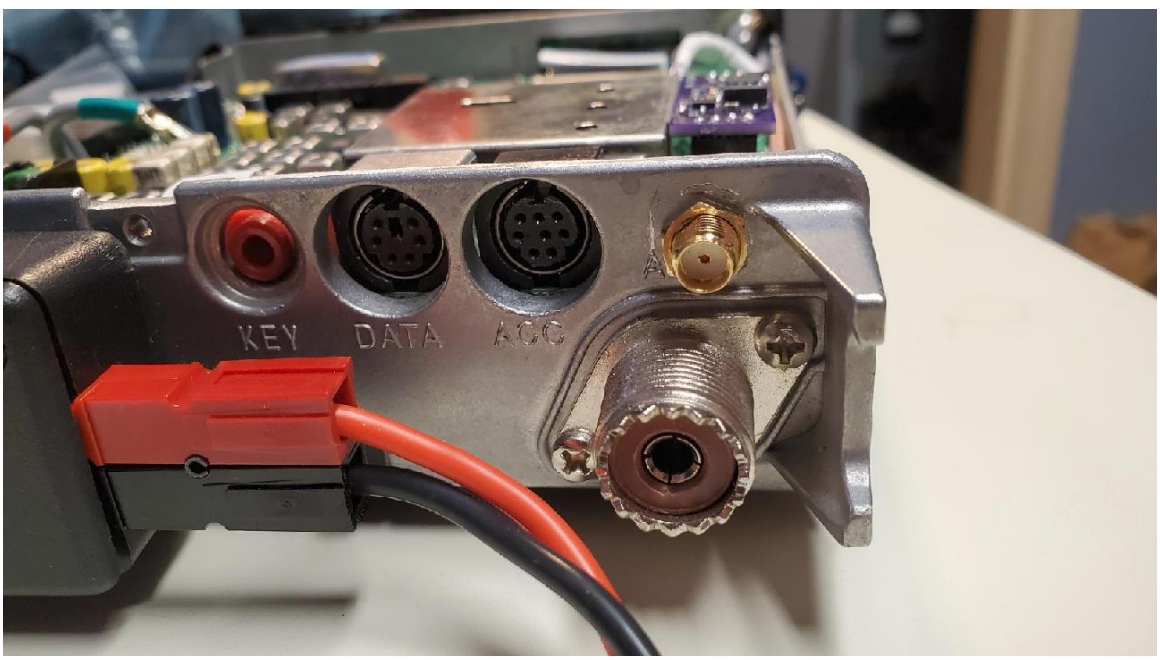
Twisted Pair of Wire Wrap Wires Wire Wrapped Around 2 Pin Connector



Connection Inside Radio

Here is an alternative installation done by K7MDL:






## 5. Specifications

Current Draw: 29 mA @ 5V

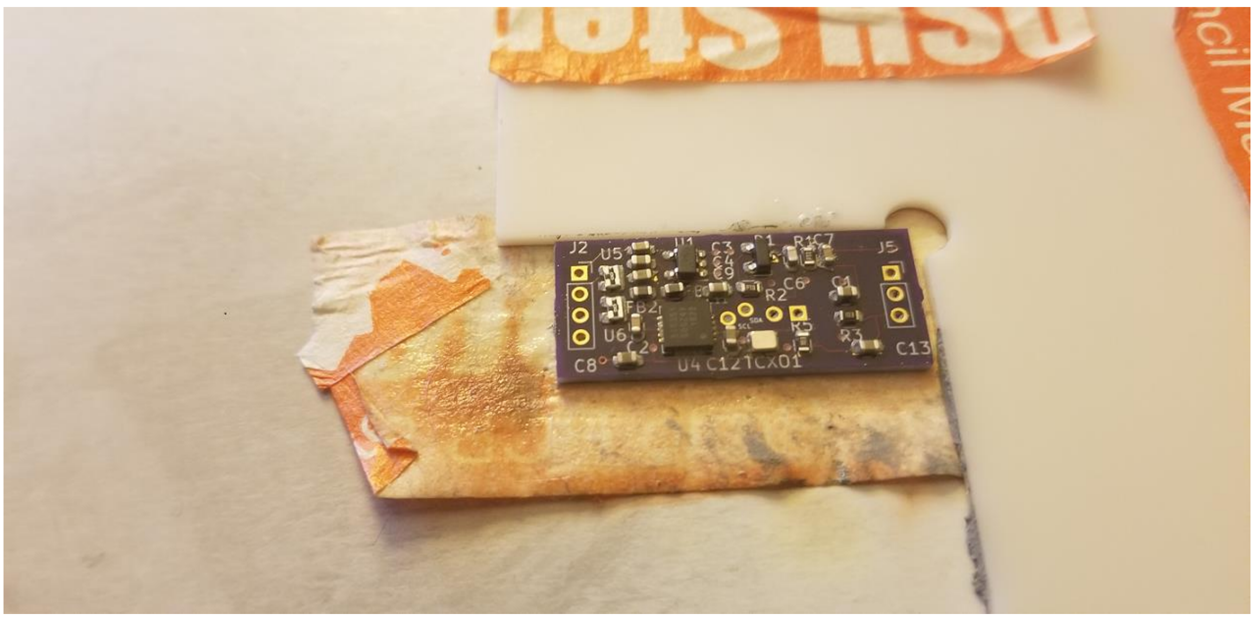
10 MHz Input Level: Minimum 0 dBm or 1.2 Vpp as measured at board connector

