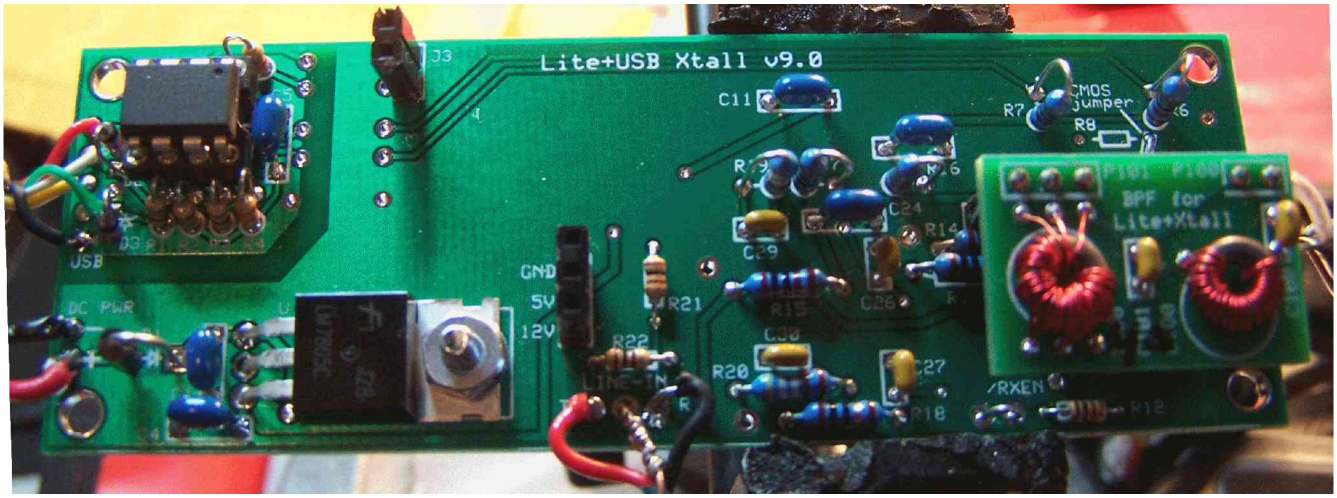
Enhanced Builder's Notes for Lite+Xtall RX V9.0

The WB5RVZ enhanced builders notes for the new V9.0 RX are now available on the WB5RVZ website.



RX Lite + USB Xtall V9.0 Kit - Home Page

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Introduction

This is the home page for the Detailed Builder's Notes for the Softrock "Lite + USB Xtall" V9.0 Software Defined Radio receiver, the latest in a series of SDR kits offered by Tony Parks KB9YIG.

The intent in providing these detailed instructions is to help the less experienced builder through what might otherwise be a daunting task. The instructions provide a stage-by-stage build process, allowing the builder to build a single stage and then test it ("sanity check") before moving on to the next stage.

Much of the documentation was initially developed entirely from the schematic and from the earlier V8.3/8.4 documentation. Over time, the author may post changes to the affected web pages, as necessary. You should check periodically to see if there have been any revisions, especially in the area of Stage-end tests. If your browser is caching pages, you may need to hit the "refresh" key (F5 on IE and Firefox) to get the latest version of the page.

Ordering Information

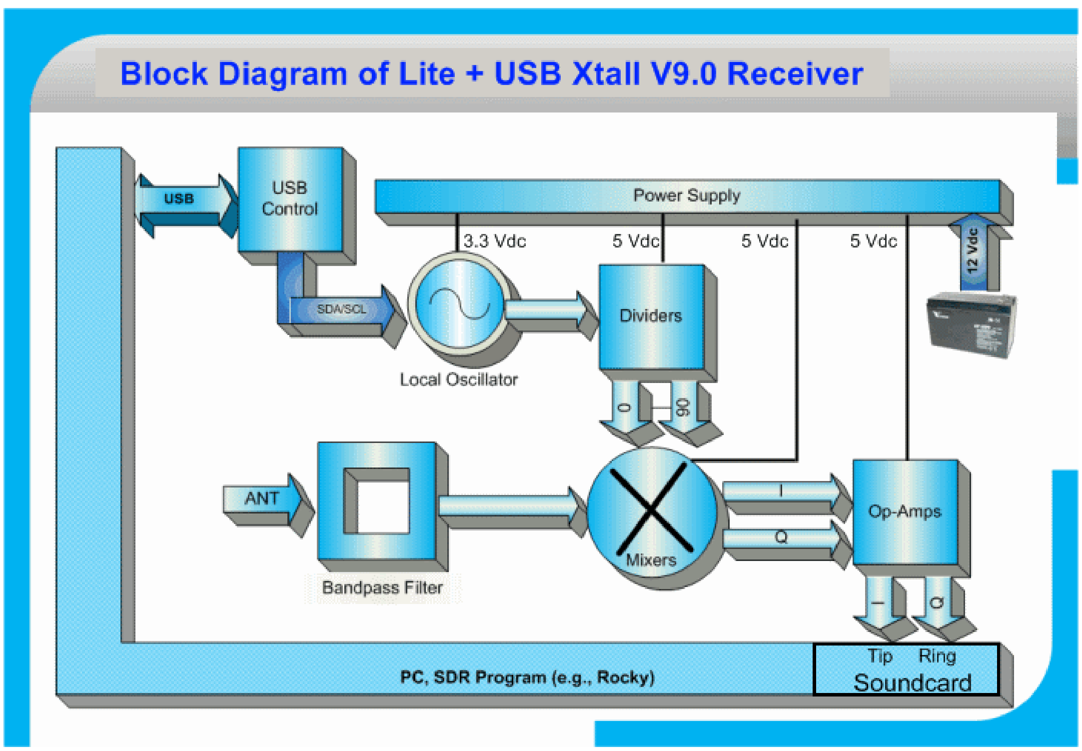
The kit price is $44 for US/Canada and $45 for DX where each kit price includes mailing cost. The kit prices include a CMOS Si570 and if a kit buyer wants the kit without the Si570 each kit price is reduced by $15.

Order via paypal to Tony using his KB9YIG email address at gmail.com.

Build Stages and Schematic

Construction Stages and Theory of Operation

(Click on a stage to view its detailed builders' notes)



Each stage will have a subset of the overall schematic diagram. Each sub-schematic is annotaded with clickable text to show "from/to" stages. The user can click on the text and link to the appropriate stage.

The schematics are annotated with red dots to designate the resistors' hairpin leads (or for a flat-mounted resistor, the left-hand or top lead).

To view the schematic diagram for the entire receiver, see Overall Schematic Diagram from Tony's original documentation.

For the more experienced builder, each stage has a "Summary Build Steps" section which outlines the sequencing of tasks within the stage and provides a link to the testing stage (bypassing the detailed installation notes).

For the most experienced builder, see the "Terse" builder's notes below.

For the rest os us, detail construction steps and tests are provided in each stage and will be highlighted by special Icons:

A step in the detailed build section

A test operation in the Testing section

A test operation requiring an HF transceiver.

An optional test using an oscilloscope.

Any comments or corrections should be directed to the author, Robby WB5RVZ, and would be most appreciated.

Theory of Operation

This kit incorporates the new SI570 programmable oscillator, along with a USB-coupled microcontroller that allows an SDR program on the PC to tune that oscillator to any desired"center" frequencies with which to drive the basic Softrock receiver.

The kit covers both types of SI570 oscillators: the CMOS version and the LVDS version. The only differences are with respect to two components, R25 and U8 (see the Bill of Materials) .

The receiver builds on the earlier Softrock RX-Lite receiver. The problem with that earlier receiver was - as "Softrock" implies - it was pretty much rockbound. For a given crystal value, you could receive one or two "bands", centered on a frequency that was one-fourth the crystal frequency. Depending upon the sound card used, the bandwidth of the receiver would be +/- 24 kHz around the center frequency or +/- 48 kHz around the center frequency, depending upon the sound card's sampling bandwidth.

In the block diagram above (which is almost identical to the block diagram for a simple direct conversion receiver), the Bandpass Filter, Mixer, and OpAmps stages are essentially the same as those on the RX-Lite. The Local Oscillator and Dividers stages replace the crystal-based LO and divider chain of the original. However, the essence remains:

- the local oscillator develops a signal that is a 4X multiple of the center frequency and the dividers bring the signal to the desired center frequency and into quadrature (90° phase difference between the two outputs of the chain).
- These 2 signals are fed to the Mixer stage, which down-converts the "chunk" of RF that is in the passband of the bandpass filters into 2 "chunks" of audio representing the difference between the incoming RF and the LO quadrature signals.
- These 2 AF signals are identical, and 90° out of phase with each other. They are amplified in the Op-Amps stage and fed into the PC's sound card to be digitized and processed.
- An essential part of that digital signal processing is using the quadrature streams to tease out the signals that are above the center frequency from those that are below the center ferequency, yielding a spectrum centered on the center frequency.

The real advance here is the use of a programmable oscillator (SI570) in the local oscillator circuit and a USB control circuit to program the Si570. (The USB control circuit also provides PTT switching outputs and keyer/stgraight key inputs for the SDR software running on the PC, but these are not needed for RX only operation). This setup allows the user to select any desired center frequency. This is a major advance over the Version 8.3 receiver, which only permitted switch selection of up to 16 pre-programmed center frequencies and leaves the rockbound RX Lite in the dust!

The other advance is changing the design of the bandpass filters to allow for removable filter boards (this radio uses the same BPFs as the Version 8.3 BPFs). Together these design changes add a multiband capability to the Softrock platform and open it up to follow-on designs that will provide even greater frequency agility.

See, also, the Detailed Theory of Operation discussion.

Bill of Materials

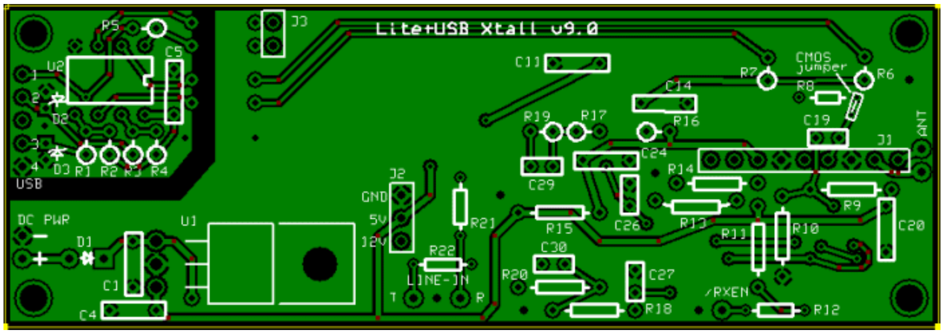
Each stage of construction will be preceded by a detailed bill of the materials for that stage, ordered in the sequence in which the different components are to be installed.

For reference and inventorying purposes, the overall bill of materials is provided in a separate "Bill of Materials" page.

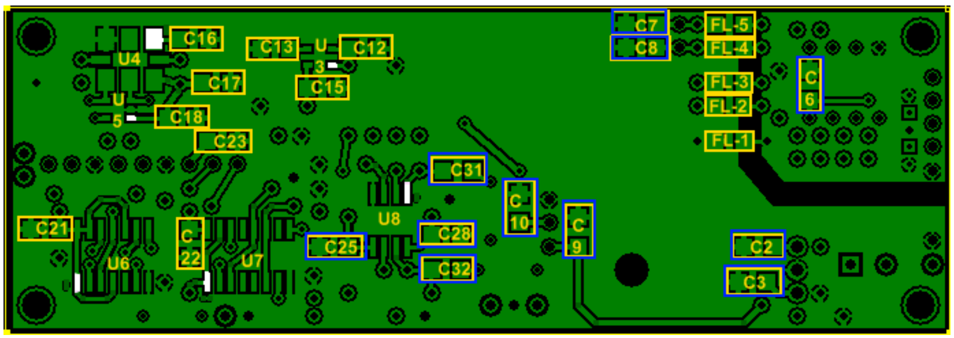
Terse Build Notes for the "Experts"

Tony's original build instructions for the expedited, "non-staged" approach are provided here for reference. If you plan to follow the staged construction approach outlined in this and subsequent pages, do not attempt to follow the steps listed below.

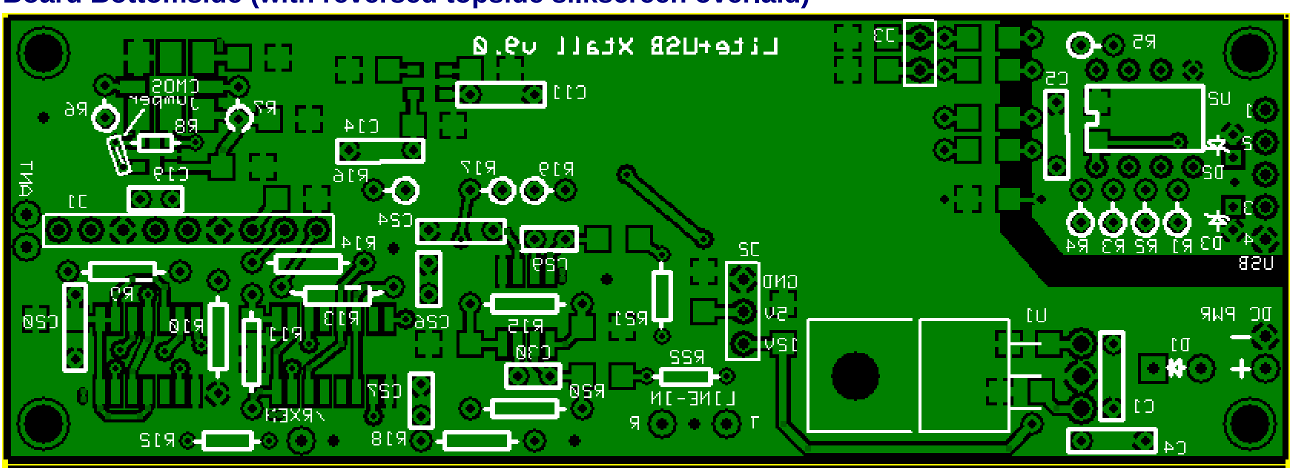
Board Topside



Board Bottomside



Board Bottomside (with reversed topside silkscreen overlaid)



Do not follow the summary approach below unless you are an accomplished builder and feel no need to build the kit using the staged approach.

Tasks for "Stuff-the-Board-and-Then-Test" approach (said Tom tersely)

- Install all SMT Capacitors to bottom of board (see graphic, above) , except for those around U3 (C12, C13m abd C15) and (if U5 is used) C18.
- Install Five pin SMT 3.3V Voltage Regulator, U3, to bottom of board.
  Make sure U3 leads are well-centered on their pads, then tack the IC in place by careful soldering of one lead.
  Apply heat to pin to reposition U3 and, when properly positioned, carefully solder the other leads.
  Use solder wick to remove excess solder or solder bridges between pins.
- If using LVDS version of Si570, install U5, Fin1002, on bottom of board in same careful fashion as U3. U5 is NOT installed if the Si570 is the CMOS version.
- Install remaining SMT capacitors (C12, C13, and C15)
- Install U4, Si570, on bottom of board with careful soldering as with U3. Note that there are 8 "pins" to be soldered.

- Install remaining SOIC SMT ICs to bottom of board (if an IC in the kit fits a location, it is the correct IC for that location).
- Mount 5V Voltage regulator U1, LM7805, to top of board with a 4-40 machine screw, #4 start lockwasher, and hex nut, attaching the tab of U1 to the board.
- Mount the 2 and 3 pin sockets, J3 and J2, in the J3 and J2 locations.
- Mount 9-pin socket J1 in its location.
- Install resistors R1-R22 on top of the board, with R1-R7, R16-R17, and R19mounted "hairpin" style. All other resistors are mounted flat.
  Note: R8 is NOT installed if the Si570 is the CMOS version
- Install capacitors to the top of the board in the locations shown on the silkscreen.
- Install the socket for U3 on top of board.
- Diodes D1-D3 on top of board in hairpin fashion with diode body above each round pad.
- Connect a shorting wire in the CMOS jumper location if the Si570 is the CMOS version.
- Connect a shorting wire between the /RXEN hole and the ground hole immediately to its left.
- Build desired band pass filter board(s) and mount on J1 such that the 3 header pins go to J1's left-most 3 pin sockets.

Detailed Build Notes

If you prefer to take the more methodical, "build a little, test a little, ..." staged approach to building this kit, this web site is for you. In the pages that follow this home page, you will find the notes for the construction and testing of each of the stages of the build.

The build will go through the following stages

- **Receipt and Inventory of the Kit using the Bill of Materials**
- **Build and test the Power Supplies (5Vdc and 3.3 Vdc)**
- **Build and test the USB Control Circuit**
- **Build and test the Local Oscillator**
- **Build and test the Divider STage**
- **Build and test the Operational Amplifier Stage**
- **Build and test the Mixer Stage**
- **Build and test the Bandpass Filter Board(s)**
- **Connect the completed board to the outside world.**

Each stage will Have the same basic sections:

- Introduction and theory of operation
- Schematic - a subset of the overall schematic diagram
- Bill of Materials - a build sequence ordered set of the items to be installed in the stage
- Summary Build Instructions - a summary of the steps in the stage
- Detailed Build Instructions - the step-by-step, detailed tasks of the build
- Testing - one or more tests that can be conducted to validate the built stage

Testing

Most of the tests specified in these pages can be accomplished with a moderately priced digital multimeter. Some tests using more sophisticated tools may be specified, but are not really essential to successfully building and testing this radio.

Measurements specified in the tests must be considered approximate and the tester should expect a fairly wide (+/- 1--20%) range of values around the specified values.

