**Appendix 1 - Hardware BOM**

1. PCB parts – see Multiband LO V1 PCB spreadsheet for complete parts list with cost, Many with DigiKey Part numbers and quantity
2. Chassis box of your choice
3. 10MHz OCXO (optional) - surplus
4. Cypress PSoC 5LP CPU mini dev board (CY8CKIT-059) (Digikey or direct from Cypress Semiconductor)
5. DigiLO PLL (standard or high stability TCXO) (Down East Microwave)
6. Programmable step attenuator 31dB (5V, serial latching)
7. Solid state example tested: https://www.ebay.com%2Fitm%2F172351799000
8. SP6T coax switch options. Could also use other sizes such as SPDT, SP4T, SP8T.
9. 3GHz RF Switch SP6T (5V, 3 line BCD): https://www.ebay.com%2Fitm%2F282746720503
10. Surplus SMA coaxial switch, SP4T or SP6T. TTL preferred but 12V works also, Ground to select each port (low side drive, not 12V to select a port aka high side drive otherwise a level shift will be needed externally).
11. Rotary Encoder with integral push button. Could also use a Rotary Encoder without an integral pushbutton and use a separate push button.
12. Single or multiple PowerPole connectors. Extra ports are handy to redistribute low current 12VDC such as when on a tripod and powering transverters, IF radios, OCXO, GPSDO, weak signal sources, and such.
13. Fuse and polarity protection diode (1 or 3Amp each as appropriate).
14. 6GHz wideband amp (5V 20dB). <https://www.ebay.com/itm/163622209490>
15. 2x16 LCD display with low power drain backlight (dimmed by the CPU). Standard Hitachi compatible. I used a white LED backlight version with only 20ma vs 100ma for yellow or some others. Can use a bezel for a cleaner look. See BOM spreadsheet for one example.
16. LCD Serial interface: For a serial interface I am using a PCF8574T based LCD backpack. There are other LCD chips and backpack chips but they will require software library changes, or just go with the 7 wire parallel interface. Many LCDs come equipped with the serial backpack. <https://www.amazon.com/dp/B01MXGXPKU/>
17. GPS connection is TTL level only. I am using the QRP-labs QLG1 GPS module  
    <https://qrp-labs.com/qlg1.html>
18. AD8318 RF Logarithmic Detector 70dB RSSI Measurement Power Meter 1-8000MHz  
    <https://www.ebay.com/itm/401741283569>

**Appendix 2 - Software Installation**

This section covers the steps to install the PSoC Creator 4.2 Integrated Development Environment (IDE) and the community library for the serial LCD driver. This is required to program a new PSoC 5 LP CPU dev module (CY8CKIT-059). The CY8CKIT-059 has a snap off programming board with USB connection. Normally for space considerations you would want to snap it off and use a 5-wire jumper for future updates and development if you are into that. If you have the space you can leave it attached. There are some CPU board changes required. See Appendix 6 – CPU Module changes required (CY8CKIT-059).

Download and install PSoC Creator IDE from  
<https://www.cypress.com/products/psoc-creator-integrated-design-environment-ide>

Download the zip file Multiband\_PLL\_LO\_PCB\_V1\_I2CLCD-000.cywrk.zip or latest version zip file from the online folder and extract it to the PSoC Creator folder located in your Documents library folder.

In the IDE Open the project with File-> Open-> Project/Workspace and navigate to the new folder and select the .cyprj project file. As of this writing it is named Multiband\_PLL\_LO\_PCB\_V1\_I2CLCD.cyprj.

Community Code installation

Serial I2C LCD Community Code is at <https://community.cypress.com/thread/11727?start=0&tstart=0>

V2.1 July 2018. is last known release as of May 2019. Initially used V1.5 library then found the 2.1 library. I have provided a copy of 2.1 on the same share as the rest of the code for this project.

The library includes a PDF data sheet (you can search the library zip file for CharLCD\_I2C\_v2\_1.pdf) which has installation instructions, so the library appears for each project. Follow the steps under the section named “Loading Community Components Using Dependency Method”. I found that I had to save and exit the IDE and upon IDE app restart, it asked if I trusted it the community module. After answering Yes, it started up without error and the Community table appeared as expected. It is possible the trust dialog was hidden behind other windows and if found, you might be able to answer and continue without restating the IDE.

Adding the first community code adds a new tab in the Component Catalog window when you have TopDesign.cysch (the project’s schematic editor) open. Be sure to follow the guidance on naming the LCD and I2C LCD driver and assign the serial backpack I2C address correctly. That work is already done in my code, just be aware should you dapple in that area or use a LCD Backpack board with a different address. The address is defined in the Multiband\_PLL\_LO.h file. Look for the line with   
#define LCD\_MODULE\_ADDRESS (0x4Eu);

The address for I2C devices may be expressed as 7 or 8 bit format with the 8 bit being a left shifted value, so pay attention to which number format you use in each location needed.

In the IDE, under Project, open Dependencies

Under user dependencies if you see CharLCD\_I2C, the 2 boxes should be checked. If not check them. If you see a red alert marking and both boxes checks to the right, then the community library is not installed, or not properly.

CharLCD\_I2C\_v2\_1.pdf has the installation step details.

The last thing to check is under Project -> Build Settings

Open the ARM GCC section and select Linker. Additional Libraries should say m and Use newlib-nano Float Formatting True. The other field are left at default.

Open Project-> Update Components dialog box. Cy\_boot 5.80 should show as Downloadable, likely a few others also. Hit Next button and then the Finish button to update the component. Not doing this will result in a build error.

Periodically the IDE auto Update tool will pop up offering to update components including the programmer, libraries, and IDE parts. These are updated often and generally want to update.

Under Build menu, select build and things should compile. If it has not seen your CPU module plugged in yet, then it may ask you to select a platform. It is easiest to plug the USB cable to the CPU module and let it be discovered.

Appendix 4 has the programming instructions once the software is installed and configured in this Appendix.