The voltage at the board connector is due to high impedance of board.  Note 0 dBm into a 50 ohm load from a 50 ohm source would be 632 mVpp – so same 50 ohm source into what is essentially an open load would be 1.264 Vpp).

## 6. Lessons Learned

I found doing reflow soldering is a bit tricky.  Do not want to put too much solder paste on the PC Board.  I started out with 5 mil stencil and finally ended up with 3 mil stencil.  I keep the solder paste in the fridge.  I remove it and give it about 20 minutes to warm up before applying it the board.

One of the tricks I learned was to build a frame to hold the circuit board while I was placing parts (see below).  I used one of the OSH Stencil pieces of tape upside down to form a sticky place to hold the board while I was placing parts.  I left the board there as I used the hot air tool for reflow the solder.  That solved the concern of moving the board while the parts were just pasted on.




Soldering Jig

When I made my first board I only reflow soldered the hardest parts: the two attenuators in the upper left of the board, the Si5351C and the TCXO.  My last two boards I got braver and placed all of the parts – that is why their alignment looks so good.  You have to be careful not to use too much airflow so you don't blow the parts off of the board.

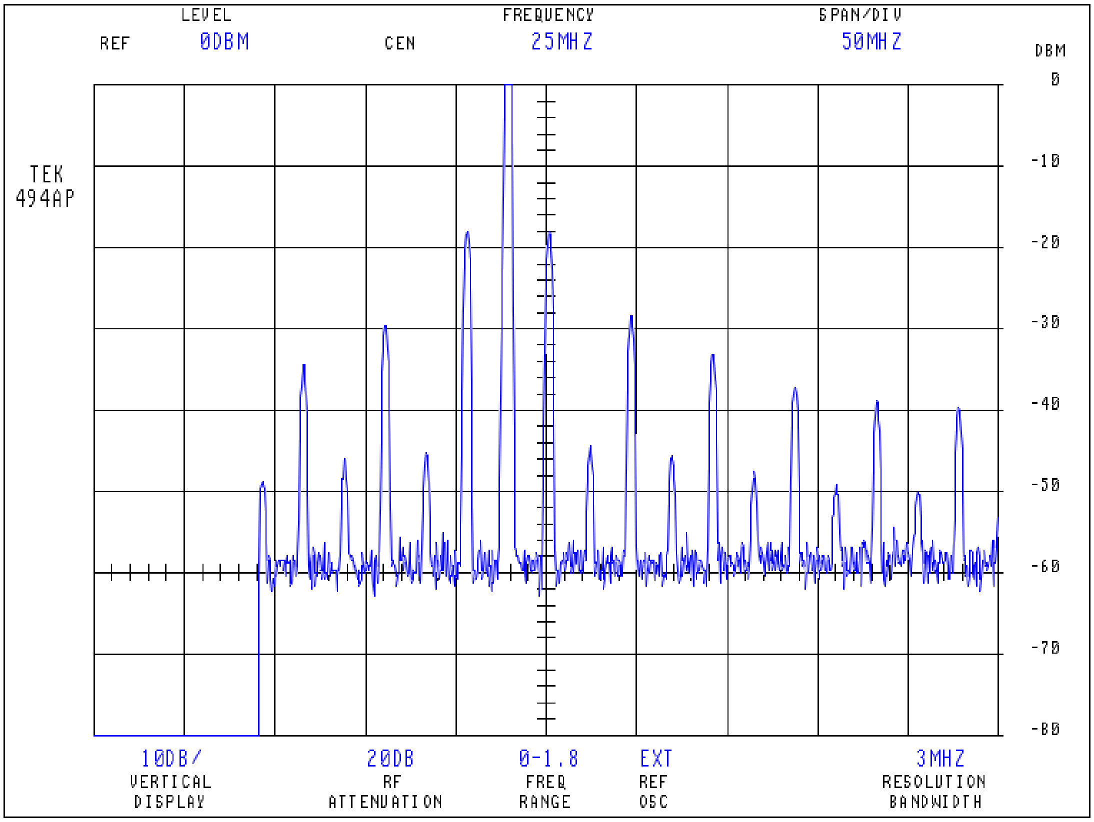
I hold the hot air reflow gun vertically above the board so the air flow is straight down and I set the temperature to about 380C and the air flow to only ¼ of the way up.  I preheat the board by holding the nozzle about 3 inches above the board for 120 seconds and then I slowly move the nozzle to about ½ to 1 inch above the board.  When the solder begins to melt I move the nozzle around over the entire board.  I use the smallest nozzle for my hot air rework station – 1/8 inch diameter.  I discovered that once the reflow starts all of the various parts reflow very quickly.  I assume that is because the PC Board itself has come up to close to the melting point of the solder paste.