Tests will be identified by the following icons:

- **A test that requires no more than your DMM**

- **An optional test that may require HF transceiver to transmit into a dummy or receive through a loosely coupled antenna wire one or more test frequencies**
- **An optional test which requires an oscilloscope**

Background Info

Tools

Soldering

View above video example on Youtube

- Read the Primer on SMT Soldering
  at the Sparkfun site. It is a very good read and it speaks great truths. Then take the time to watch the video tutorial on soldering an SOIC SMD IC.
- For more general "how-tos" on soldering, Craig KB5UEJ highly recommends the videos at this site.
- "Splashover": Be careful when soldering SMT components to the bottom of the board. In some cases There are holes through which topside component leads must pass and which can easily get clogged with "solder splashover", where the hole is very close to an SMT pad.
- Solder Stations. Don't skimp here. Soldering deficiencies account for 80 percent of the problems uncovered in troubleshooting. It is preferable to have an ESD-safe station, with a grounded tip. A couple of good stations that are relatively inexpensive are:

  - Velleman VTSS5U 50W Solder Station (approx $20 at Frys)

Harbor Freight ESD Solder Station (under $50)

Electro Static Discharge (ESD) Protection

- Whenever you see the symbol on the left, this means to take ESD precautions:
- Avoid carpets in cool, dry areas.
- Leave PC cards and memory modules in their anti-static packaging until ready to be installed.
- Dissipate static electricity before handling any system components (PC cards, memory modules) by touching a grounded metal object, such as the system unit unpainted metal chassis.
- If possible, use antistatic devices, such as wrist straps and antistatic mats (see Radio Shack's Set for $25 or the JameCo AntiStatic mat for $15)).
- Always hold a PC card or memory module by its edges. Avoid touching the contacts and components on the component.
- Before removing chips from their insulator, put on the wrist strap connected to the ESD mat. All work with CMOS chips should be done with the wrist strap on.
- As an added precaution before first touching a chip, you should touch a finger to a grounded metal surface.
- If using a DMM, its outside should be in contact with the ground of the ESD mat, and both leads shorted to this ground before use.
- See the review of ESD Precautions at this link.

Work Area

- You will need a well-lit work area and a minimum of 3X magnification (the author uses a cheap magnifying flourescent light with a 3X lens. This is suplemented by a hand-held 10 X loupe - with light - for close-in inspection of solder joints and SMT installation.
- You should use a cookie sheet or baking pan (with four sides raised approximately a half an inch) for your actual work space. It is highly recommended for building on top of in order to catch stray parts, especially the tiny SMT chips which, once they are launched by an errant tweezer squeeze, are nigh on impossible to find if they are not caught on the cookie sheet.

Misc Tools

- It is most important to solidly clamp the PCB in a holder when soldering. A "third-hand" (e.g., Panavise or the Hendricks kits PCB Vise) can hold your board while soldering. In a pinch, you can get by with a simple third-hand, alligator clip vise. Jan G0BBL suggests "A very cheap way is to screw a Large Document Clip to a woodblock which will clamp the the side of a PCB."
- Magnifying Head Strap
- Tweezers (bent tip is preferable).
- Diagonal side cutters.
- Small, rounded jaw needle-nose pliers.
- Set of jewelers' screwdrivers
- An Exacto knife.
- Fine-grit emery paper.

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Softrock Lite + USB Xtall V9.0 Bill of Materials

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Introduction

Inventorying the Bill of Materials

The tables below provide a complete listing of the parts/components required for and provided with the Softrock Lite +Xtall V9.0 RX kit.

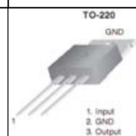
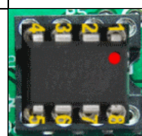
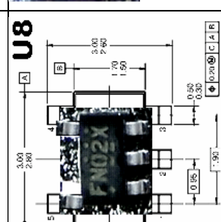
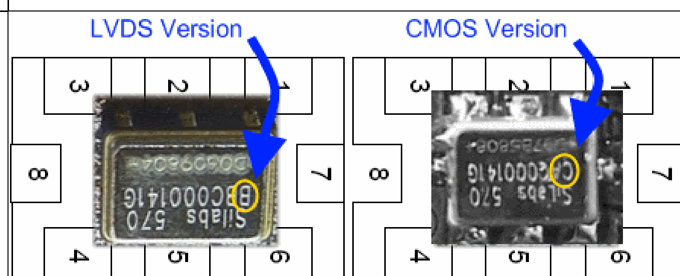
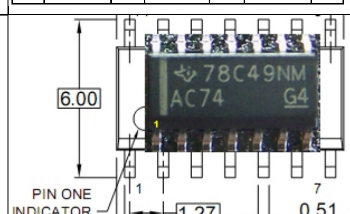
When inventorying the parts, you should be wary of relying upon your color vision alone to identify each resistor's value. Each resistor is depicted in the table by a graphic showing the color code (most are 5-band codes; some are 4-band). However, monitor tolerances/calibration can cause the display of colors different than those intended in the original web page design. You should always use an ohmmeter to validate your decoding of the resistor's value.

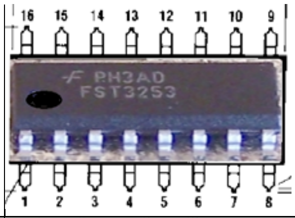
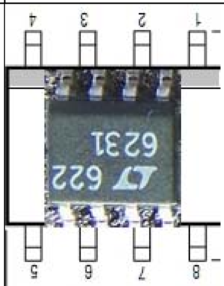
Schematic

Bill of Materials

Designation	Value	Color/Code	Orientation	Category	Notes
BPF-1					160m board
BPF-2					80/40m board
BPF-3					30/20/17m board
BPF-4					15/12/10m board
V9.0					Main board
C19	0.01uF	103		ceramic	
C100-3	180 pF	181		ceramic	
C101-3	220 pF	221		ceramic	
C29	220pF	221		ceramic	
C30	220pF	221		ceramic	
C101-4	330 pF	331		ceramic	
C100-1	390 pF	391		ceramic	
C26	0.047uF	473		ceramic	
C27	0.047uF	473		ceramic	
C01	4.7uF	475		ceramic	
C04	4.7uF	475		ceramic	
C05	4.7uF	475		ceramic	
C11	4.7uF	475		ceramic	
C14	4.7uF	475		ceramic	
C20	4.7uF	475		ceramic	
C24	4.7uF	475		ceramic	
C100-2	560 pF	561		ceramic	
C101-1	5600 pF	562		ceramic	
C101-2	680 pF	681		ceramic	
C100-4	82 pF	82		ceramic	
C12	0.01uF			SMT 1206	
C13	0.01uF			SMT 1206	
C15	0.01uF			SMT 1206	
C16	0.01uF			SMT 1206	
C17	0.01uF			SMT 1206	
C18	0.01uF			SMT 1206	

C21	0.01uF			SMT 1206	
C22	0.01uF			SMT 1206	
C23	0.01uF			SMT 1206	
C02	0.1uF			SMT 1206	black marked strip
C03	0.1uF			SMT 1206	black marked strip
C06	0.1uF			SMT 1206	black marked strip
C07	0.1uF			SMT 1206	black marked strip
C08	0.1uF			SMT 1206	black marked strip
C09	0.1uF			SMT 1206	black marked strip
C10	0.1uF			SMT 1206	black marked strip
C25	0.1uF			SMT 1206	black marked strip
C28	0.1uF			SMT 1206	black marked strip
C31	0.1uF			SMT 1206	black marked strip
C32	0.1uF			SMT 1206	black marked strip
J3	2 pin header				
P100-1	2 pin header				
P100-2	2 pin header				
P100-3	2 pin header				
P100-4	2 pin header				
J2	3 pin socket				
P101-1	3 pin header				
P101-2	3 pin header				
P101-3	3 pin header				
P101-4	3 pin header				
J1	9 pin socket				
cable1	USB				4 wire shielded USB cable with USB male on 1 end
D1	1N4003		W-E		
D2	1N5227B,3.6v		N-S		do NOT use
D3	1N5227B,3.6v		S-N		do NOT use
D2	BZY55,3.6v		N-S		Use this. See "Update" note in USB stage Introduction

					Section, above.
D3	BZY55,3.6v		S-N		Use this. See "Update" note in USB stage Introduction Section, above.
FL1	ferrite filter	grey		SMT 1206	
FL2	ferrite filter	grey		SMT 1206	
FL3	ferrite filter	grey		SMT 1206	
FL4	ferrite filter	grey		SMT 1206	
FL5	ferrite filter	grey		SMT 1206	
U1	LM7805				
U2	ATTiny45-20PU				with socket
U3	LP2992AIM5-3.3V	LFEA		SMT	marked "LFEA"
U5	FIN1002	FN02X		SMT	only required for LVDS version of Si570. See Tony Park' 6 Nov 2008 message.
U4	Si570 (CMOS version of device)			SMT	
U6	74AC74			SMT	

U7	FST3253			SMT	
U8	LT6231CS8			SMT	
T100-4	0.13 uH	T25-6 (yellow)			7T/4T bifilar #30
L100-4	0.53 uH	T25-6 (yellow)			14T #30
T100-3	0.6 uH	T25-6 (yellow)			14T/7T bifilar #30
L100-3	0.78 uH	T25-6 (yellow)			17T #30
T100-2	1.2 uH	T25-2 (red)			18T/9T bifilar #30
T100-1	1.4 uH	T30-2 (red)			18T/9T bifilar #30
L100-2	1.6 uH	T25-2 (red)			22T #30
L100-1	18.7 uH	T30-2 (red)			66T #30
R13	10		flat-h		
R14	10		flat-h		
R17	10		W-E		
R18	10		flat-h		
R01	68		S-N		
R03	68		S-N		
R08	100		flat-h		omit for CMOS
R12	100		flat-h		
R21	100		flat-v		
R22	100		flat-h		
R09	10k		flat-h		
R10	10k		flat-v		
R11	10k		flat-v		
R06	1k		S-N		
R07	1k		S-N		
R15	1k		flat-h		
R16	1k		W-E		
R02	1M		S-N		
R04	2.21k		S-N		
R05	4.7k		E-W		
R19	4.99k		E-W		
R20	4.99k		flat-h		
4-40 3/8in mach screw					
4-40 3/8in mach screw					
4-40 3/8in mach screw					
4-40 3/8in mach screw					

4-40 3/8in mach screw					
4-40 hex nut					
4-40 hex nut					
4-40 hex nut					
4-40 hex nut					
4-40 hex nut					
#4 star lock washer					
#4 1/8in nylon spacer					
#4 1/8in nylon spacer					
#4 1/8in nylon spacer					
#4 1/8in nylon spacer					
#4 nylon washer					
#4 nylon washer					
#4 nylon washer					
#4 nylon washer					

Softrock Lite + USB Xtall V9.0 Power Supply Stage

Home BOM USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Introduction

Theory of Operation

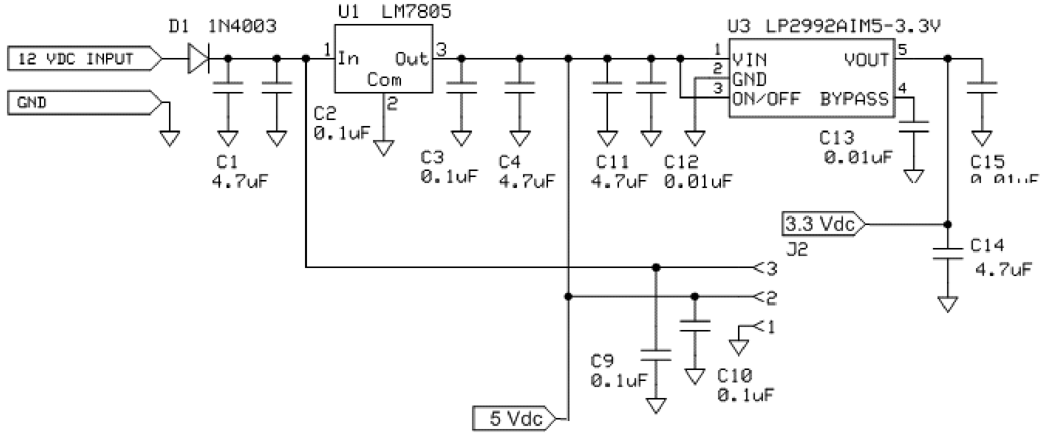
This stage provides the two power rails for the radio:

- a regulated +3.3 Vdc for the local oscillator stage
- a regulated +5 Vdc for the divider, mixer, and opamp stages

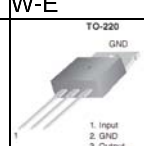
Note that the USB stage is powered from the PC's USB port's +5Vdc

Schematic

This is a subset of the overall schematic.



Bill of Materials

Designation	Value	Color/Code	Orientation	Category	Notes
U3	LP2992AIM5-3.3V	LFEA		SMT	marked "LFEA"
C12	0.01uF			SMT 1206	
C13	0.01uF			SMT 1206	
C15	0.01uF			SMT 1206	
C02	0.1uF			SMT 1206	black marked strip
C03	0.1uF			SMT 1206	black marked strip
C09	0.1uF			SMT 1206	black marked strip
C10	0.1uF			SMT 1206	black marked strip
C01	4.7uF	475		ceramic	
C04	4.7uF	475		ceramic	
C11	4.7uF	475		ceramic	
C14	4.7uF	475		ceramic	
D1	1N4003		W-E		
U1	LM7805				
J2	3 pin				

Summary Build Notes

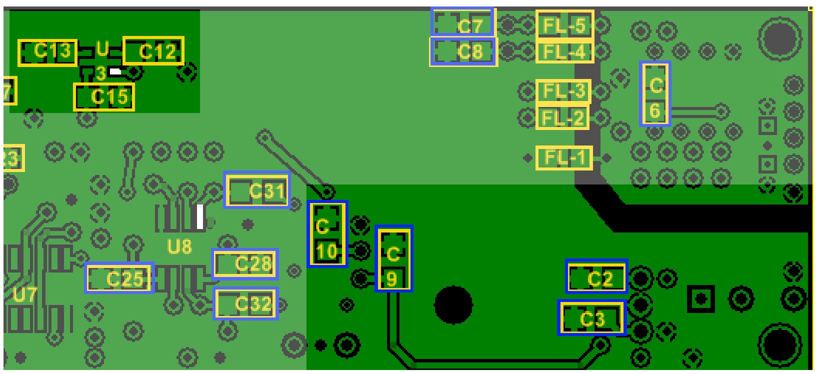
- Install SMT ICs and Capacitors (bottom)
- Install ceramic capacitors (top)
- Install D1 (top)
- Install U1 (top)
- Install power bus jack, J2
- Test the Stage

Detailed Build Notes

Bottom of the Board

The challenge here is the extremely tiny 3.3V regulator, U3. Be very careful tweezing this component, as, if it ever gets launched into space, it will be nigh on impossible to find.

Also be careful to note the two different types of SMT caps. There are three 0.01 uF caps and four 0.1 uF caps, the latter being identified by a black stripe drawn on the plastic carrier strip.



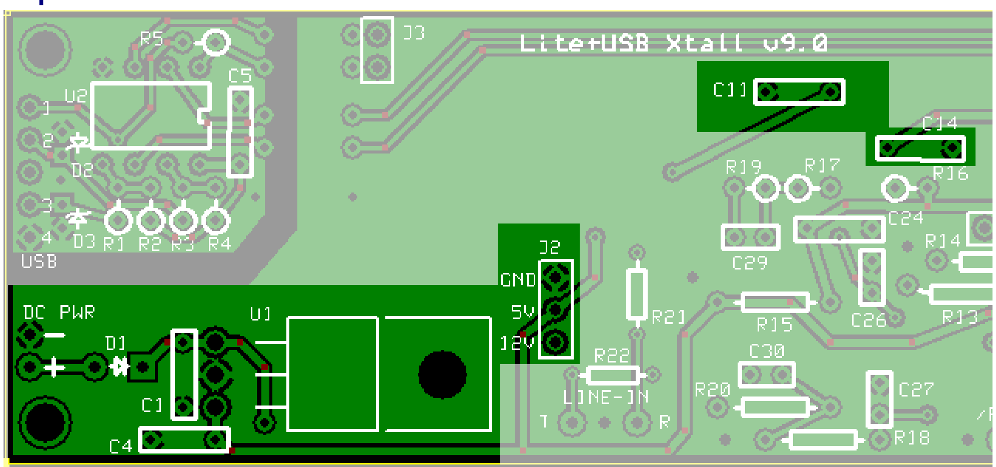
Install SMT ICs and Caps

Do not confuse U3, marked "LFEA", with the FIN 1002 (U5) marked "FN02X"

Designation	Value	Color/Code	Orientation	Category	Notes
U3	LP2992AIM5-3.3V	LFEA		SMT SOT-23	**Install Five pin SMT 3.3V Voltage Regulator, U3, to bottom of board. Make sure U3 leads are well-centered on their pads, then tack the IC in place by careful soldering of one lead. Apply heat to pin to reposition U3 and, when properly positioned, carefully solder the other leads. Use solder wick to remove**

					excess solder or solder bridges between pins
C12	0.01uF			SMT 1206	
C13	0.01uF			SMT 1206	
C15	0.01uF			SMT 1206	
C02	0.1uF			SMT 1206	black marked strip
C03	0.1uF			SMT 1206	black marked strip
C09	0.1uF			SMT 1206	black marked strip
C10	0.1uF			SMT 1206	black marked strip

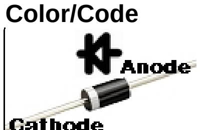
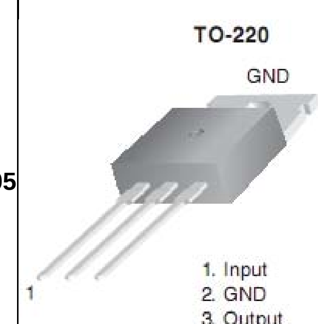
Top of the Board



Install Ceramic Caps

Designation	Value	Color/Code	Orientation	Category	Notes
C01	4.7uF	475		ceramic	
C04	4.7uF	475		ceramic	
C11	4.7uF	475		ceramic	
C14	4.7uF	475		ceramic	