**Appendix 3 – Configuration Mode Screen Menu Table**

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
| Menu Label | Function | Type | Data Allowed | Notes |
| Version | Displays Version number and date |  |  | Hardcoded in the firmware. |
| Callsign | Display and edit Callsign | Alphanumeric scroll list, push to Edit. | Alphanumeric to 7 digits max | Value saved to EEPROM immediately. |
| Auto Enable | If Enabled, Band Decode Input is read and the band slot is changed to the first matching configured pattern. | Push to Toggle State | YES or NO | Assign input patterns for each band slot in the Config -> Band Decode Input menu.  Value saved to EEPROM immediately. |
| Configuration | Access to config menu options | Push to change to Configuration Menu |  | Most items changed are not automatically saved. Use the Config Save menu to preserve all changes else some changes may be lost at next power up. |
| Band Name | Descriptive name for current Band Slot. Limited to 12 characters. | Free Form, letters, numbers and symbols. | Pick from the scrolling list one character at a time building up your label name. | Any of the 7 band slots can be assigned to any frequency. Choose a unique descriptive band name. You could include WSS in the name for weak signal source frequencies vs, and LO frequency. |
| Coax Switch | Assigns a switch port to the current Band Slot. | Select from a list of 17 possible values that match your LO switch. | 0-6. A value of 0 usually disconnects all switch ports and can save power. | Maps the LO frequency to the transverter LO input or WSS antenna cable via the LO switch. |
| Frequency | Select a preprogrammed DigiLO frequency value for the current Band Slot. | Select from a list of up to 255 values | Values are within 25 to 6GHz range | Real time value changes. The V1 PCB connector requires the connector to plug into the underside of the DigiLO PCB to work properly.  Real time value changes.  The Power detector level is displayed in the lower right corner while setting a new value IF the level detected is > -15dBm. |
| Attenuator | Step Attenuation value for the current Band Slot. | Select from a list of 32 possible values (1 dB steps) | 0-31dB | This version software assumes a specific 3 wire serial latched attenuator module is used. Other version permits a 4 or 5 wire parallel port connection.  Real time value changes.  The Power detector level is displayed in the lower right corner while setting a new value if the level detected is > -15dBm. Normal full LO output with amp inline and ATT = 0 would be about +20dBm at 1296Mhz. |
| Band Decode Input | Pattern selection to match your radio’s output for each band when Auto modes is enabled. | Select from list | 0x0 to 0xF | Used in AUTO mode only. Ignored in Manual mode (Auto Enable = No). |
| Remote Antenna Output | Pattern selection to match your antenna switch input for each band. | Select from list | 0x00 to OxFF | Changes to value configured per band slot regardless of Auto mode. |
| LCD Backlight Level | Choose a LCD backlight level. | Select from list | 0-100% | Realtime value change. |
| GPS Setup | Choose Baud rate or Enable/Disable GPS Mode Screen |  | 4800 or 9600 bps or Toggle Yes/No | TTL level interface only on connector. GPS Enabled = No skips the GPS Info screens.  Value saved to EEPROM immediately. |
| PTT In Hi/Lo | Polarity choice for PTT input from radio. | Select from list | Hi or Lo | Tx or RX displayed in upper right area of screen. |
| PTT Out Hi/Lo | Polarity choice for PTT output from LO box to transverters/amps. | Select from list | Hi or Lo | Linked to the PTT In, but polarity is independently configured. |
| Config Save | Saves all current active configuration values. |  |  | Upon next power up all values will remain the same as was active before power down. Some values are saved immediately or on mode change as noted in each menu text notes. |
| Factory Reset | Marks the EEPROM invalid. Next power up will copy default values into EEPROM. |  |  | Upon next power up all values are returned to the defaults in the firmware. |

**Appendix 4 - Programming a New CPU module**

1. Start PSoC Creator program
2. Connect a USB cable to the programming board.
3. The programming board should be connected to the CPU module by a 5 wire cable or it is still attached to the CPU module.
4. In PSoC Creator open file and load the Multiband LO project file (.prj file)
5. Configure Build Dependencies for libraries if not already done (See Appendix 2).
6. Math library (See Appendix 2).
7. Community Code (See Appendix 2).
8. Update components (See Appendix 2).
9. Other such as device selector
10. Build project
11. Debug (which will program and run)
12. If all is well, stop debug and power off the device
13. Disconnect the programmer cable
14. Power up and verify it starts up and runs OK.

**Appendix 5 – Online Documents, Photos, Videos, Drawings**

https://1drv.ms/f/s!AiKFVkEi7IjLoq0z4Jk7tv5d9ZGrTg contains the latest online pics, a menu walkthrough video, PCB schematics, BOM (an Excel spreadsheet), PCB layout images, PCB board file, schematic files of the PCB and printout of the CPU related configuration schematic pages from the IDE. The blue enclosure version in some pictures is the vector board prototype, the gray plastic enclosure is the PCB version with solid state components. The remote antenna switch build is also pictured and parts are listed in the BOM also.

The BOM is an Excel spreadsheet and is a compilation of Digikey, Amazon and eBay orders. For the first version build I dove deep into my junk box, less so for the PCB version so some parts have no part number supplied.

**Appendix 6 – CPU Module changes required (CY8CKIT-059)**

For new boards you must remove some parts, or they will interfere with proper operation.

Remove R20 0ohm jumper to separate programmer power from the USB port.

Remove R5 0ohm jumper bypassing 2200pf cap.

Solder in J4 – a 2Pin header.

Remove the 1.0uF cap C7 (intended for A/D demo). Affects the LCD operation.

Solder in 5P Molex for SWD programmer cable.

Break off the programmer board section. I score both sides of the board with a sharp knife to make it a bit easier to snap off. The programmer board is capable of supplying voltage to operate the CPU if a shorting block (jumper block) is installed on J4. USB power is not enough to power the whole box however. The jumper on J4 is normally left off and programming is done.