I had a couple of instances where some of the pins of the Si5351C were not soldered.  It can be tricky especially if you have too much solder on the center pad.  Once I had to remove the chip, clean some of the solder off and then replace the chip.  Later times I made sure to use the 3 mil stencil and used the card that OSH Stencils sent and press firmly to get a thin layer of solder paste applied.

In my last case I determined that the I2C DAT and/or the I2C CLK pin were not connected because I could see the I2C traffic and the Si5351C keep returning NACK to the valid command stream being sent by the CPU.  The Si5351C was clearly not seeing the commands.  I used a Kester 186 Liquid Soldering Flux pen to flux all of the Si5351C connection pins and reheated the chip to reflow all of the solder connections for just that one chip.  That worked great!

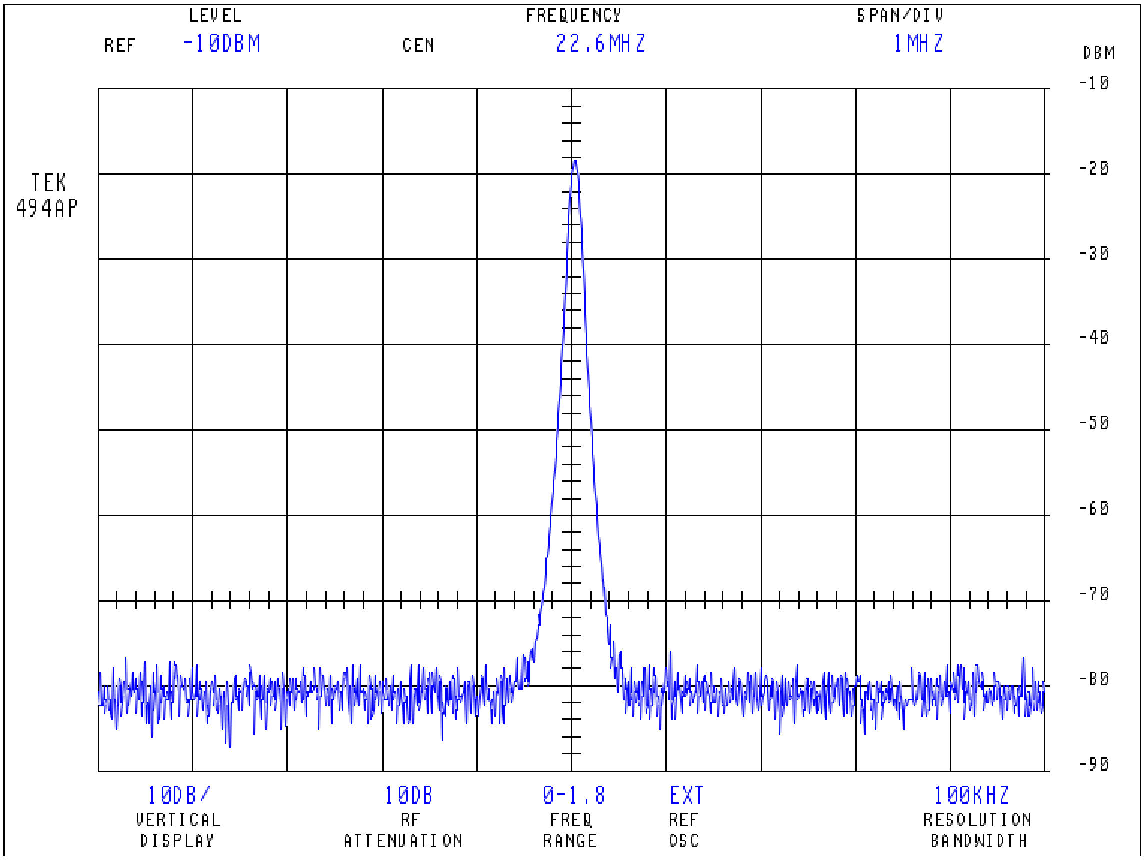
In order to watch the I2C traffic I used a KeeYees USB Logic Analyzer 24MHz 8CH 8 Channel UART IIC SPI from Amazon with the software Sigrok PulseView.