Install Diode and U1

Designation	Value	Color/Code	Orientation	Category	Notes
D1	1N4003		W-E		Hairpin style with the hairpin on the cathode lead
U1	LM7805				Mount 5V Voltage regulator U1, LM7805, to top of board with a 4-40 machine screw, #4 start lockwasher, and hex

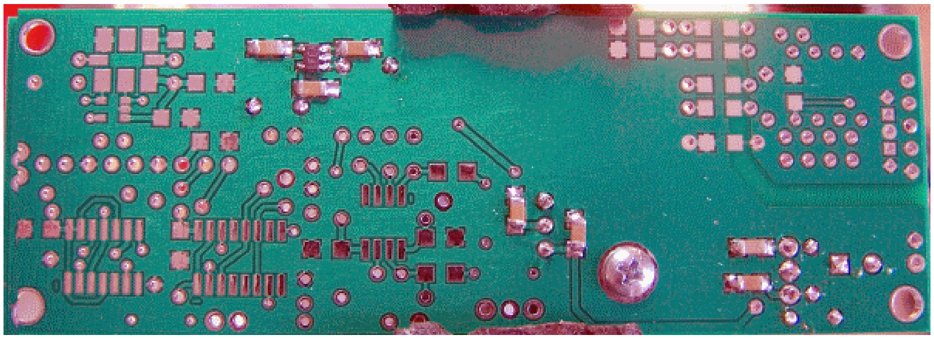
						nut, attaching the tab of U1 to the board

Install Power Bus Jack J2

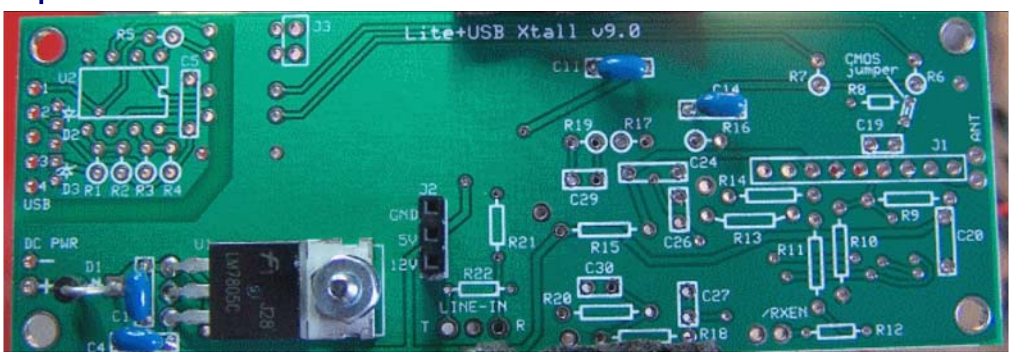
Designation	Value	Color/Code	Orientation	Category	Notes
J2	3 pin				

Completed Stage Photos

BottomSide



Topside



Testing

Current Draw

Test Setup

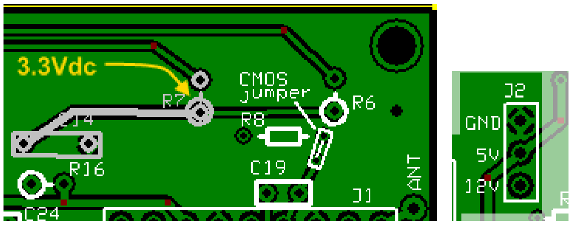
- To power the v9.0 receiver you will need a 9 volt to 12 volt DC source at a little over 100 mA. A supply that is free of ground connnections works best.
- Before you power the board up for the first time, connect a mA meter in series with the power lead and to be safe, put a 1k ohm resistor in series with the power lead. This can be in either the + or - line. This will limit the current flow to <=12 mA if you have a short on the board.
- After you see that the current isn't excessive, remove it, and re-measure the current draw.
- The current draw with this initial stage and no other loads should be < 5 mA

Test Measurements

Testpoint	Nominal Value	Author's	Yours
Current draw thru 1 k limiting resistor	3-5 mA	3.1 mA	____________
Current draw without limiting resistor	3-5 mA	3.2 mA	____________

Voltages

Test Setup



- **Power up the board with 12 Vdc**
- **Using a DMM, measure the voltages with respect to ground (ground = J2 pin 1).**
- **The 12 volt rail should be be approximately 12 volts DC. It should show a voltage drop from the power source on the order of .5 to .7 Vdc, representing the effect of D1's ohmic resistance in the circuit.**

Test Measurements

Testpoint	Nominal Value	Author's	Yours
3.3 V Rail: R7 body hole (see above)	3.3 Vdc	3.29	____________
5 V Rail: J2-Pin2	5 Vdc	4.97 Vdc	____________
12 V Rail: J2-Pin3	12 Vdc	11.4 Vdc	____________

Softrock Lite + USB Xtall V9.0 USB Control Stage

Home BOM Power Supply Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Introduction

Theory of Operation

This stage provides the control interface between the hardware SDR and a PC that is running the appropriate SDR software which can provide I^2C (**I**nter **I**ntegrated **C**ircuit) bidirectional control signals over a USB connection. There are two lines in the I2C bus: the clock (SCL) and the date (SDA). For a more in-depth discussion of the I^2C protocol, see the formal Specification.

The heart of the control circuit is U3, an ATTiny45-20PU 8 bit AVR Microcontroller (caution! the pdf for this device is over 4MB). The unit is powered off of the PC's USB 5 Vdc bus and provides a 6 bit bi-directional I/O port. It draws less than 10mA. The zener diodes in the schematic help ensure that the USB data lines (D+ and D-) are at 3.3V only.

U3 uses the AVR firmware by DG8SAQ to perform the following functions:

- Accept control signals via the pins 2 and 3 of the Universal Serial Bus (USB)
- Translate input signals into I^2C control signals (bi-directional SDA and input-only SCL lines) for the Si570 programmable Oscillator
- Translate output I^2C signals from the Si570 back to USB signals to the PC
- Translate incoming bandswitching commands into appropriate signals to J3 (for control of the new HF-BPF board.
- Future: new firmware for U3 is being developed to implement automatic band switching of the new HF-BPF switchable BPF.

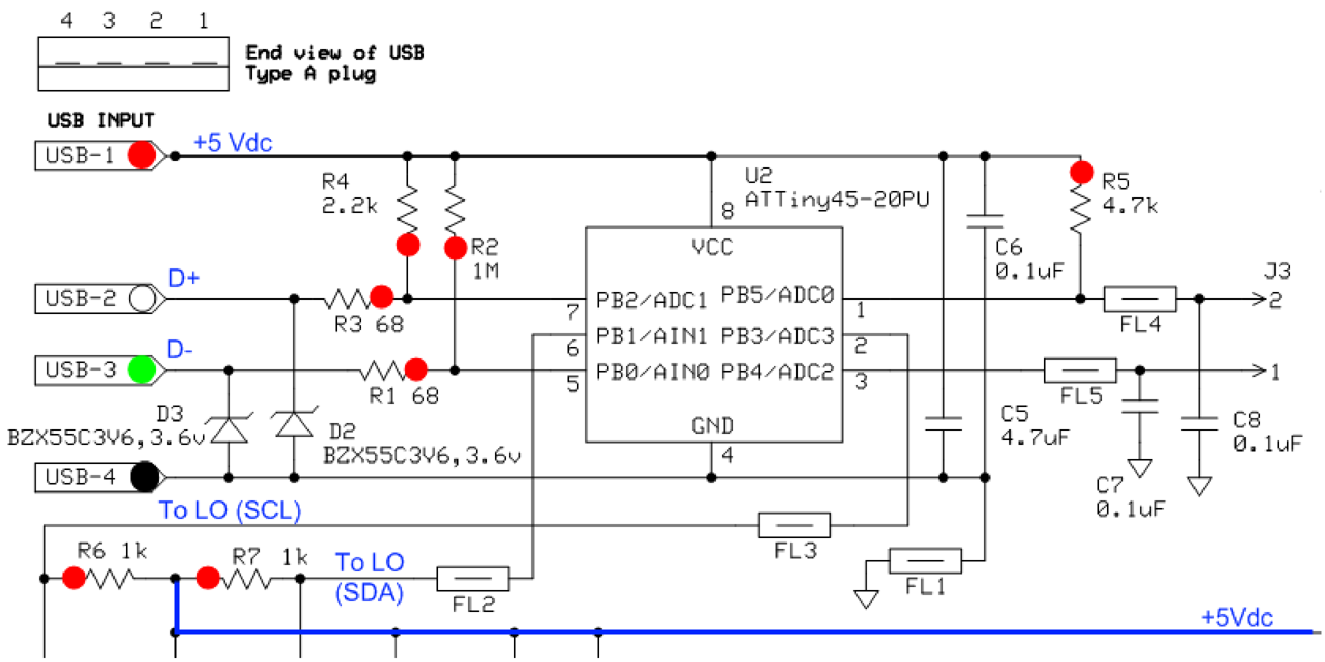
Update

Several builders have experienced issues with the voltages on the USB-2 and USB-3 lines and diodes D2 and D3. Following a long series of messages on the Softrock Yahoo Group, Jan G0BBL and Tony KB9YIG have decided to address the issue as follows:

- Each new kit will be packed with two each of the BZY55 3.3 volt and 3.6 volt zener diodes in place of the two 1N5227B zener diodes for the D2 and D3 locations.
- Builders are advised to try the 3.6 volts BZY55 zener pair for D2 and D3 first, (marked on the glass body with 3V6), and if they still have USB communications reliability problems then go to the 3.3 volt BZY55 zener pair.

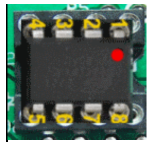
Schematic

This is a subset of the overall schematic. Note: red dot indicates resistor testpoints (hairpin, top, or left-hand lead)



Bill of Materials

Designation	Value	Color/Code	Orientation	Category	Notes
C06	0.1uF			SMT 1206	black marked strip
C07	0.1uF			SMT 1206	black marked strip
C08	0.1uF			SMT 1206	black marked strip
FL1	ferrite filter			SMT 1206	
FL2	ferrite filter			SMT 1206	
FL3	ferrite filter			SMT 1206	
FL4	ferrite filter			SMT 1206	
FL5	ferrite filter			SMT 1206	
D2	1N5227B,3.6v		N-S		do not use
D3	1N5227B,3.6v		S-N		do not use
D2	BZY55,3.3v		N-S		See "Update" note in Introduction Section, above.
D3	BZY55,3.3v		S-N		See "Update" note in Introduction Section, above.
D2	BZY55,3.6v		N-S		Use this. See "Update" note in Introduction Section, above.
D3	BZY55,3.6v		S-N		Use this. See "Update" note in Introduction Section, above.
C05	4.7uF	475		ceramic	
R01	68 1/6W		S-N		
R03	68 1/6W		S-N		
R05	4.7k 1/6W		E-W		
R02	1M 1/6W		S-N		
R04	2.2k 1/6W		S-N		
R06	1k		S-N		
R07	1k		S-N		

U2	ATTiny45-20PU				with socket
J3	2 pin header				
cable1	USB				4 wire shielded USB cable with USB male on 1 end

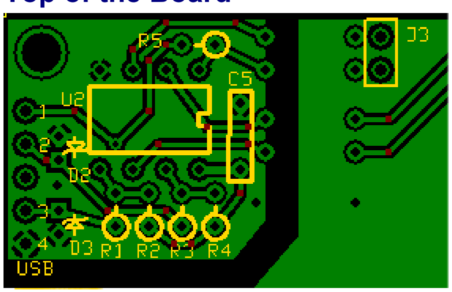
Summary Build Notes

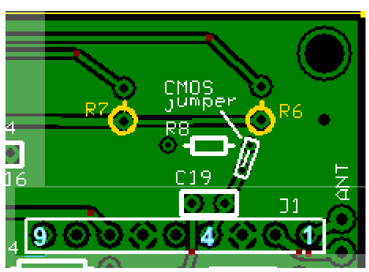
- Install Zener diodes D2 and D3 (top)
- Install resistors and C5 ceramic cap (top)
- Install U2 and socket (top)
- Install J3 (top)
- Install and connect USB cable (top)
- Test the Stage
- Install SMT capacitors and ferrite filters (bottom)

Detailed Build Notes

In other stages, we prefer to begin with the bottom side of the board. However, in this stage, the thru-holes for the topside coponents are very close together and are just begging to get solder splashed into them if we were to install the SMT components first. Thus, in this stage, we have reversed the bottom-then-top sequence.

Top of the Board





Install Zener Diodes

Several builders have experienced issues with the voltages on the USB-2 and USB-3 lines. Jan G0BBL and Tony KB9YIG have decided to address the issue as follows:

- **Each new kit will be packed with two each of the BZY55 3.3 volt and 3.6 volt zener diodes in place of the two 1N5227B zener diodes for the D2 and D3 locations.**
- **Builders are advised to try the 3.6 volts BZY55 zener pair for D2 and D3 first, (marked on the glass body with 3V6), and if they still have USB communications reliability problems then go to the 3.3 volt BZY55 zener pair.**

The two zener diodes are mounted hairpin style, with the cathode (banded) lead forming the hairpin.

Designation	Value	Color/Code	Orientation	Category	Notes

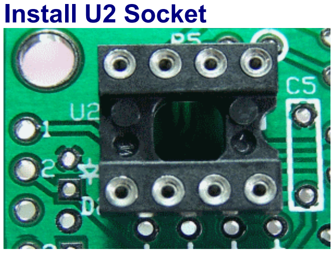
D2	BZY55,3.6v	N-S		Use this. See "Update" note in Introduction Section, above.
D3	BZY55,3.6v	S-N		Use this. See "Update" note in Introduction Section, above.

Install Resistors and ceramic capacitor C5

Note: 1/6W resistors are used due to the tight spacing of the resistors on the board.

Designation	Value	Color/Code	Orientation	Category	Notes
C05	4.7uF	475		ceramic	
R01	68 1/6W		S-N		
R02	1M 1/6W		S-N		
R03	68 1/6W		S-N		
R04	2.2k 1/6W		S-N		
R05	4.7k 1/6W		E-W		
R06	1k		S-N		
R07	1k		S-N		

Install U2 Socket



Install the socket for U2. Note the orientation on the notch, which should face eastward on the board

Install J3

J3 is reserved for a future use to provide control signals to the new HF_BPF (electronically switched bandpass fiters) kit.

Designation	Value	Color/Code	Orientation	Category	Notes
J3	2 pin header				

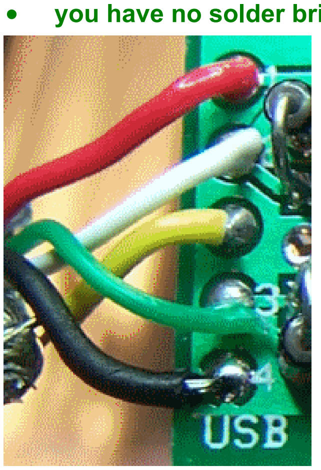
Install USB Cable

Solder a piece of hookup wire to the cable's shielding (see yellow lead in photo below) to serve as a strain-relief for the cable. Solder the strain-relief into the hole between leads 2 and 3.

Solder the USB cable leads in the order of red, white, green, and black to holes marked, respectively, 1, 2, 3, and 4

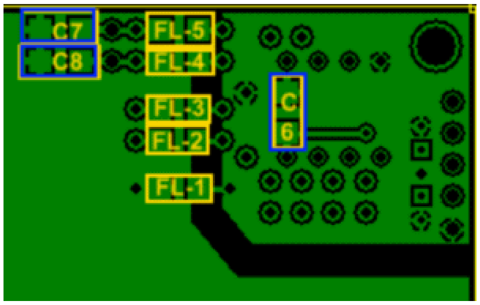
After soldering, carefully check (with good lighting and magnification) to ensure:

- **you have not accidentally switched the wires from the sequence shown above (red, white, yellow, green, black**
- **you have no solder bridges on any of the connections or across to either of the two zener diodes.**



Designation	Value	Color/Code	Orientation	Category	Notes

Bottom of the Board



Install SMT Capacitors

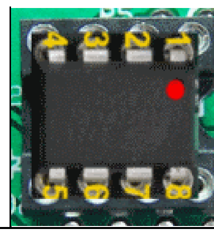
Designation	Value	Color/Code	Orientation	Category	Notes
C06	0.1uF			SMT 1206	black marked strip
C07	0.1uF			SMT 1206	black marked strip
C08	0.1uF			SMT 1206	black marked strip

Install SMT Ferrite Filters

Designation	Value	Color/Code	Orientation	Category	Notes
FL1	ferrite filter	grey		SMT 1206	
FL2	ferrite filter	grey		SMT 1206	
FL3	ferrite filter	grey		SMT 1206	
FL4	ferrite filter	grey		SMT 1206	
FL5	ferrite filter	grey		SMT 1206	