Appendix 7 – PCB

Board errors **on V1.0 Only**. Fixed on V1.1 boards

5VDC regulator pins located at the edge of the board are shorted to the ground plane on the bottom side. Use a sharp knife or Dremel tool to isolate the 2 outer pins. One is 5V and the other is 12V so applying power before isolating these will not turn out well. See Figure 9.

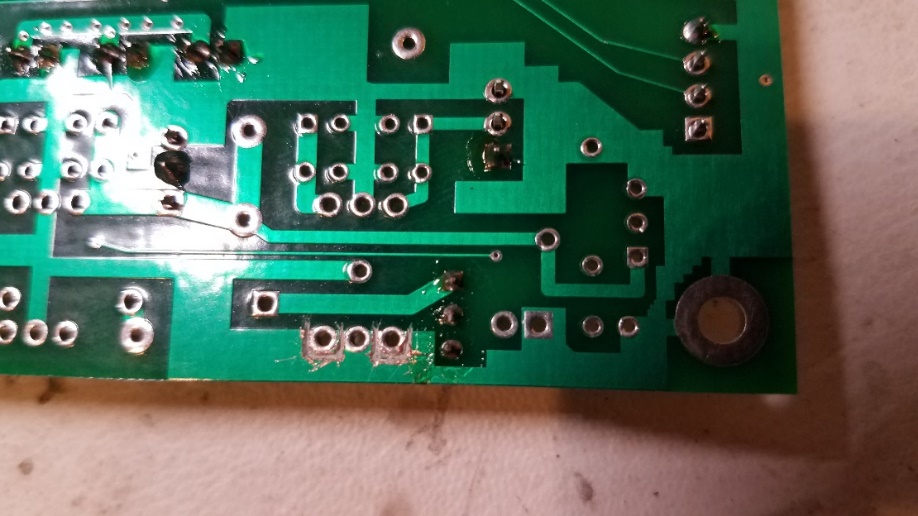




Figure 9: 5V regulator bottom side PCB traces shorted. Isolate the 2 pads with a knife.

Missing GND plane section causing interrupted GND connection to part of the board. Add a jumper wire. See Figure 10.

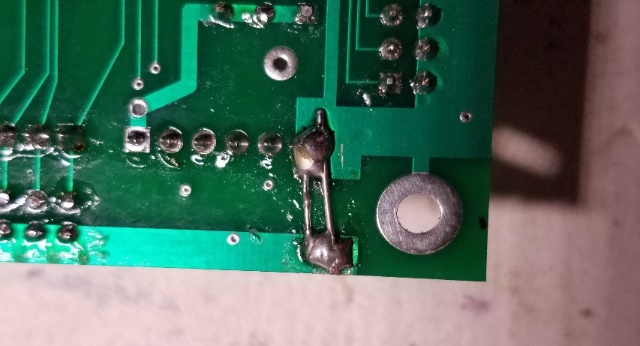
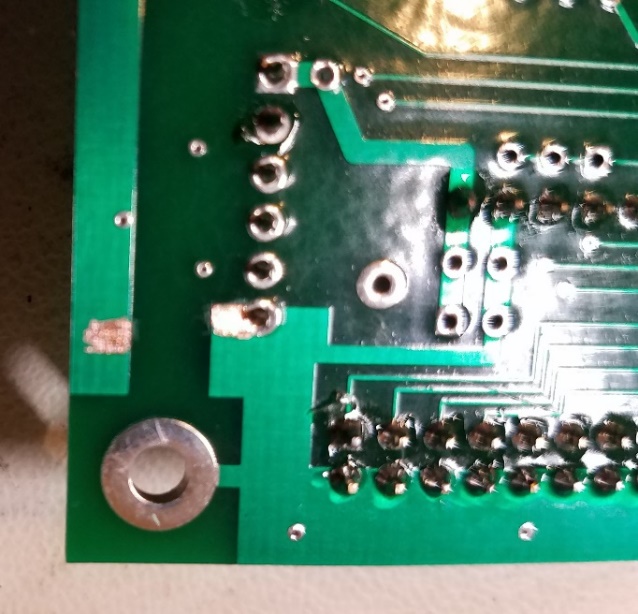
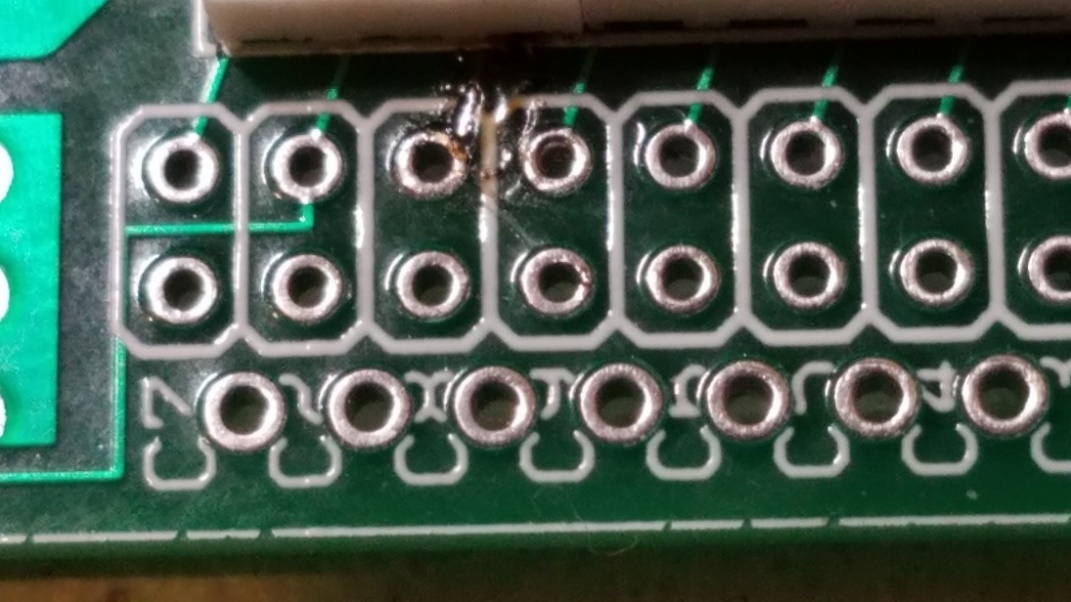




Figure 10: Ground plane missing connection. Solder a jumper wire after scraping the solder mask clear.