I did some testing of the output of the synthesizer as shown below.  The snapshot below shows a wide view from 0 Hz (the tallest peak) to 300 MHz.  The first peak is at 22.625.  The other peaks are typical of a square output and fall on the odd harmonics of 22.625.  The even harmonics are 25 dB down.  I did some testing to see if I could hear the frequency 46.25 when tuned at 1 MHz on my FT817 (which would be 46.25 – (2x22.625) MHz = 1 MHz).  I then tried 68.875 – (3x22.625) = 1 MHZ.  No signals from higher order harmonics were heard.




Wide Spectrum Output of Si5351C

The snapshot below shows a narrow view from 17.625 to 27.625 MHz. As shown there are no spurs visible above -60 dBm




Narrow Spectrum Output of Si5351C

At this point I have built 4 boards – all are version 9A of the board. The earlier versions were mostly just to save intermediate board designs.

## 7. Conclusion

I have enjoyed the challenge of learning new techniques of printed circuit board design and construction. This was a project that took about 8 months to come to fruition. I had several false starts which ended up taking too much board space so the real trick was deciding on how to minimize the size of the parts and the functionality required to achieve the goal. At first I was thinking of using an Arduino but that required too much board space. I also was planning on using MOSFETs to do the SCL and DAT I2C line switching between the onboard computer and an external connector but again it was taking too much board space. I decided if someone really wants to use an external processor to do the Si5351C programming they can remove the PIC12LF1552 from the board. If they are very careful they might be able to cut the SCL and DAT traces on the board and outboard their own MOSFET switches.

I also decided to restrict my SMD parts to be no smaller than 0603 (.06” by .03”) since my experiences with 0402 (.04” by .02”) have been mainly a source of frustration. Also as I have gotten older my hands are not as steady as they once were. When placing the parts I found I needed to use one hand to steady the other to get the 0603 parts on their pads. You can push the parts around a little but you need to be careful because any sliding reduces the amount of paste on pad. I found I ended up doing a little touch up on the solder joints on several parts. For that I used a small solder tip – a Hakko 900M-T-B (T18-I) 0.5mm (.02”) radius conical tip and small diameter .020” leaded Kester solder and some 1/16” (1.5 mm) solder braid to repair any spots on the board that did not properly reflow the solder.

Another trick is using the fine solder tip to swipe the sides of the Si5351C or to use either 2.54 mm (1/10”) or 1.5 mm (1/16”) SolderWick. If the wick is pre-coated with a tiny bit of solder you can press the solderwick lengthwise against a side of the Si5351C to get rid of solder bridges. I have used this technique before on multi legged SMD parts and it works very well.