Plug in U2

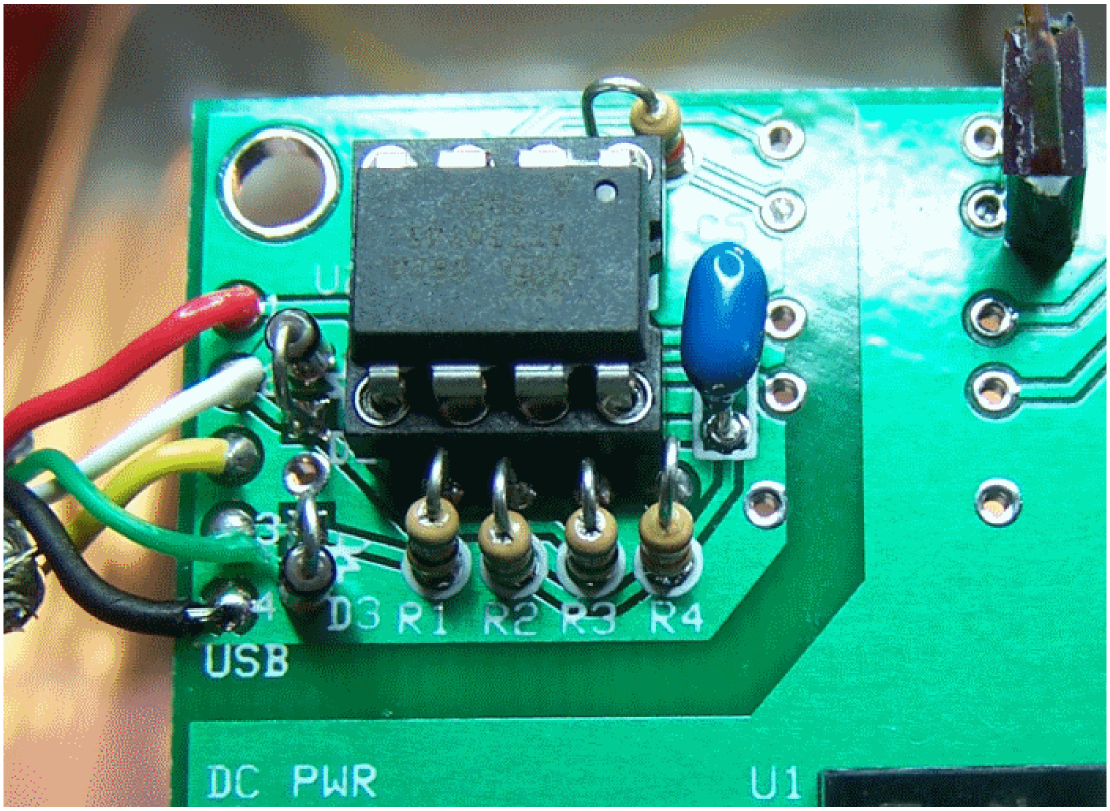
Note orientation - pin 1 is designated by the dimple/dot (see below)

Designation	Value	Color/Code	Orientation	Category	Notes
U2	ATTiny45-20PU				with socket

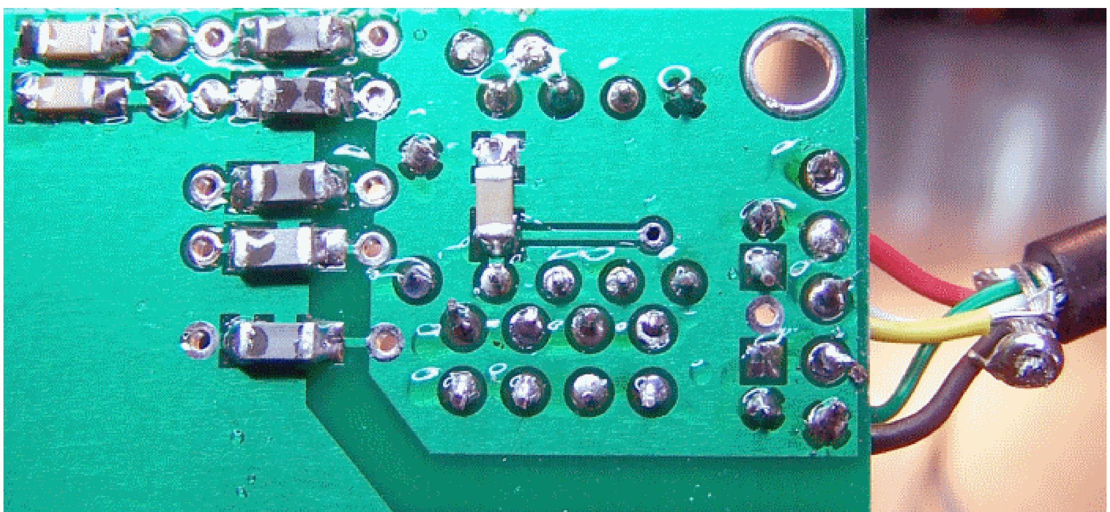
Completed Stage

Topside

(Note: resistors R6 and R7 not shown below - see completed topside picture of next (LO) stage)



Bottomside



Testing

Resistances

Test Setup

Make sure the USB cable is NOT connected to the PC

Test Measurements

Testpoint	Nominal Value	Author's	Yours
R5 hairpin to ground	~∞ Ω	starts ~34 MΩ (increasing to ∞)	_______Ω
R5 hairpin to USB-2	2.268 kΩ	2.269 kΩ	_______Ω
R5 hairpin to USB-3	~1 MΩ	990 kΩ	_______Ω

Voltages

Test Setup

If the resistance tests are successful, plug in the USB cable to the PC USB port and test the voltages

Voltage Test Measurements

(actual values may be +/- 10% of nominal values)

Testpoint	Nominal Value	Author's	Yours
R5 hairpin (U2-8) to ground	5 Vdc	4.96 Vdc	________
R1 hairpin (U2-5) to ground	60-100 mVdc	96 mVdc	________
R3 hairpin (U2-7) to ground	2.5-3.0 Vdc	2.54 Vdc	________

USB Polling - Courtesy of JAN G0BBL

Test Setup

Plug in the USB cable to the PC USB port

Test

The USB polling can be checked. Connect a short piece of wire to R3 hairpin and an audio tone should be audible as a S9 Plus signal on a RX in AM Mode tuned to about 1100 KHz in the AM Band (Medium Wave Broadcast band)

Test End

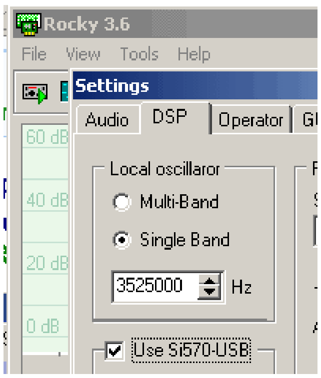
Unplug the USB cable from the PC

Functional Test Setup

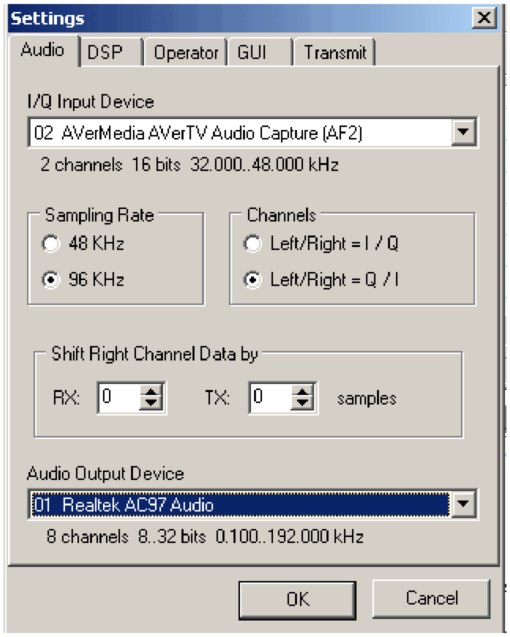
The functional testing of this kit assumes you will be using the Rocky SDR program to control your Local Oscillator and set your center frequencies.

The following steps outline how to set up Rocky for this (and later) tests:

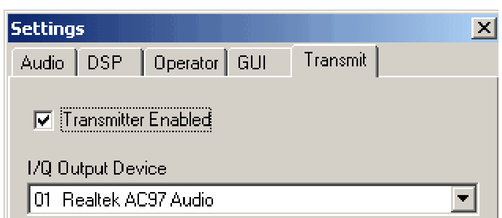
- Download Rocky V3.6 and install it
- Download the USB Interface zip file to a directory of your choice
- The driver files are located in the `Si570\AVR-USB-Driver` folder inside the zip. Extract them into a new directory, and connect the RX board's USB cable to a USB port. When prompted for the driver location, navigate to the extracted files and let windows install the drivers.
- Once the driver is installed, enable the Si570 support in Rocky by ticking the "Use Si570-USB" check box in the Settings/DSP dialog. Do not change the "address" and "divider" settings.



- In the In the "Hz" fiield, type in `7046000` to set Rocy's default center frequency to 7.046 MHz.
- Set up Rocky for RX=only:
  - Click on `View/Settings and the "Audio" tab`



- Select the "`I/Q Input Device`" to be the your on-board sound card
- Select the "`Audio Output Device`" to be your on-board sound card
- Click on the "`Transmit`" tab



- Clear the "`Transmitter Enable`" checkbox to disable transmit mode

The PC is now set up to use Rocky to control the Si570 Local Oscillator.

Test USB

It is very important to follow the procedure below `exactly`. The USB interface and Rocky can interact in very strange fashion if you do not. Also, do not try to use Rocky's frequency changing mechanism until after the LO stage.

With Rocky setup as above, follow the steps below:

- **Apply power to the board**
- **Plug in the interface's USB cable to your PC**
- **Run Rocky (previously had been set up to use the USB interface at address 85 and has TX disabled)**
- **Rocky will issue an "error -5 message: `Set frequency (28184000) failed: USB_control_msg error -5`**
  signifying it tried to set the frequency to 4 times the default center frequency (7046000) in Rocky and did not receive any acknowledgement from the Si570 back through the USB Interface
- **Click "OK" on Rocky's "error -5"**

Home BOM Power Supply Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s);
External Connections Comments Revisions WB5RVZ Home

Softrock Lite + USB Xtall V9.0 Local Oscillator Stage

Introduction

Theory of Operation

The local oscillator (U4) is a programmable oscillator, whose programmatic parameters are set by the USB interface, U2.

The user, operating special SDR software on the PC, selects a desired center frequency. The PC issues commands, via a USB port, to U2 (the USB control chip from the preceding stage. These commands will result in U4's producing a frequency that is exactly 4 times the desired center frequency selected in the SDR program.

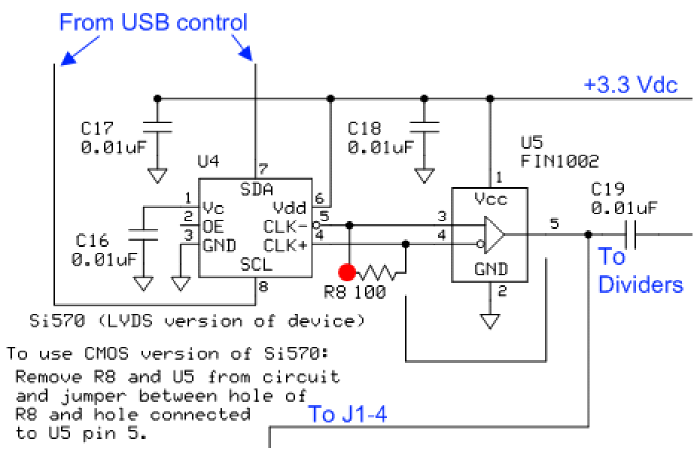
U2 responds to commands from the PC, translating them into commands in the I2C protocol to control the programmable oscillator U4. (If the I2C commands are not received by U4, it would default to an output frequency of 56.320 mHz.)

The IC U5 is needed if the version of U4 is the "LVDS" version. If U4 is a CMOS version, U5 (and R8) are not required and, instead, a jumper wire is installed to bypass them.

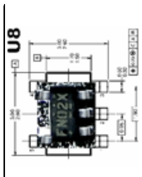
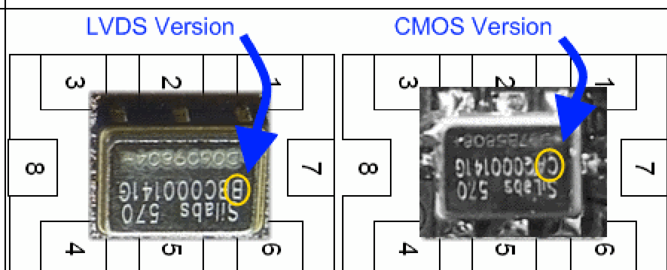
This LO stage must produce an output rf signal (available at J1 pin 4) that is four times the desired center frequency for the radio. This is then fed to the dividers/phasors section to produce the two center-frequency signals that are in quadrature and ¼ the LO frequency.

Schematic

This is a subset of the overall schematic. Note: red dot indicates resistor testpoints (hairpin, top, or left-hand lead)



Bill of Materials

Designation	Value	Color/Code	Orientation	Category	Notes
U5	FIN1002	FN02X		SMT	only required for LVDS version of Si570
U4	Si570 (CMOS version of device)			SMT	
C16	0.01uF			SMT 1206	

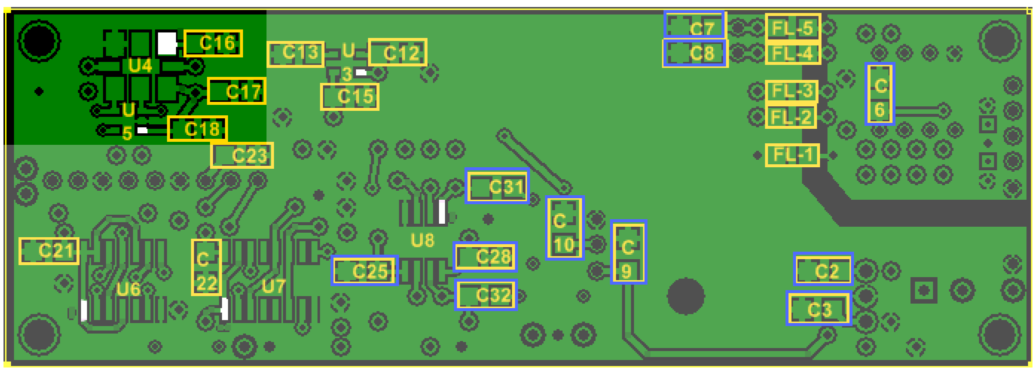
C17	0.01uF			SMT 1206		
C18	0.01uF			SMT 1206		
R08	100			flat-h		omit for CMOS
J1	9 pin					
C19	0.01uF	103		ceramic		

Summary Build Notes

- Install U5 - only for LVDS version of Si570 (bottom)
- Install U4 (bottom)
- Install 3 SMT capacitors (bottom)
- Install resistor (top, if LVDS)
- Install main bus jack, J1
- Install ceramic cap C19
- Install CMOS jumper wire (if Si570 is CMOS versio)
- Test the Stage

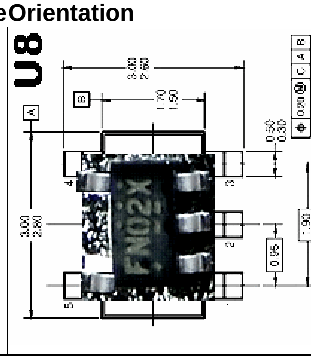
Detailed Build Notes

Bottom of the Board



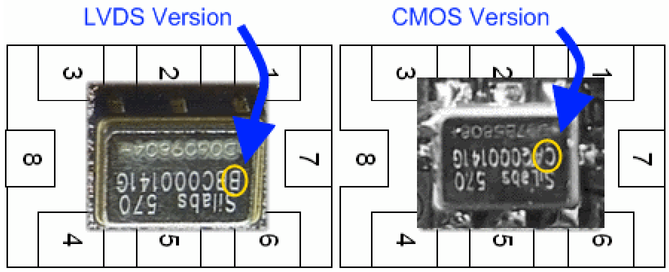
Install U5

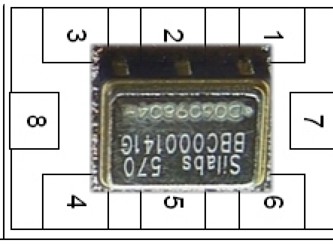
Do not confuse U5 with the voltage regulator (U3) marked "LFEA"

Designation	Value	Color/Code	Orientation	Category	Notes
U5	FIN1002	FN02X		SMT SOT-23	**LVDS only: do not install if U4 is a CMOS version.**

Install U4

Note: There are two versions of the Si570, the CMOS and the LVDS. See the chart below for how to distincuish them (the chips are shown in their mounting orientation for this kit). See Softrock Group message for discussion of differences.