A control wire connection trace to the pad for C8 has a gap, so needs a small jumper. See Figure 11.





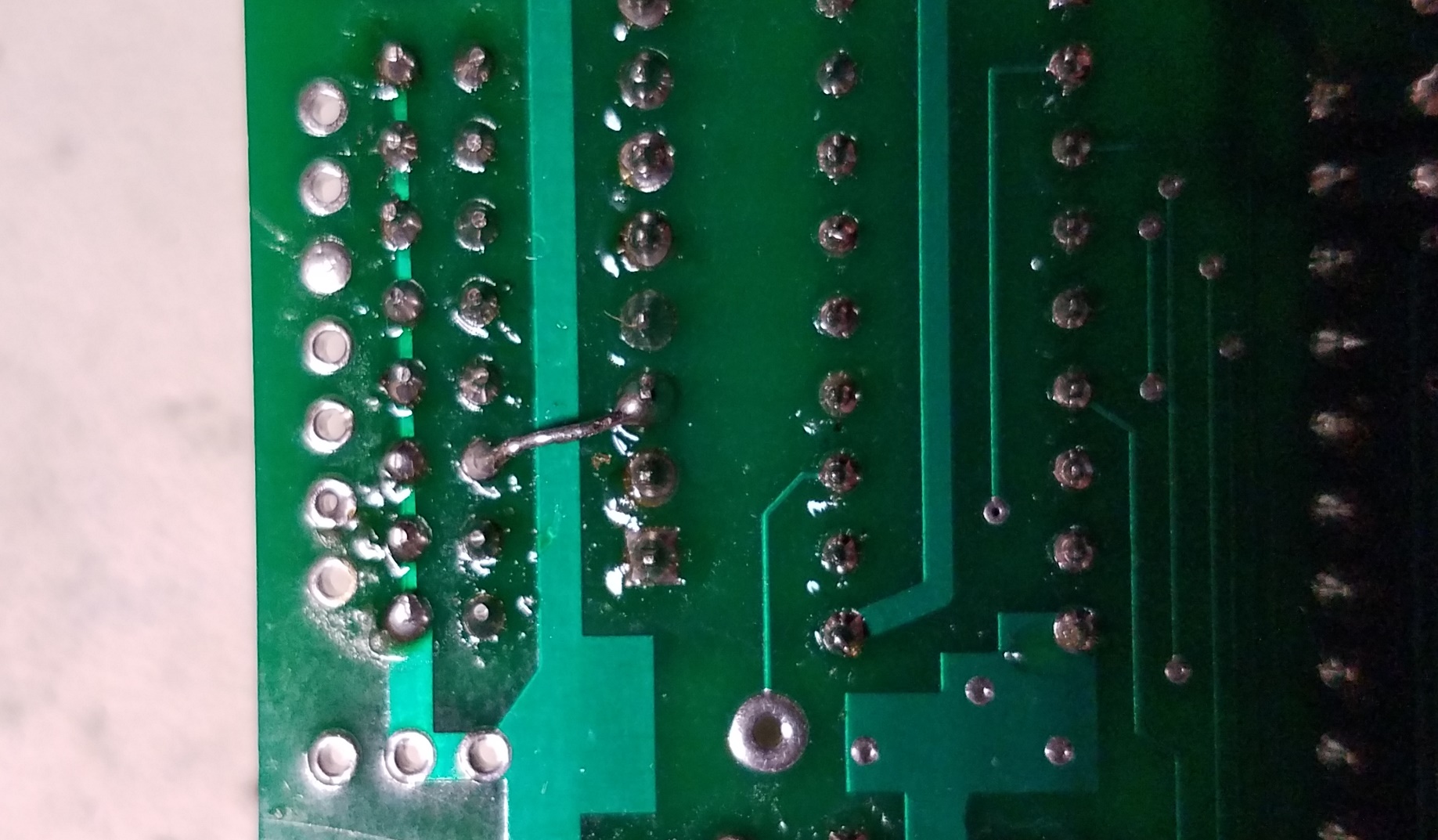




Figure 11: Trace to C8 not connected. Solder a short jumper wire on the bottom side as shown.

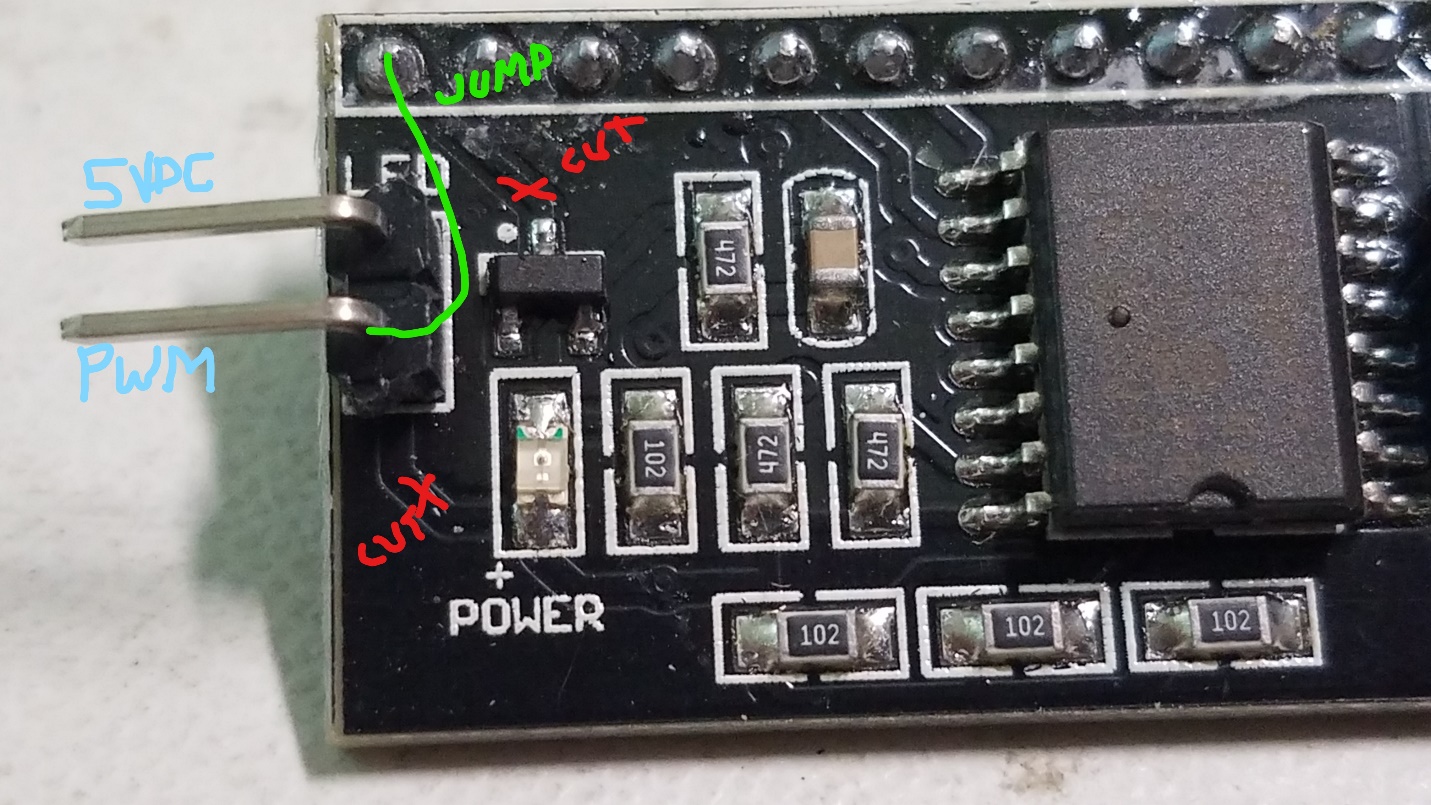


Figure 12: LCD Backpack PWM Modification Details

Pin 16 on the LCD is GND or in this case switched ground. Pin 15 is +5V. The Green jumper wire is connecting the PCB PWM driver to the LCD backlight ground pin 16. +5V is supplied on the other pin which runs to the LCD pion 15. Ensure your LCD Pin 16 and pin 15 is isolated from the LCD main power supply. Many LCD have jumpers for backlight power options.

Appendix 8 – PCB Layout Images for V1.1 boards

Below are the Silkscreen, Top and Bottom images of the V1.1 PCB which has the 3 PCB trace errors corrected and extra holes to use either vertical or horizontal contract pots and is otherwise the same. The ExpressPCB files in the online folder or V1.1 so you can easily load them up into ExpressPCB layout program and order them online.

The actual board size is 3.0” x 3.8”. Standard double sided fiberglass with optional solder mask and silkscreen. For the 26 pin CPU sockets I used combinations of 10, 16 or 6 pin single row sockets as they are easier to find. A little filing on the ends may be required however to test fit before soldering. The V1.1 board can accept either a vertical or horizontal single turn pot for LCD contrast control at R4.