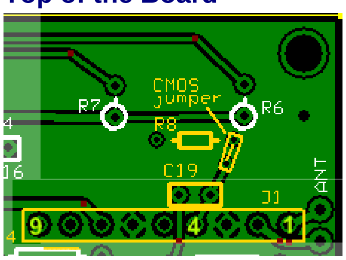


Designation	Value	Color/Code	Orientation	Category	Notes
U4	Si570			SMT	(LVDS version shown)

Install SMT Capacitors

Designation	Value	Color/Code	Orientation	Category	Notes
C16	0.01uF			SMT 1206	
C17	0.01uF			SMT 1206	
C18	0.01uF			SMT 1206	

Top of the Board



Install Resistor (see notes)

Designation	Value	Color/Code	Orientation	Category	Notes	
R08	100			flat-h		omit for CMOS

Install Ceramic Capacitor C19

Designation	Value	Color/Code	Orientation	Category	Notes
C19	0.01uF	103		ceramic	

Install CMOS Jumper (only required of Si570 is CMOS version

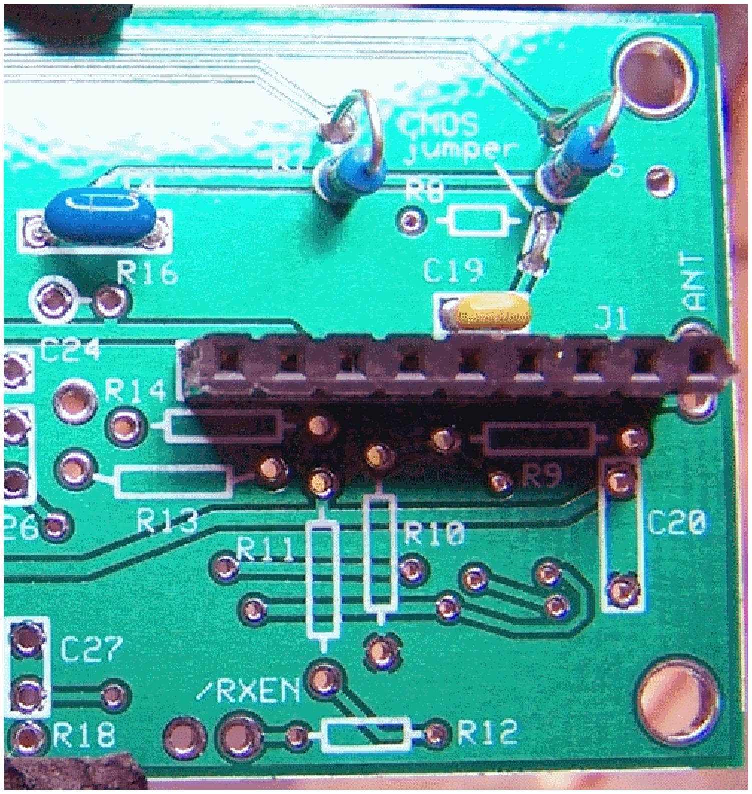
Designation	Value	Color/Code	Orientation	Category	Notes
short jumper wire					

Install Main bus Jack J1

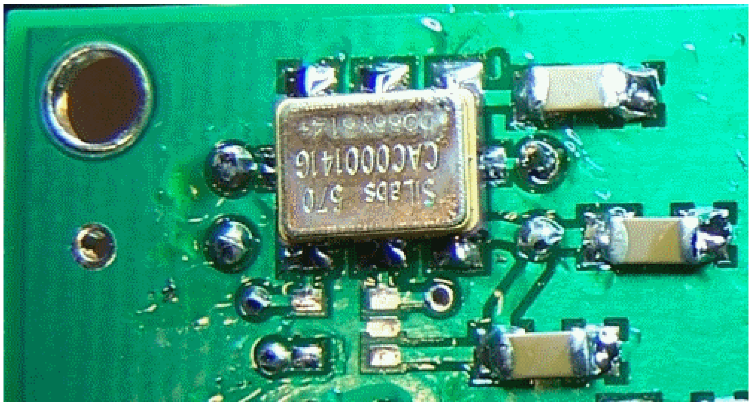
Designation	Value	Color/Code	Orientation	Category	Notes
J1	9 pin				

Completed Stage

Topside



Bottomside



Testing

Current Limited Power Test

- **Connect a 1 k ohm resistor in series with the power line and apply 12 V dc power**
- **the current should be relatively low (around 10 mA or less). Author's results = 8.1 mA**
- **Measure the voltage WRT ground at the +5 V testpoint at J2.**
- **A voltage of around 1-2 V dc indicates the power rail is not shorted. Author's results = 997 mV**

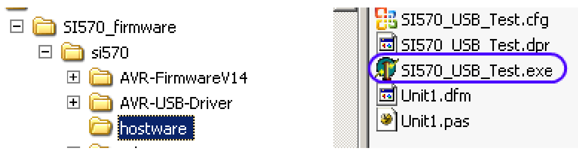
Current Draw (DMM)

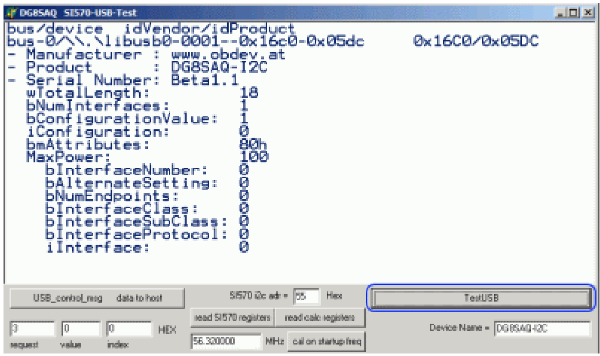
- **Current draw here is for the CMOS version of the Si570. Adjust these numbers up by about 14 - 20 mA for the LVDS version.**
- **With the USB cable unplugged, power up the board, and measure the current draw. This should now go to around 70-80 mA. Author's results 75.4 mA**

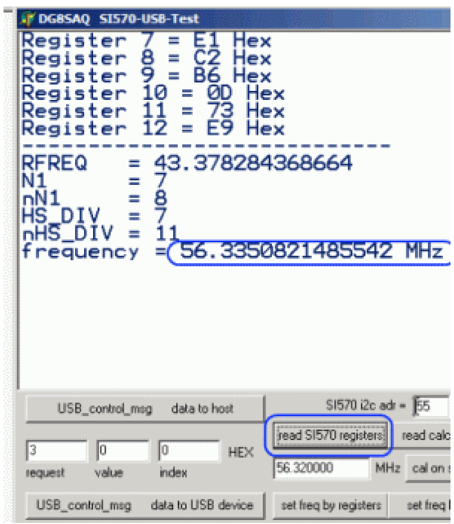
Si570 Test (courtesy of DG8SAQ)

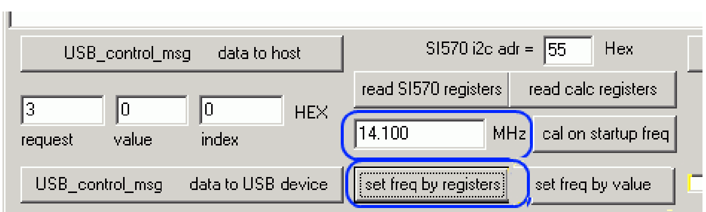
This test uses the `Si570_USB_Test.exe`program which comes with the Si570 drivers you downloaded and installed in the preceding (USB Control) stage.

Test Setup

- 

go to the directory into which you extracted the contents of `SI570_firmware.zip` and run the `Si570_USB_Test.exe`program
- power up the board and plu.g in the USB cable - note: there has been considerable discussion on the users' group regarding the power-up sequence. The consensus is that the V9.0 board must first be powered up, then the USB cable plugged into the PC
- 

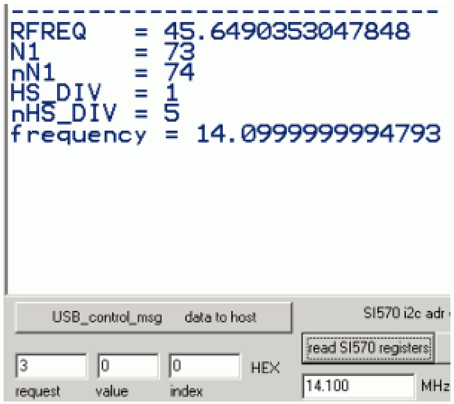
Click on the `TestUSB` button - you should see the results above
- 

Click on the `Read Si570 Registers`
button - you may see results similar to those above. No Si570 is the same, registers 10, 11 & 12 are likely to be different, the frequency will be around 56.32MHz> The main thing is that the frequency should be dose to the expected startup, normally 56.32 +/- perhaps 0.001MHz. See also Alan G4ZFQ's message on the Softrock reflector.
- 

Enter 14.1 in the "MHz" field and click on the `set frequency by` **`register`** button (see above) to command the Si570 to operate at 14,100,000 Hz. This sets the Si570's frequency to 14.1 MHz

  Note: the Si570's address is 55 hex. Some softwares use the hex address, others, e.g., Rocky, use the decimal representation, 85.

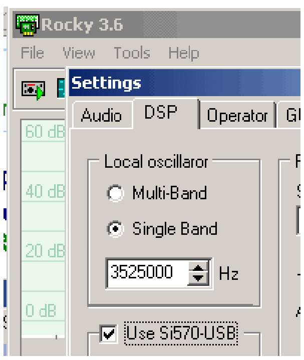
- Now, press the `Read Si570 Registers` button ant the following screen should be displayed:



LO Output - Receiver

Test Setup

This procedure will test this stage to determine whether it is outputting the correct frequency (4x the desired center frequency). It involves setting Rocky up for a desired center frequency of 3.525 MHz, such that Rocky will command the Si570 to output a signal at 4x that value, or 14.100 MHz. A ham transceiver will be required to detect the signal at 14.100 MHz.



The best order for connection of cables to the v9.0 board would be plug in the audio cable to the soundcard line-in, connect the +12 to the v9.0 board and then plug in the USB cable followed by the antenna connection.

- **download and install Rocky (if not already done)**
- **Run Rocky and click on `View/Settings/DSP`**
- **Check in the "Use Si570 USB" checkbox**
- **Click on the "single band" option button in the "Local Oscillator" box**
- **Enter 3525000 into the "Hz" field. This tells Rocky to configure the Si570 for 4 times the desired center frequency of 3.525 MHz**
- **fashion a small wireloop "antenna" to plug into Pin 4 of J1**
- **connect a wire to your transceiver's RX ANT jack and loop it through the "antenna" in pin 4 of J1**
- **Tune the transceiver to receive at 14.100 MHz (4x 3.525 MHz)**
- **Apply power to the board**
- **Connect the USB cable from the board to the PC**
- **In Rocky, click on `File/Start Radio` to turn on Rocky's SDR program and send the frequency command to the board**
- **The receiver should detect the signal**
- **go back to `View/Settings/DSP` and change the "Hz" field to 3530000 and tune the RX to receive at 14.120 MHz. You should hear the signal at this new frequency**

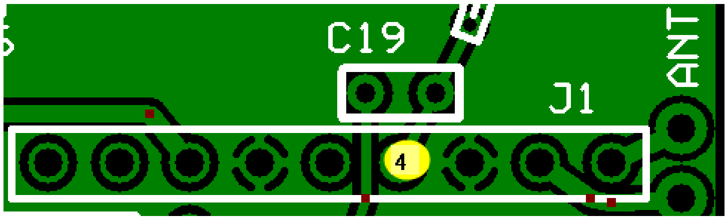
LO Output (Scope/Freq Counter)

The Local Oscillator should output a signal at the four times the center frequency selected by Rocky.

Do not attempt this measurement unless you have a calibrated scope of very good quality and

correctly compensated probes.

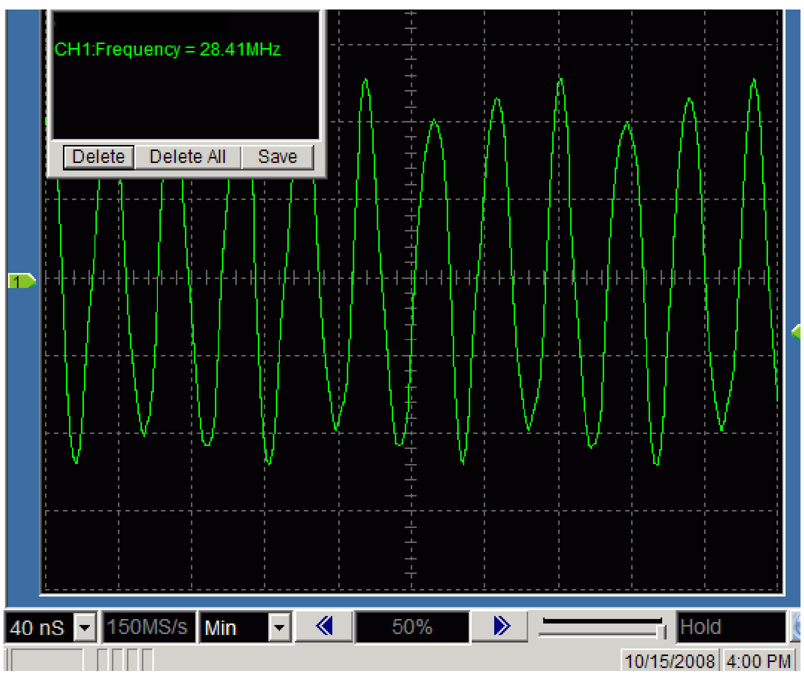
Test



- Tune the USB oscillator settings to get a center frequency of 7.046 MHz
- Apply power to the board
- Test the output of (U8 in the LVDS version of the kit or U4 in the CMOS version) at pin 4 of J1: the frequency should be 28.184 MHz (4 times the desired center frequency of 7.046 MHz).

  The AC pk-pk voltage should be aproximately or less than 3.3 V p-p.

  The waveform should approximate be a square wave.
- If you get 56.32 MHz (or 14.08 times 4) with tuning set as above, or regardless of the frequency selected, this means:
  - U2 has been incorrectly installed or
  - pins 7 or 8 of U4 may have bad solder joints
  - The USB interface is not working to receive the control signals from the software
  - The software is not configured correctly to use the USB interface



LO Output Test for 40m (frequency measurement is approximate, at best)

tml>

Softrock Lite + USB Xtall V9.0 Dividers Stage

Home BOM Power Supply USB Control Local Oscillator RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Introduction

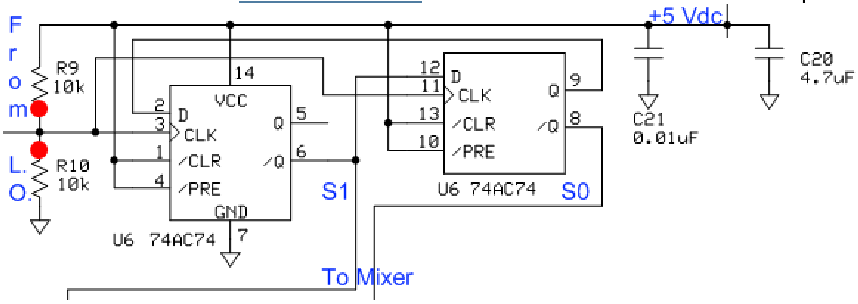
Theory of Operation

The Dividers stage takes in the local oscillator's signal and divides it by 4, producing two output signals. Each output signal is at a frequency that is ¼ the stage's input signal and is a square wave with 50% duty cyde. The 50% duty cycle is with respect to the4 5V rail.

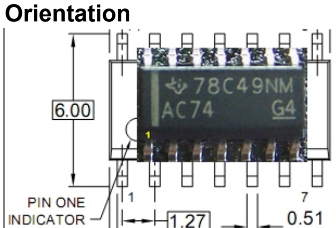
The signals are "in quadrature", that is, they are 90° out of phase with each other. These are provided to the TX and RX mixer stages as clocking signals. They are called out on testpoints marked S0 and S1.

Schematic

This is a subset of the overall schematic. Note: red dot indicates resistor testpoints (hairpin, top, or left-hand lead)



Bill of Materials

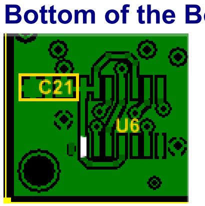
Designation	Value	Color/Code	Orientation	Category	Notes
U6	74AC74			SMT	
C21	0.01uF			SMT 1206	
R09	10k		flat-h		
R10	10k		flat-v		
C20	4.7uF	475		ceramic	

Summary Build Notes

- Install C21 (bottom)
- Install SMT U6 (bottom)
- Install 2 resistors (top)
- Install 1 ceramic capacitor (top)
- Test the Stage

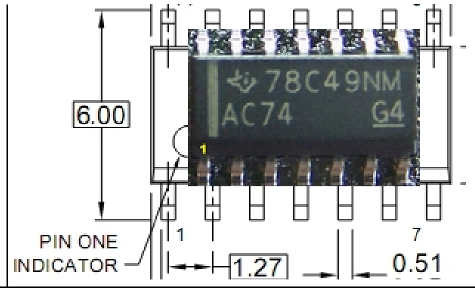
Detailed Build Notes

Bottom of the Board



Install U6

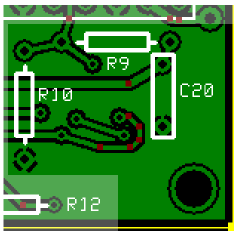
Designation	Value	Color/Code	Orientation	Category	Notes

U6	74AC74				SMT SOIC-14	

Install C21 SNT Cap

Designation	Value	Color/Code	Orientation	Category	Notes
C21	0.01uF			SMT 1206	

Top of the Board



Install Resistors

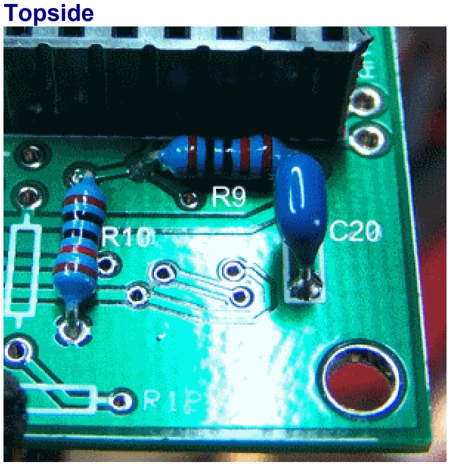
Designation	Value	Color/Code	Orientation	Category	Notes
R09	10k		flat-h		
R10	10k		flat-v		

Install Ceramic Capacitor

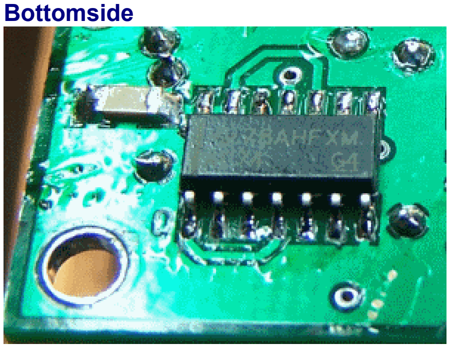
Designation	Value	Color/Code	Orientation	Category	Notes
C20	4.7uF	475		ceramic	

Completed Stage

Topside



Bottomside



Testing

Current Draw(DMM)

- Current numbers here are for the CMOS version of the Si570. You will need to adjust these up by about 14 mA for the LVDS version.
- Power the board up
- Measure the current draw and 5 V rail voltage with a 1K Ω limiting resistor
- Measure the current draw without the limiting resistor.

Testpoint	Nominal Value	Author's	Yours
Current Limited mA	6-10 mA	7.5 mA	______
Current limited 5V rail	1-2 Vdc	975 mV	______
Non limited draw mA	80-90 mA	85.7 mA	______

Voltage Tests (DMM)

If the output of the dividers are not as expected, check the voltages at the pins of U6. Unexpected values here usually point to problems with soldering U5 and/or the voltage dividing resistors R9 and R10. Using a DMM:

- Measure the output of the voltage divider with respect to ground. Measure at the top lead of R10 (or the left-hand lead of R9). This should yield approximately ½ the 5 volt rail voltage.
- Measure the voltages (with respect to ground) on the pins of U6. It is best to test for these voltages at the actual pins (not the pads), thereby ensuring correct soldering of the pins to the pads.