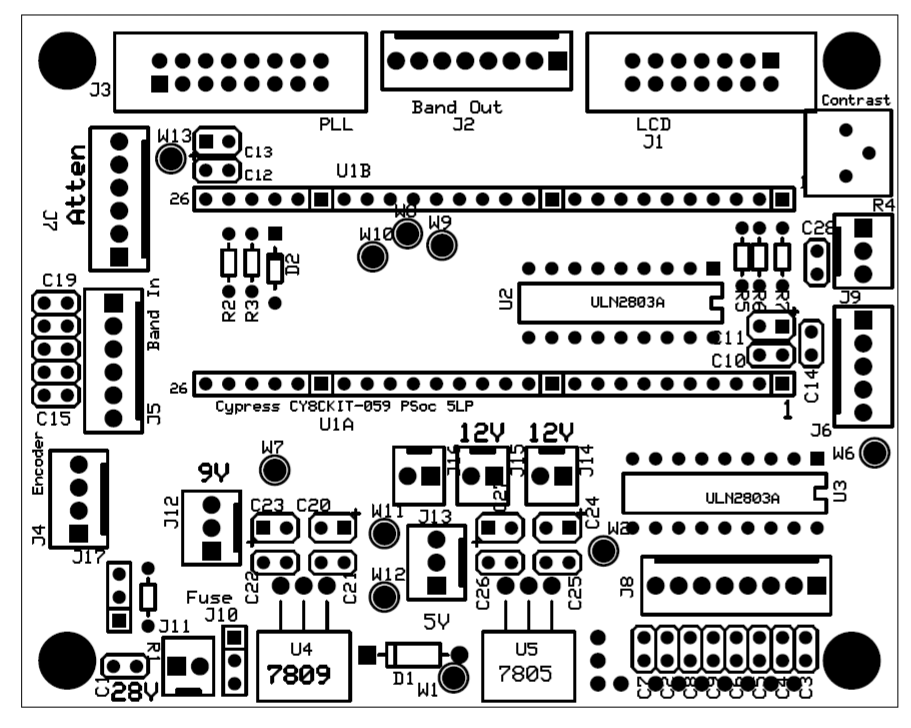


Figure : Silkscreen layer

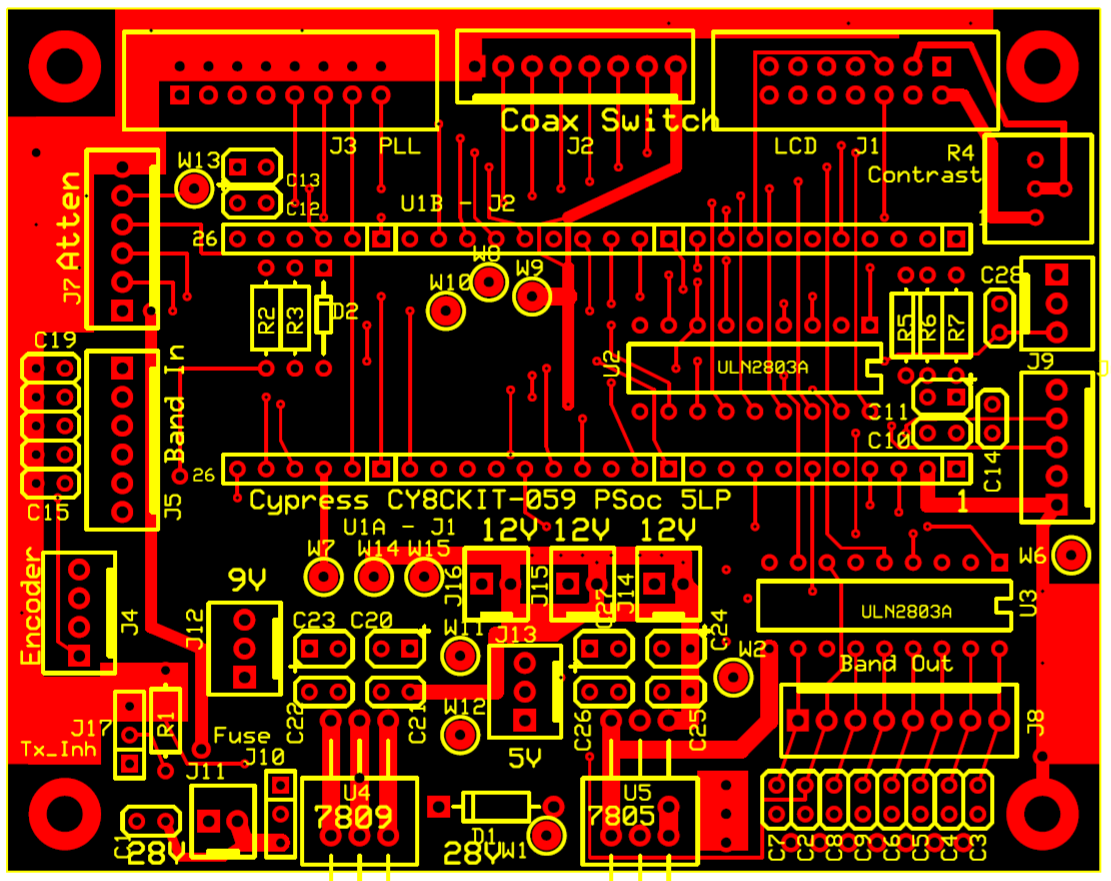


Figure : Top Copper and Silkscreen layers

The 2 regulators may be mounted in several ways to suit your choice of chassis. Extra holes are in parallel for each, one set at the edge, another inboard a bit. You might use the inboard set to mount the regulators under the board and bolt to the chassis bottom. The mounting holes will be exposed just past the edge of the board. The edge holes are for mounting against a vertical side wall of a chassis or larger heat sink. For a low power version you can just use clip on heat sinks like I have shown in my pictures.

How much power you consume is highly dependent on your choice of mechanical vs solid state modules and LCD backlight current. In my low power version I used a smaller plastic case vs metal, used all solid state modules, no 28V or remote antenna switch, and no OCXO, opting instead for the high stability TXCO version of the DigiLO and so only draw about 100ma total. I used a low dropout 9V regulator for battery power while on a tripod I the field. I repurposed the 28V circuit to be the 12V input protection circuit. It consists of J11 for the 12V input, a 1A pico fuse at J10, a diode D1 (flipped around) and a jumper form W1 to one of the spare 5 or 9V regulator 12V side holes.

In my home/mobile version it had all the goodies, and all were mechanical. Including the remote antenna switch, it draws between 1-2 amps depending on OCXO temp and the number of internal attenuator relays engaged. I used a 3 amp glass chassis mounted fuse.

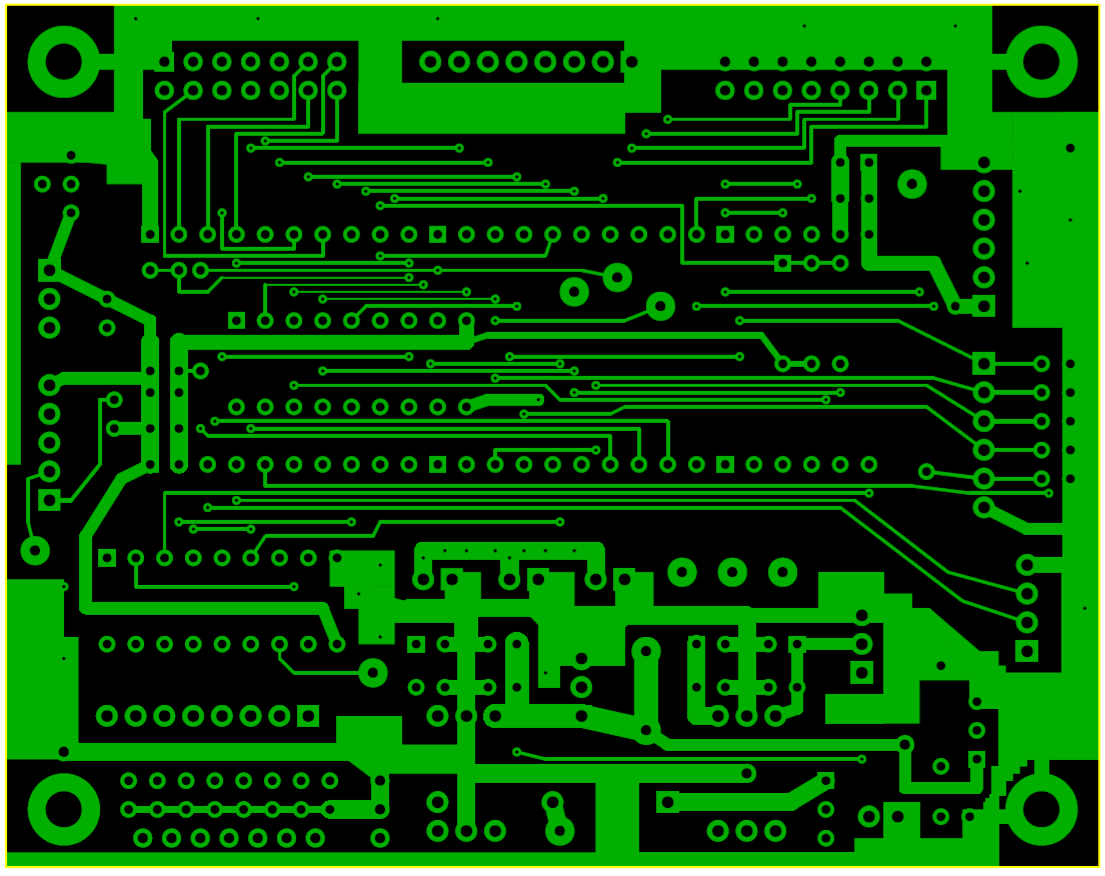


Figure : Bottom Copper Layer

Appendix 9 – Remote Antenna Switch

This is a fairly simple option. I used 2 identical 28VDC SP6T coax switches connected in parallel. These use 24 or 28VDC on the common terminal and GND on one of the 6 port terminals. A ULN2803A (U3) is provided to drive each terminal up to 500ma. You need to jumper U3 to your choice of power for the suppression diodes to work properly.







Figure : External Remote Antenna Switch box views

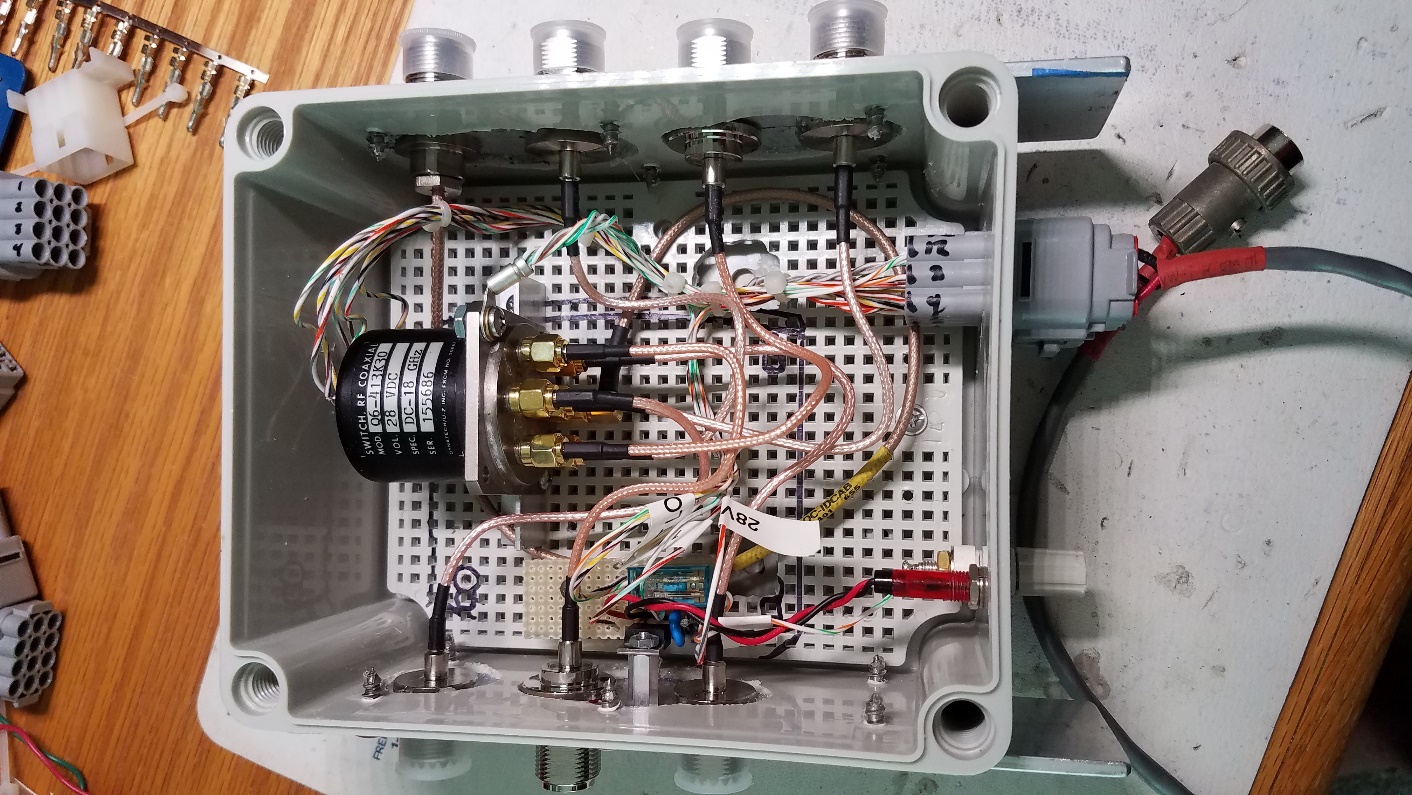


Figure :Inside look at Remote Antenna Switch

Short RG-316 SMA to UHF and SMA to N bulkhead cables were used. 1/8” x 2” aluminum flat stock was screwed down to the box sides (with silicone sealant under) to help bond the connectors to ground better and act as a heat sink for the 12V regulator. The 2 sides were further connected with heavy wire using electrical grounding compression fittings and tied to an earth ground rod.