Testpoint	Nominal Value	Author's	Yours
Topside, R10's top lead	2.5 Vdc	2.50 Vdc	______
U6, Pins 1, 4, 10, 13, 14	5 Vdc	4.96 Vdc	______
U6, pins 2, 3, 5, 6, 8, 9, 11, 12	2.5 Vdc	2.48-3.50 Vdc	______
U6, pin 7	0 Vdc	0 Vdc	______

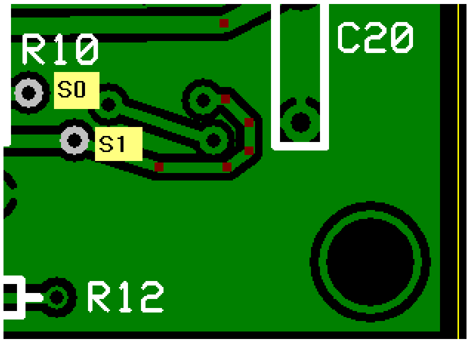
Test Center Frequency Output

- Connect a short piece of wire as an "antenna" for your HF RX and lay it over the board
- No need to use the USB control in this test
- Apply power to the board
- Tune your radio to find the signal at 14.084 MHz (¼ the Si570 default frequency of 56.336 MHz
- If you can detect the signal and have passed the voltage tests above, your divider stage is pretty well assured to be working correctly.

U5 Output (Optional Test)

In the event that you have or have access to a dual channel oscilloscope, you can test the divider outputs here.

Do not attempt this measurement unless you have a calibrated scope of very good quality and correctly compensated probes.

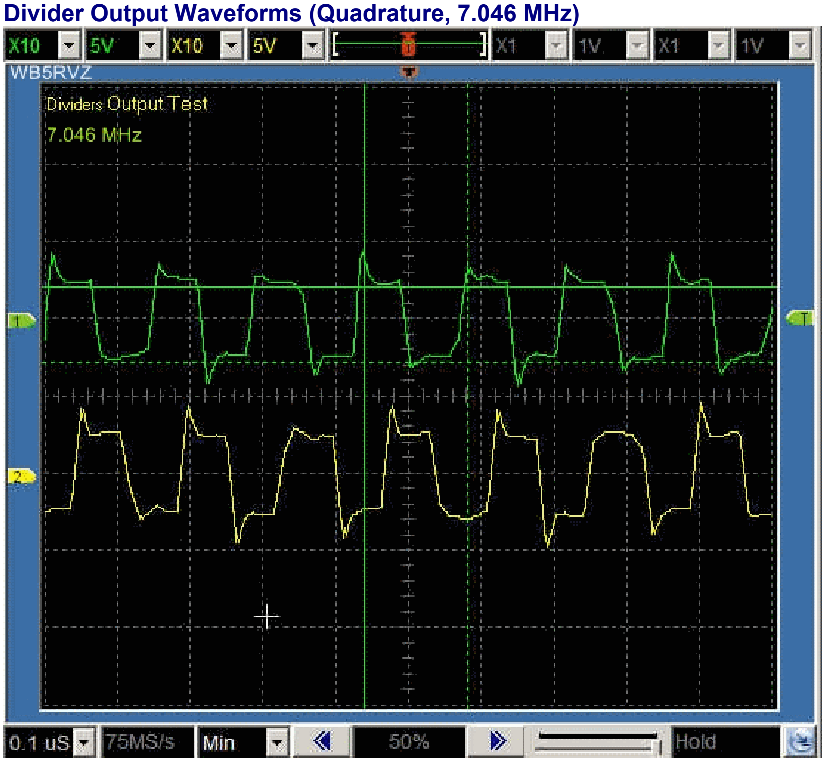


U5 sends I and Q signals to the mixer's S0 and S1 inputs.

- Use a dual channel oscilloscope, triggering on Channel 1
- Power up the board and plug in the USB cable
- Run Rocky and set the center frequency to 7.046 MHz

- **measure the S0 and S1 outputs at the corresponding testpoints on the top side of the board, as indicated above.**
- **They should both be the same frequency (¼ of the LO Output - assuming you use the settings from the LO test, that would be 7.046 mHz) and should be in quadrature (90° out of phase with each other). The image below shows approximations of p-p voltages and frequencies of the 2 quadrature signals.**
- **They should be approximately 5 volt p-p square waves. The square waves may have a fair amount of ringing on them depending a bit on your scope quality and connection to the circuit board (see Waveforms below).**

Divider Output Waveforms (Quadrature, 7.046 MHz)



Home BOM Power Supply USB Control Local Oscillator RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Softrock Lite + USB Xtall V9.0 Op Amps Stage

Home BOM Power Supply USB Control Local Oscillator Dividers RX Mixer(QSD) RX BPF(s);
External Connections Comments Revisions WB5RVZ Home

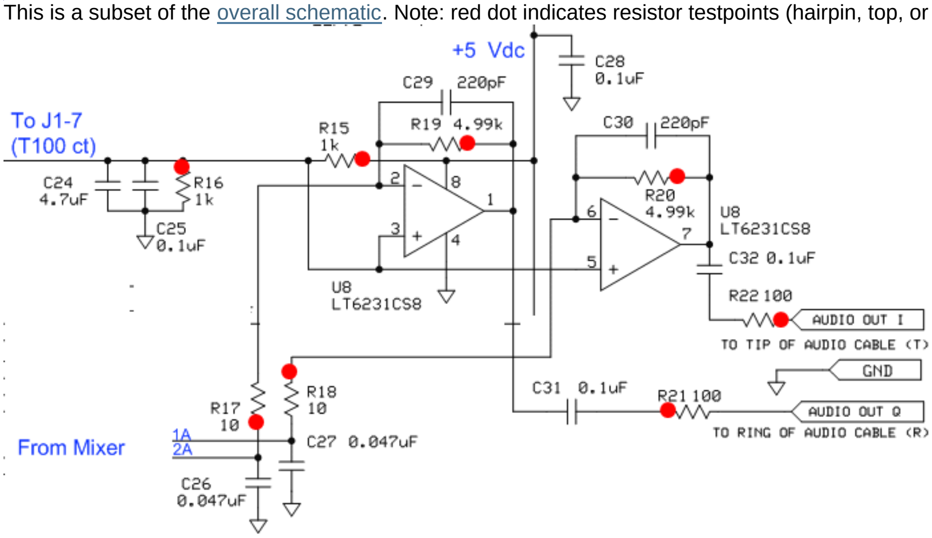
Introduction

Theory of Operation

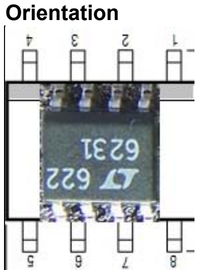
This stage amplifies the quadrature audio frequency difference products from the Mixer stage via R17 and R18. R19 and R20 make up a voltage divider that provides the 2.5 Vdc bias to the Op-Amps, configured as an inverting amplifier. The ratios of R19/R17 and R20/R18, respectively, determine the voltage gain of the output over the input for each Op-Amp. That voltage gain is theoretically 499:1, or about 54 dB. Each Op-Amp's output is capacitively coupled through a 100 ohm resistor to the "Ring" (Q) and "Tip" (I) Audio Out terminals for input to the PC's sound card

Schematic

This is a subset of the overall schematic. Note: red dot indicates resistor testpoints (hairpin, top, or left-hand lead)



Bill of Materials

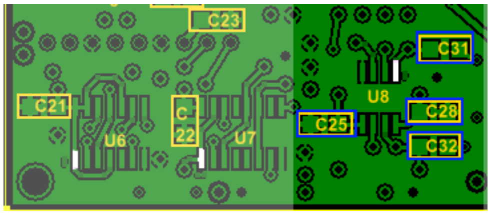
Designation	Value	Color/Code	Orientation	Category	Notes
U8	LT6231CS8			SMT	
C31	0.1uF			SMT 1206	black marked strip
C32	0.1uF			SMT 1206	black marked strip
C28	0.1uF			SMT 1206	black marked strip
C25	0.1uF			SMT 1206	black marked strip
R17	10		W-E		
R18	10		flat-h		
R15	1k		flat-h		
R16	1k		W-E		
R19	4.99k		E-W		
R20	4.99k		flat-h		
R21	100		flat-v		
R22	100		flat-h		
C26	0.047uF	473		ceramic	
C27	0.047uF	473		ceramic	
C29	220pF	221		ceramic	
C30	220pF	221		ceramic	
C24	4.7uF	475		ceramic	

Summary Build Notes

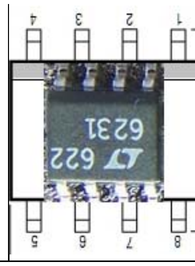
- Install SMT IC U8 (bottom)
- Install 4 x 0.1 µF SMT capacitors (bottom)
- Install 8 resistors R15-R22 (top)
- Install 5 ceramic capacitors C24 and C26-C30(top)
- Test the Stage

Detailed Build Notes

Bottom of the Board



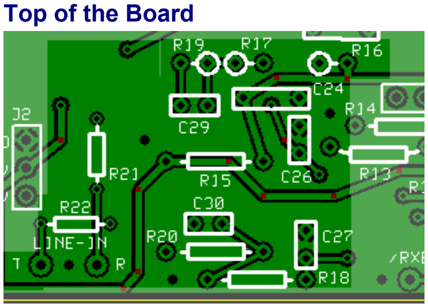
Install U8

Designation	Value	Color/Code	Orientation	Category	Notes
U8	LT6231CS8			SMT SOIC-8	

Install SMT Caps

Designation	Value	Color/Code	Orientation	Category	Notes
C31	0.1uF			SMT 1206	black marked strip
C32	0.1uF			SMT 1206	black marked strip
C28	0.1uF			SMT 1206	black marked strip
C25	0.1uF			SMT 1206	black marked strip

Top of the Board



Install Resistors

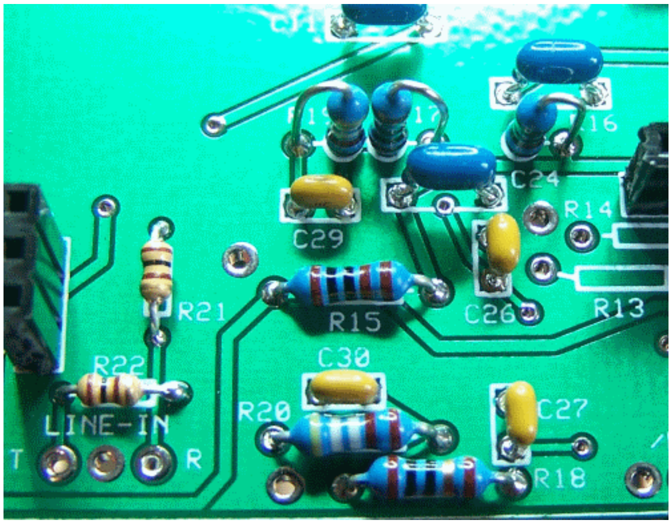
Designation	Value	Color/Code	Orientation	Category	Notes
R17	10		W-E		
R18	10		flat-h		
R15	1k		flat-h		
R16	1k		W-E		
R19	4.99k		E-W		
R20	4.99k		flat-h		
R21	100		flat-v		
R22	100		flat-h		

Install Ceramic Capacitors

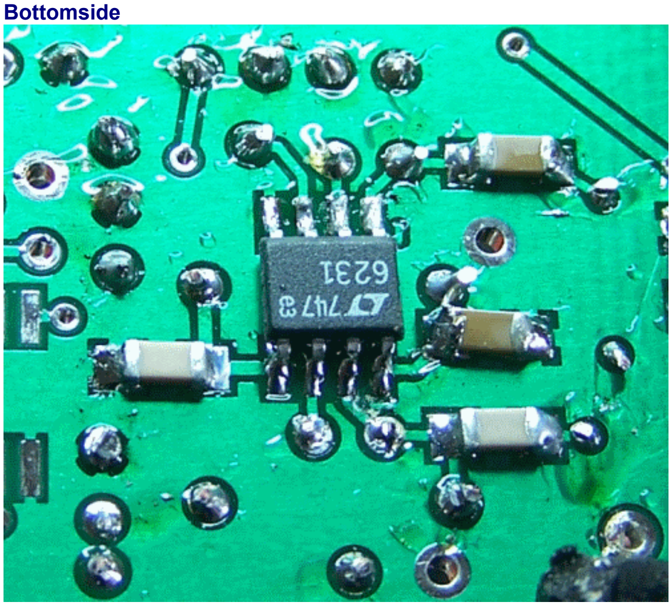
Designation	Value	Color/Code	Orientation	Category	Notes
C26	0.047uF	473		ceramic	
C27	0.047uF	473		ceramic	
C29	220pF	221		ceramic	
C30	220pF	221		ceramic	
C24	4.7uF	475		ceramic	

Completed Stage

Topside



Bottomside



Testing

Current Draw(DMM)

- Current numbers here are for the CMOS version of the Si570. You will need to adjust these up by about 14 mA for the LVDS version.
- Power the board up
- Measure the current draw and 5 V rail voltage with a 1K Ω limiting resistor
- Measure the current draw without the limiting resistor.

Testpoint	Nominal Value	Author's	Yours
Current Limited mA	6-10 mA	7.5 mA	______
Current limited 5V rail	1-2 Vdc	971 mV	______
Non limited draw mA	90-100 mA	94.9 mA	______

#voltage_divider_test

Voltage Divider R15/R16(DMM)

- Measure the voltage at the R16 hairpin lead with respect to ground.
- It should read approximately 2.5 Vdc (½ the 5 volt rail).

Testpoint	Nominal Value	Author's	Yours
R16 hairpin lead	2.5 Vdc	2.48 Vdc	______

Pin Voltages (DMM - 5, 2.5, and 0 Vdc)

- Measure the voltages at the pins of U8. (see bottomside image above)
- It is best to test for pin voltages at the actual pins (not the pads), thereby ensuring correct soldering of the pins to the pads.

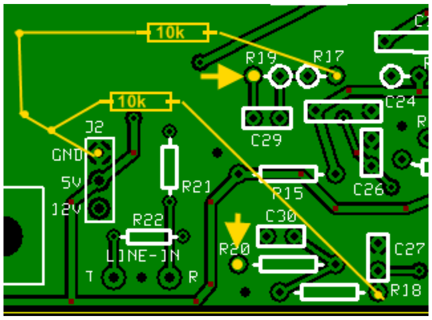
Testpoint	Nominal Value	Author's	Yours

U8, Pins 1, 2, 3, 5, 6 & 7	2.5 Vdc	2.48-2.51 Vdc	______
U8, Pin 8	5 Vdc	4.96 Vdc	______
U8, Pin 4	0 Vdc	0 Vdc	______

OpAmp Test - DMM (No Scope)

Tony Parks suggested this next test,which requires only a DMM, a 10 k resistor, and some clip leads.

The test will test each of the two Op-Amps. If the Op-Amp being tested is working, then the voltage measured at the output of the Op-Amp will increase to accomodate the effect of the changed bias on the input. Passing these tests gives you more than enough confidence to move on to the Mixer stage.



Testpoint	Nominal Value	Author's
R19 hairpin - no bridge	2.5 Vdc	2.51 Vdc
R19 hairpin - R17 bridged	3.75 Vdc	3.75 Vdc
R20 left-hand lead - no bridge	2.5 Vdc	2.51 Vdc
R20 left-hand lead - R18 bridged	3.75 Vdc	3.75 Vdc

- **Obtain a 10k resistor (you can use R11 from the next stage's BOM)**
- **using the DMM, measure the dc voltage with respect to ground at the hairpin of R19.**
  The result should be approximately 2.5 Vdc (½ the 5 Vdc rail).
- **keep the DMM lead on R19's hairpin**
- **Using two clip leads, "bridge" the 10k resistor between the hairpin of R17 and ground. See the diagram to the left.**
- **Observe the voltage reading at R19 hairpin. If OpAmp 1 is working, the voltage should have jumped to approximately 3.75 Vdc**
- **Remove the resistor/clip lead from R17 and the voltage at R19 should go back to the 2.5 Vdc level.**
- **Follow these same steps for OpAmp2, substituting:**
  - **R18 (right-hand lead) for R17 (hairpin) and**
  - **R20 (left-hand lead) for R19 (hairpin).**

Softrock Lite + USB Xtall V9.0 Mixer Stage

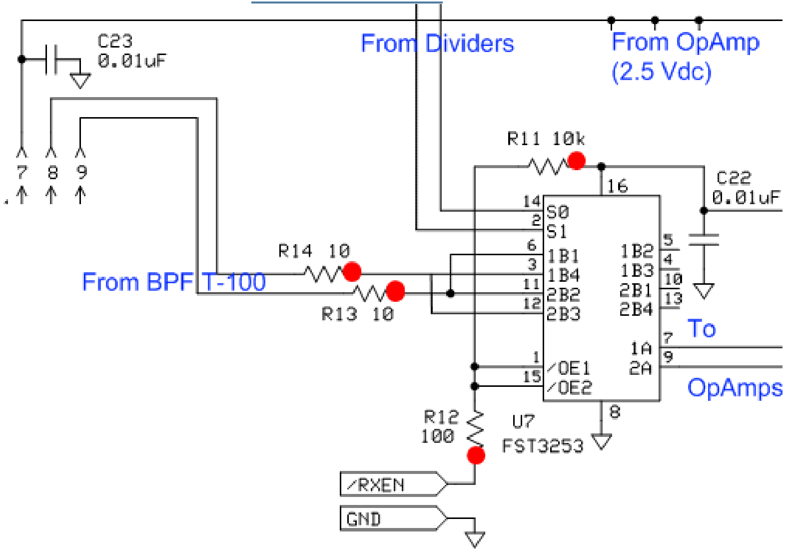
Introduction

Theory of Operation

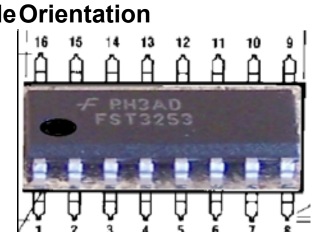
The mixer stage acts like two traditional direct conversion mixers operating in tandem. Each takes in half of the filtered RF from the bandpass filter stage and one of the quadrature center frequency signals, then "mixes" them, with an output being the traditional mixer products, in this case, two audio frequency signals that represent the difference between the two inputs (RF and Local Oscillator). These two signals are referred to as the I (in-phase) and Q (Quadrature) signals and are fed into the high gain Op-Amps stage for amplification and delivery to the audio outputs (and, thence, to the PC's sound card). Resistors R11 and R12 form a voltage divider to produce approximately 50 mV dc at pins 1 and 15 to enable the mixer's operation when the /RXEN is grounded.

Schematic

This is a subset of the overall schematic. Note: red dot indicates resistor testpoints (hairpin, top, or left-hand lead)



Bill of Materials

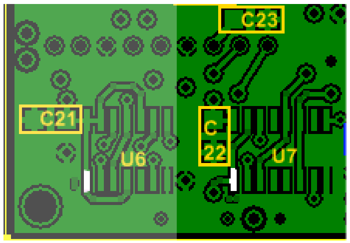
Designation	Value	Color/Code	Orientation	Category	Notes
U7	FST3253			SMT	
C22	0.01uF			SMT 1206	
C23	0.01uF			SMT 1206	
R11	10k		flat-v		
R12	100		flat-h		
R13	10		flat-h		
R14	10		flat-h		
Lead wire				connector	Install a short, stout wire from a cut-off lead between the hole marked /RXEN and the ground hole to its left

Summary Build Notes

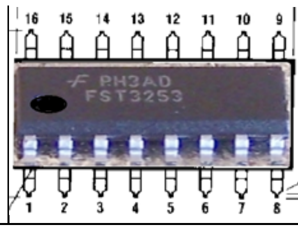
- Install SMT IC U7 (bottom)
- Install 2 SMT capacitors (bottom)
- Install 4 resistors
- Install ground strap for /RXEN
- Test the Stage

Detailed Build Notes

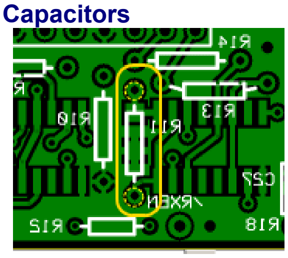
Bottom of the Board



Install U7

Designation	Value	Color/Code	Orientation	Category	Notes
U7	FST3253			SMT SOIC-16	

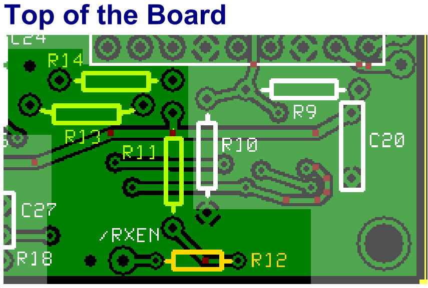
Install SMT Capacitors



Watch out for solder splash when soldering C22 - the pads are very close to the holes for R11's leads! You might want to insert R11's leads into their holes prior to installing C22.

Designation	Value	Color/Code	Orientation	Category	Notes
C22	0.01uF			SMT 1206	
C23	0.01uF			SMT 1206	

Top of the Board



Install Resistors

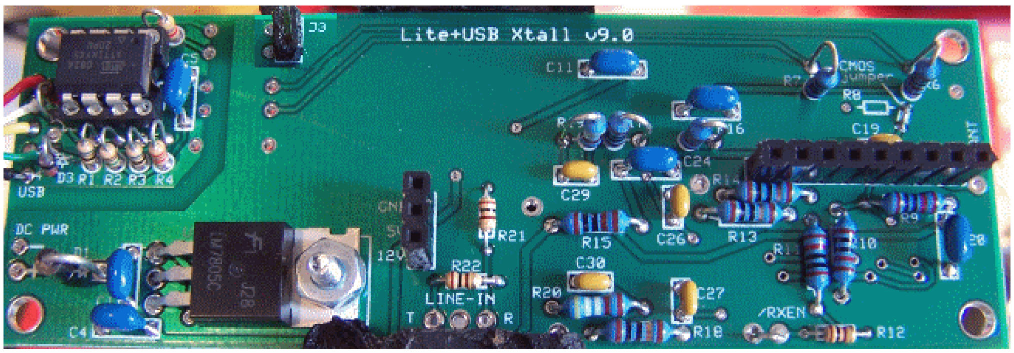
Designation	Value	Color/Code	Orientation	Category	Notes
R11	10k		flat-v		
R12	100		flat-h		
R13	10		flat-h		
R14	10		flat-h		

Install ground strap for RX Enable

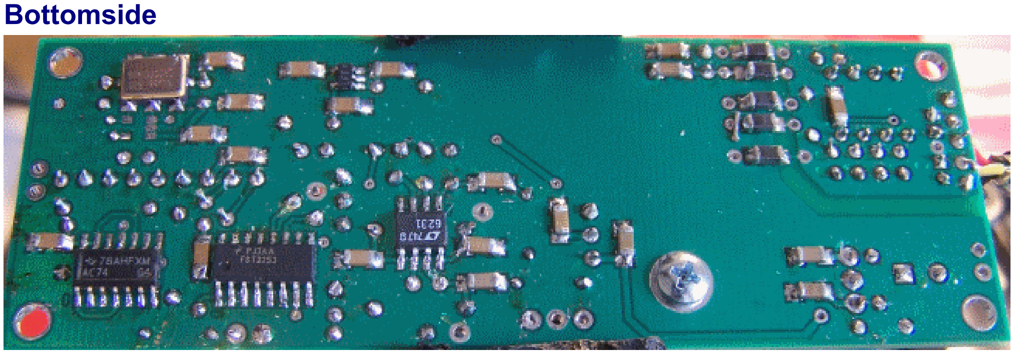
Designation	Value	Color/Code	Orientation	Category	Notes
Lead wire				connector	Install a short, stout wire from a cut-off lead between the hole marked /RXEN and the ground hole to its left

Completed Stage

Topside



Bottomside



Testing

Note: Some tests in this stage require you to have built and plugged in at least one bandpass filter.

If you have not yet done so, you can still conduct the current and voltage tests provided you short pins 7, 8, and 9 of J1 together to provide the DC equivalent of the T100 secondaries.

Current Draw(DMM)

- **Current numbers here are for the CMOS version of the Si570. You will need to adjust these up by about 14 mA for the LVDS version.**
- **Power the board up (author has been using an 11.6 Vdc battery pack**
- **Measure the current draw and 5 V rail voltage with a 1K Ω limiting resistor**
- **Measure the current draw without the limiting resistor.**

Testpoint	Nominal Value	Author's	Yours
Current Limited mA	6-10 mA	7.5 mA	______
Current limited 5V rail	1-2 Vdc	971 mV	______
Non limited draw mA	90-100 mA	97.4 mA	______

Pin Voltages (DMM)

Measure U7 Pin Voltages

- **Using a DMM, measure the dc voltage (with respect to ground) of the pins of U7.**
- **It is best to test for these voltages at the actual pins (not the pads), thereby ensuring correct soldering of the pins to the pads.**

Testpoint	Nominal Value	Author's	Yours
U7, Pin 16	5 Vdc	4.97 Vdc	______
U7, Pin 8	0 Vdc	0 Vdc	______
U7, Pins 1and 15	50 mVdc	49 mVdc	______
U7, Pin 2	2.5 Vdc	2.48 Vdc	______
U7, Pin 14	2.5 Vdc	2.48 Vdc	______
U7, Pin 7	2.5 Vdc	2.48 Vdc	______
U7, Pin 9	2.5 Vdc	2.48 Vdc	______

If the voltage at pins 1 and 15 is not in the area of 50 mV, then the mixer will not be enabled and there will be no outputs at pins 7 and 9.

If you see a high (~5 Vdc) voltage at pins 1 and 15, check your /ENRX to be sure it is grounded

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX BPF(s);
External Connections Comments Revisions WB5RVZ Home

Softrock Lite + USB Xtall V9.0 Band Pass Filter Stage

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD)
External Connections Comments Revisions WB5RVZ Home

Introduction

Theory of Operation

This stage lets the SDR filter out the RF spectrum arriving at the antenna into a "chunk" of the RF spectrum corresponding to the desired band(s). This is filtering "in the large", and is designed to minimize interference/harmonics from very strong, out-of-band signals.

There are four separate, pluggable boards which can be built to provide BPF functionality over the range from 160m to 10m.

Mike KF4BQ has conducted tests on the BPF boards to determine the frequency boundaries of these "chunks" (the passbands) of RF spectrum. You can view the results here.

Note: the pluggable bandpass filters may be replaced by the new switchable HF BPF board kit, which implements 4 switchable BPFs on a single board, which can be manually switched or (once firmware is updated) switched via USB control.

Bill of Materials

Band specific values

Bands	C100	C101	L100	T100
160m	390pF	5600pF	18.7uH, T30-2(red) core 66T #30 AWG	1.4uH(primary), T30-2(red) core 18T #30 AWG on primary 9T #30 AWG on each bifilar secondary
80m/40m	560pF	680pF	1.6uH, T25-2(red) core 22T #30 AWG	1.2uH(primary), T25-2(red) core 18T #30 AWG on primary 9T #30 AWG on each bifilar secondary
30m/20m/17m	180pF	220pF	0.78uH, T25-6(yellow) core 17T #30 AWG	0.6uH(primary), T25-6(yellow) core 14T #30 AWG on primary 7T #30 AWG on each bifilar secondary
15m/12m/10m	82pF	330pF	0.53uH, T25-6(yellow) core 14T #30 AWG	0.13uH(primary), T25-6(yellow) core 7T #30 AWG on primary 4T #30 AWG on each bifilar secondary

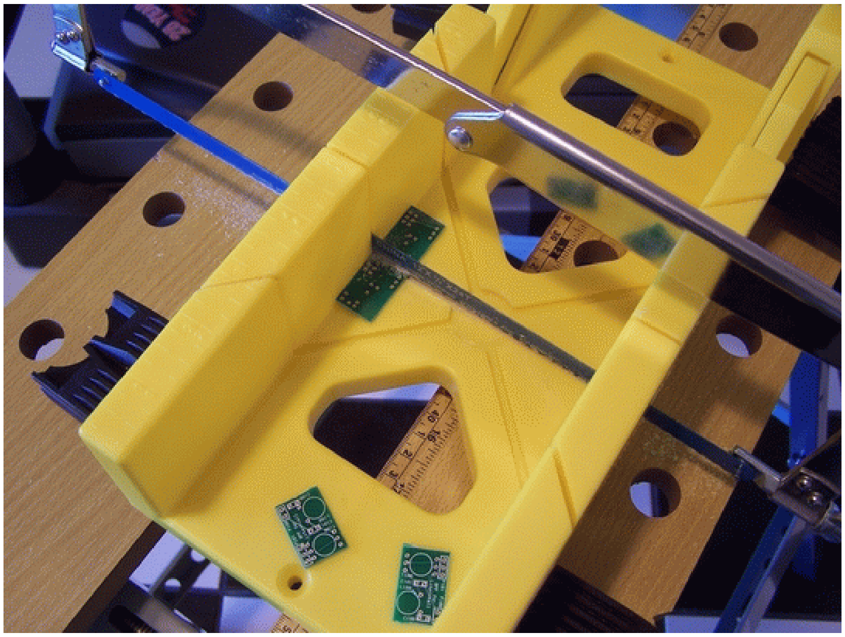
Designation	Value	Color/Code	Orientation	Category	Notes
-					
BPF-1					160m board
P100-1	2 pin				
P101-1	3 pin				
C100-1	390 pF	391		ceramic	
C101-1	5600 pF	562		ceramic	
T100-1	1.4 uH	T30-2 (red)			18T/9T bifilar #30 (10"/5")
L100-1	18.7 uH	T30-2 (red)			66T #30 (32")
-					
BPF-2					80/40m board
P100-2	2 pin				
P101-2	3 pin				
C100-2	560 pF	561		ceramic	
C101-2	680 pF	681		ceramic	
T100-2	1.2 uH	T25-2 (red)			18T/9T bifilar #30 (10"/5")
L100-2	1.6 uH	T25-2 (red)			22T #30 (11")
-					
BPF-3					30/20/17m board
P100-3	2 pin				
P101-3	3 pin				
C100-3	180 pF	181		ceramic	
C101-3	220 pF	221		ceramic	
T100-3	0.6 uH	T25-6 (yellow)			14T/7T bifilar #30 (8"/5")
L100-3	0.78 uH	T25-6 (yellow)			17T #30 (9")
-					
BPF-4					15/12/10m board
P100-4	2 pin				
P101-4	3 pin				

C100-4	82 pF	82		ceramic	
C101-4	330 pF	331		ceramic	
T100-4	0.13 uH	T25-6 (yellow)			7T/4T bifilar #30 (5"/4")
L100-4	0.53 uH	T25-6 (yellow)			14T #30 (8")

Detailed Build Notes

There are four bandpass filters (BPFs) you can build, each on its own board with 2 caps, a coil, a transformer, and two sockets for plugging it into the main board. The Bill of Materials above provides you with the parts list for each board. You only need to build one BPF to test out your receiver capability. It is recommended - especially if you are inexperienced in winding coils and toroids - to begin with a BPF for the band you are least interested in (just to get the practice in a non-threatening fashion).

Saw The Boards



The BPF filter boards are in a strip of four boards and will require the kit builder to hacksaw between the boards to separate the individual BPF boads. It is suggested to use a small plastic miter box and a fine-toothed blade (24 tpi or better) to help cut perpendicularly across the 0.65 inch wide strip. This seems to work well. However, please note the safety warnings on the Softrock reflector (message 23126) concerning the danger in inhaling the dust resulting from sawing.

Winding Inductors

To learn how to wind coils and transformers, please read the tips from the experts and then view the excellent videos on KC0WOXs Website to solidify your understanding of the task.

Concernimg the number of turns in the windings, David WW2R has reported that he had to adjust the number of windings on L100-1 (the 66 turn coil on the 160m band) because of the fact that the toroid was not able to accept 66 turns as a single layer, without winding back over some of the existing winding. Overlapping turns caused him to need 69 turns to reach the required inductance of 18.7 uH.

Pete N4ZR chimed in on this subject, too, adding: "The 160-meter L100 requires 66 turns, but only about 40-45 turns will fit on the core in a single layer. You need to keep winding in the same direction in a second layer until you complete the 66-69 turns. I wound 69 originally, but on checking with my MFJ-259, which may not be very accurate the inductance appeared to be a little high.

When winding bifilar windings, it is a lot easier to wind the bifilar winding if you fold the wire in half but don't cut, and use the folded (closed) end (with or without a sewing needle) to feed through the toroid or binocular core.

Wire Lengths

:

Refer to the BOM above to see the recommended length of wire (in inches) for each inductor. These lengths include generous

SWAGS to accomodate lead lengths, etc.. These were determined using DL5WWB's calculator (adding an inch or so to the resultant length, just for good measure.

When the BOM states `BPF-80/40: 18T/9T bifilar #30 (10"/5")` this means:

- Primary: 18 turns of #30, using 10" for the single winding.
- Secondaries: 9 turns of #30, using a 10" length of wire and fold it over at the 5" point, twisting it together into a bifilar strand, winding it evenly distributed over the primary winding for 9 turns. The bifilar strand should be about two-three twists per inch.

Core Sizes

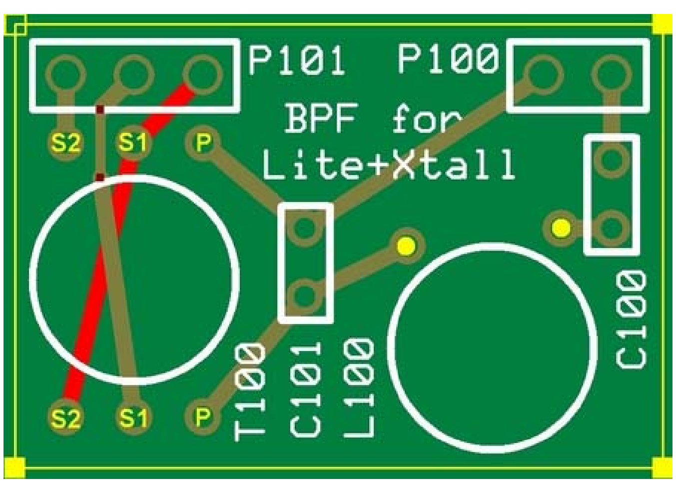
: The chart below provides the capacitance values and the winding instructions by band group. Carefully note that some bands use different size and color cores. Be sure to use the right core for the board you are building:

0. 1. 160 m: T30-2 (red)
2. 80/40m: T25-2 (red)
3. 30/20/17/15/12/10m : T25-6 (yellow)

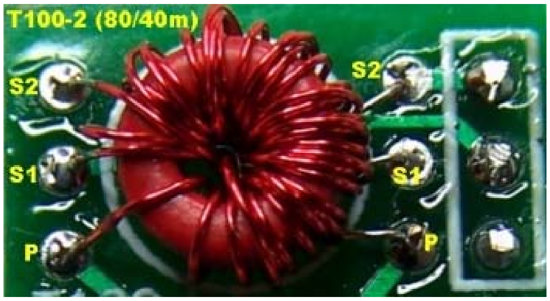
Band specific values

Bands	C100	C101	L100	T100
160m	390pF	5600pF	18.7uH, T30-2(red) core 66T #30 AWG	1.4uH(primary), T30-2(red) core 18T #30 AWG on primary 9T #30 AWG on each bifilar secondary
80m/40m	560pF	680pF	1.6uH, T25-2(red) core 22T #30 AWG	1.2uH(primary), T25-2(red) core 18T #30 AWG on primary 9T #30 AWG on each bifilar secondary
30m/20m/17m	180pF	220pF	0.78uH, T25-6(yellow) core 17T #30 AWG	0.6uH(primary), T25-6(yellow) core 14T #30 AWG on primary 7T #30 AWG on each bifilar secondary
15m/12m/10m	82pF	330pF	0.53uH, T25-6(yellow) core 14T #30 AWG	0.13uH(primary), T25-6(yellow) core 7T #30 AWG on primary 4T #30 AWG on each bifilar secondary

For Each BPF Board



(referring to the Band Specific Values chart, above):

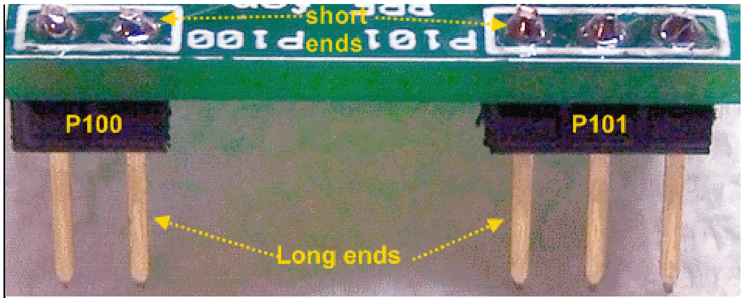


Build Steps for each BPF Board

Check	Designation	Type	Notes
	L100-#	Coil	Wind, prepare, horizontally mount, and solder the coil, L100, using the correct core size and color and turn count.. • Carefully count the turns[1]. Each pass thru the center is 1 turn. • Leave approximately 1/2 inch for each lead. • Use an emery cloth to scrape the insulation off the leads up to the last 1/8 inch. • Pull the leads through the holes directly above the circle for L100 on the BPF board (marked in yellow above). • Flatten the core horizontally, pull the leads snug, bend them on the bottom side of the board, and solder the leads. • Test for continuity (~0 ohms) from the lower hole of C100 through the coil to the lower hole of C101. • If there is no continuity, check soldering of the leads and resolder as necessary.

	T100-#	Transformer	Wind, prepare, horizontally mount, and solder the transformer, T100
			• Transformer T100-# will be mounted horizontally and raised above the board about 1/16 of an inch. In winding T100-#, first wind the primary winding with enameled wire so that the primary winding starts and ends at about the same point on the core and is uniformly spread around the core.
			• Twist two pieces of enameled wire together (bifilar) at about 3 twists per inch and wind the secondary windings with the windings starting and ending where the primary winding starts and ends. When you have wound the transformer, you will have 6 leads, 3 (one primary, one secondary 1, and one secondary 2) on each side of the core.
			• When trimming the wires, recognize that the 3 leads coming from one side of the core may need to be a little longer than those from the other side (to facilitate mounting the transformer horizontally.
			• Insert the leads, following the annotations on the BPF board above:
			    ○ "P" represents the primary leads on each side of the core;
			    ○ "S1" represents the leads for the first secondary winding on each side;
			    ○ "S2" represents the leads for the second secondary winding on each side.
			• Test for continuity on the two primary leads ("P" in the image above) by putting your ohmmeter leads on the two holes for C101. If you do not have continuity, then you likely have a soldering issue on the primary leads.
			• Test for continuity between either of the primary leads and each of the secondary leads. You should see an open circuit.
			• If you **do** get continuity, look for a short in the transformer or in its solder joints.
			• Test for continuity between pins 2 and 3 of P101. You should get continuity.
			• If you do not get continuity, one or more of your secondary leads has a solder problem.

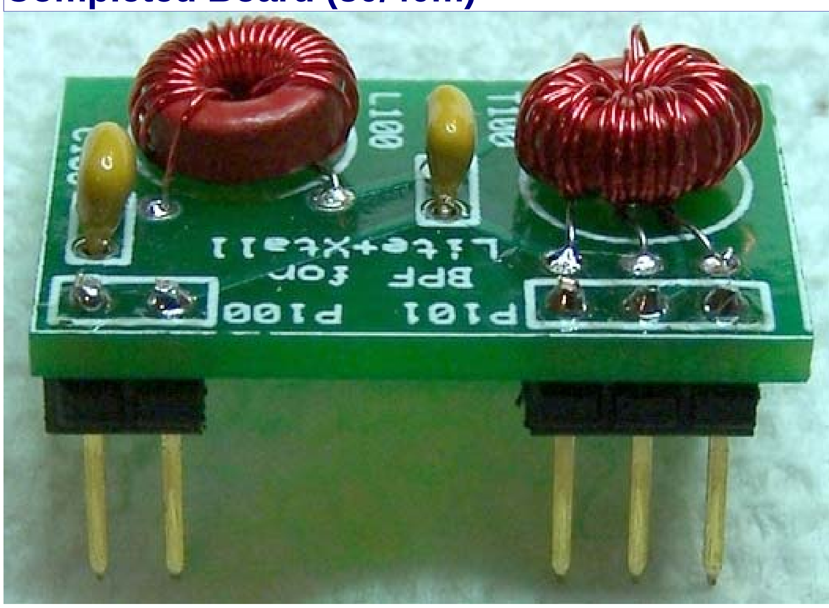
	C100-#	ceramic capacitor	Mount and solder the capacitor, C100
	C101-#	ceramic capacitor	Mount and solder the capacitor, C101

			
	P100-#	2 pin header	Mount and solder the 2-pin header, P100, on the underside of the board, with the shorter pins going through the holes from the bottomside to the topside and the longer pins extending out from the bottom side to mate with the main board [2].
	P101-#	3-pin header	Mount and solder the 3-pin header, P101, on the underside of the board, with the shorter pins going through the holes from the bottomside to the topside and the longer pins extending out from the bottom side to mate with the main board. [2]

1 The L-100 for the 160m BPF will require overlapping the windings in order to fit all of them on the toroid. The first layer pretty well fills up after 45 or so turns.

2
The BPF board connectors (P100 and P101 headers) are mounted, short ends into the holes for P100-# and P101-#, on the bottom of the board with the other components on top.
Use the main board 9-pin socket (J1) as a "tool" to align the pin headers on each BPF board so that the two will mate properly.

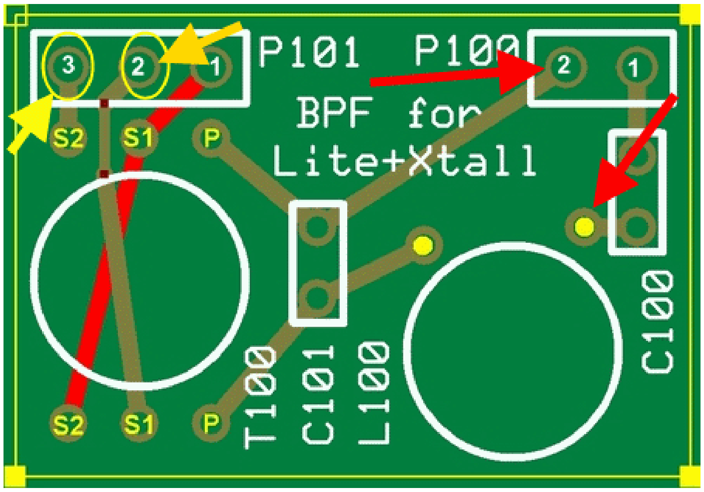
Completed Board (80/40m)



<!------------------------Detailed Notes Section------------------->

Testing

Continuity



Test T100 Primary Resistance

- Using your ohmmeter, measure the resistance from The C100 hole farthest away from P100 to ANT Return. It should be ~0 ohms, indicating continuity in the primary windings of T100, through the L100 windings.
- If you get any appreciable resistance or an open circuit, you should inspect/touch up the solder joints on T100 primary and/or L100.

Test T100 Secondaries Resistance

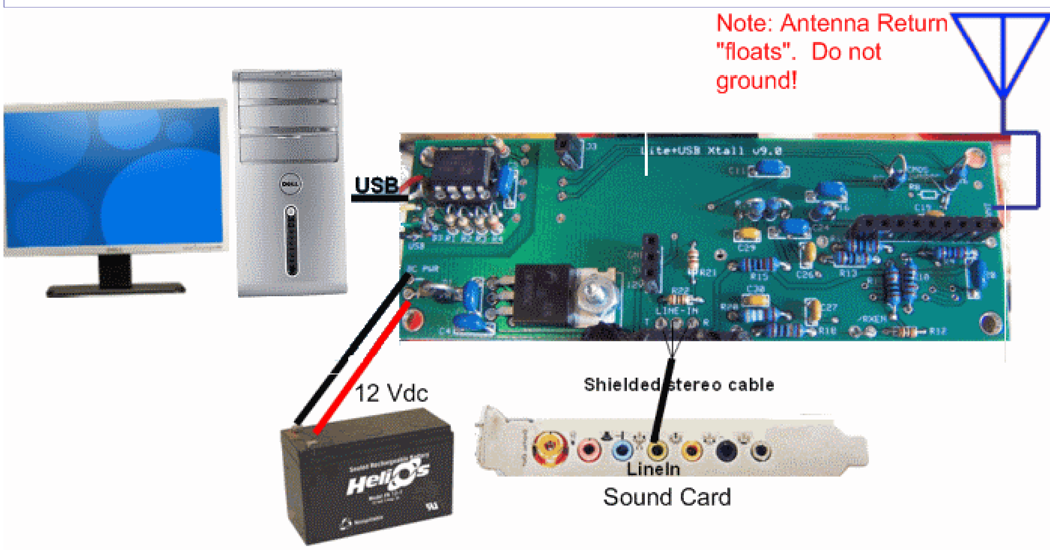
- Using your ohmmeter, measure the resistance between pins 2 and 3 of P101.
  It should be ~0 ohms, indicating continuity between the ends of the two secondary windings and through the center tap.
- If you get any resistance or an open circuit, you should inspect and/or touch up the solder joints.
- **Note: that the two secondaries are center-tapped so both windings are "connected" continuously in the circuit from pin 2 to pin 3.**

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD)
External Connections Comments Revisions WB5RVZ Home

Softrock Lite + USB Xtall V9.0 - External Connections

Home BOM Power Supply USB Control Local Oscillator Dividers RX OpAmp RX Mixer(QSD) RX BPF(s); External Connections Comments Revisions WB5RVZ Home

Introduction

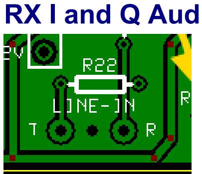


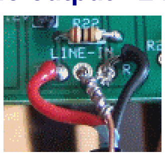
Summary Build Notes

- Connect the I and Q output lines
- Connect the Antenna
- Connect the power leads/connector
- Test the Stage

Detailed Build Notes

RX I and Q Audio output - LINE IN

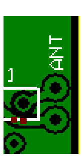


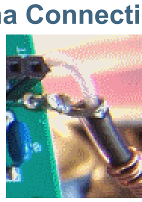


These are the Ring(Q) and Tip audio outputs of the board, located at the bottom center edge of the board.

Depending upon your ultimate enclosure/mounting requirements, you want to connect these three pads to good quality shielded 2 conductor audio cable, terminated either by a 3.5 mm mini plug or a mini jack.
Use a short length of solid hookup wire, soldered to the shielding and to the ground/common connection, and wrapped firmly around the outer insulation of the cable as a strain relief mechanism.

Antenna Connection





Sample Antenna Connection

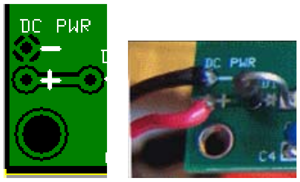
ANT/RET

These are the ANT and (unmarked) Return connections located on the right-hand side of the board, near the top.

Use RG-174U 50 ohm "micro" coax for the antenna connection, There is a good discussion of RG-174 coax and techniques for installing connectors available on the internet.

Finally, regarding the "floating antenna RET" connection, review the messages in this topic where the builder was getting no signal and the cause was the improper ANT RET connection.

Power Connection



Sample Power Connection

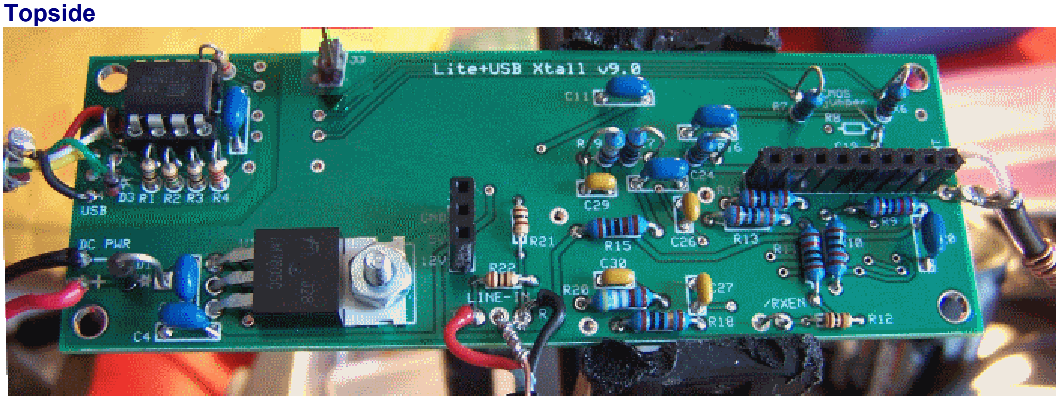
PWR

To power the v9.0 receiver you will need a 9 volt to 12 volt DC source at a little over 100 mA. A supply that is free of ground connnections works best.

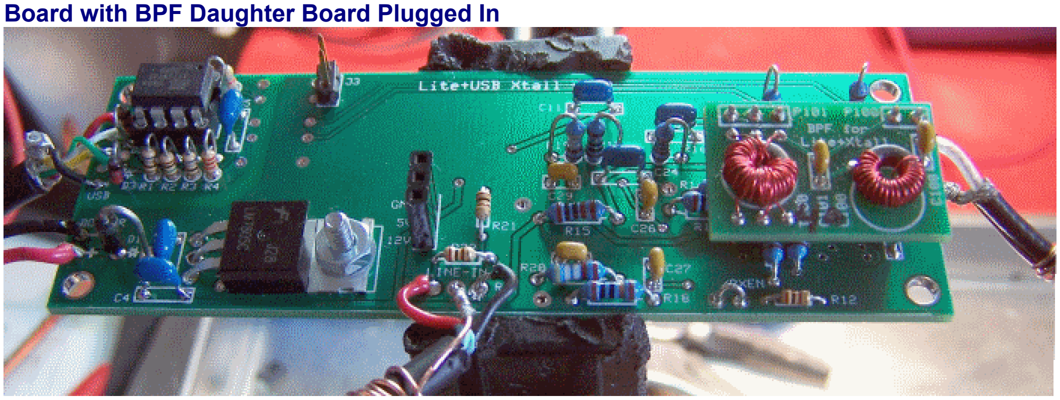
Use the conventional red/black wire for the power line +/- connections with the connector of your choice.

Completed Stage

Topside



Board with BPF Daughter Board Plugged In



Testing

Note: This stage test requires you to have built and plugged in at least one bandpass filter.

The test assumes you have built an 80/40m BPF.

Current Draw (DMM)

Since you have just installed the various connections, it is a good idea to check the current draw one more time.

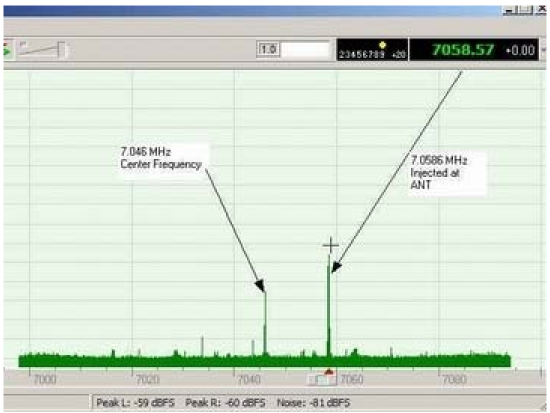
- **Current numbers here are for the CMOS version of the Si570. You will need to adjust these up by about 14 mA for the LVDS version.**
- **Power the board up (author has been using an 11.6 Vdc battery pack)**
- **Measure the current draw and 5 V rail voltage with a 1K Ω limiting resistor**
- **Measure the current draw without the limiting resistor.**

- **Remove power**

Testpoint	Nominal Value	Author's	Yours
Current Limited mA	6-10 mA	7.5 mA	______
Current limited 5V rail	1-2 Vdc	971 mV	______
Non limited draw mA	90-100 mA	97.0 mA	______

RX Test in Rocky

The ultimate test is to run Rocky, feeding the Ring and Tip outputs to the Line In inputs of your PC's sound card. You must have built at least one of the Bandpass Filter Boards (BPF) to conduct this test. The values below are appropriate for the 80/40m BPF Board.



With the 80/40 BPF board in place and a stereo cable installed (see above) for the ring, tip, and common audio output connections:

- **Follow this sequence to connect the board and the PC (important note: there have been cases where this sequence was not followed and damage to the board resulted)**
  - **Plug the audio cable into your sound card's Line-In input**
  - **Power up the board**
  - **Plug the USB cable into the PC**
- **Run Rocky, click the `File > Start Radio` menu choice, and click on the `View > Settings` menu**
- **Click on the "Audio" Tab and select your sound card**
- **Set Rocky's center frequency at 7.046 MHz (if it is not already selected) in the "I/Q Input Device" dropdown box).**
- **Click on OK to close the "Settings" Menu**
- **Set up your transceiver (or other signal source) to transmit a low power signal at 7.059 KHz into a dummy load and loosely couple it to the board with a short wire**
- **Click on Rocky's `File/Start Radio`**
- **You should see the Rocky spectrum display resembling the image above.**
- **If your signal source can sweep the frequency, observe Rocky's spectrum display as the generator sweeps through the "chunk" of bandwidth centered on the center frequency.**
- **If you see an unwanted "mirror image" of the desired signal, you may want to check out the image rejection hints on this website.**