There is a small PCB with a relay and 12V regulator bolted to one side. PTT Out keys the relay to provide PTT switched GND and +12V for preamps and remote T/R relays. The switched 12V also powers a LED visible in the bottom of the box to verify PTT switching is working.

A long 9 or 10 conductor cable (I used sprinkler wire) is connectorized with waterproof automotive type 12Pin connectors. These can be found in bulk online and are very affordable. You could use 915MHz long range radio data transceivers to replace the cable so only power would be required at the remote end. This is a future goal of mine for enhanced lightning protection and can be more cost effective for long wire runs. I used a 24VDC 850ma DC-DC converter in the house to power the remote system components including the preamps via a 12V 7812 regulator with 4 or 5 1N4007 diodes in series to drop the voltage some to reduce heat.

The preamps, the remote antenna switch and lightning arrestors are all bolted to a common metal panel with lightning arrestors on cabling headed to the house. The box sides were bonded to the metal plate and to the ground rod system.

The box pictured is in use in Florida in a mostly shaded area at the base of a tree next to the house. The 12V 7812 regulator is mounted to the external aluminum flat stock via a ¾” aluminum standoff. The flat stock does get quite warm so heat is transferring OK and has been working for 2 months in 90F temps and partial direct low angle sun, but I suggest beefing up the heat sinking or use more diodes to drop the voltage, I think it is too warm as is for long term reliability.

A link to the source of the automotive connectors I found are in the BOM Appendix and the BOM spreadsheet.

Appendix 10 – PCB Schematic for V1

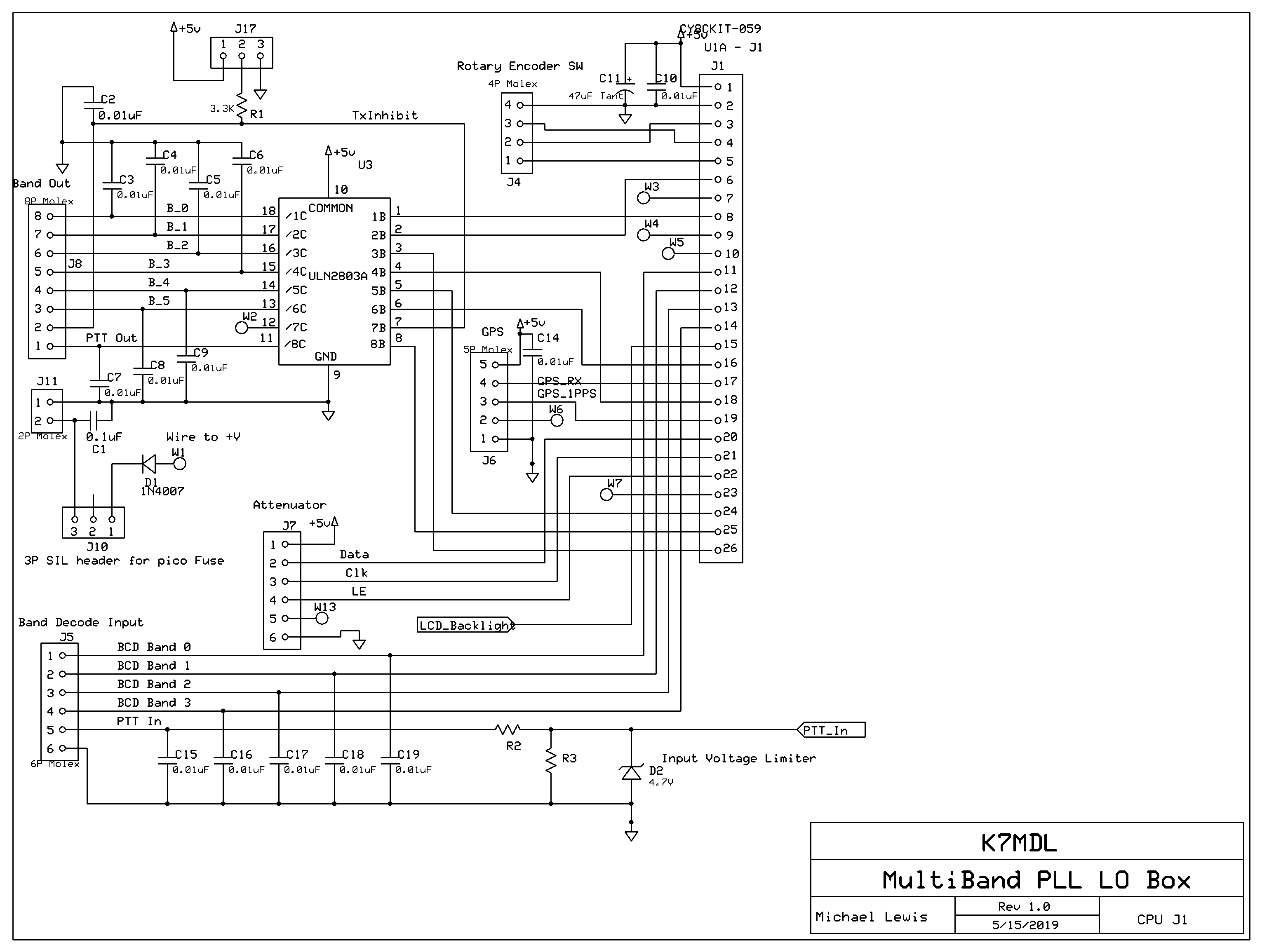
This is only the PCB portion of the project. The PSoC Creator IDE files contain more schematics. Those drawings are somewhat generalized when it comes to the external circuitry as it is only there to better understand what the CPU is connecting to. The PCB drawings are the ones to refer to in 98% of the cases. The CPU internal circuitry however is 100% accurate as the IDE uses the drawings to configure the internal hardware and map pin assignments in another screen.

Figure : Page 1 of PCB Schematic

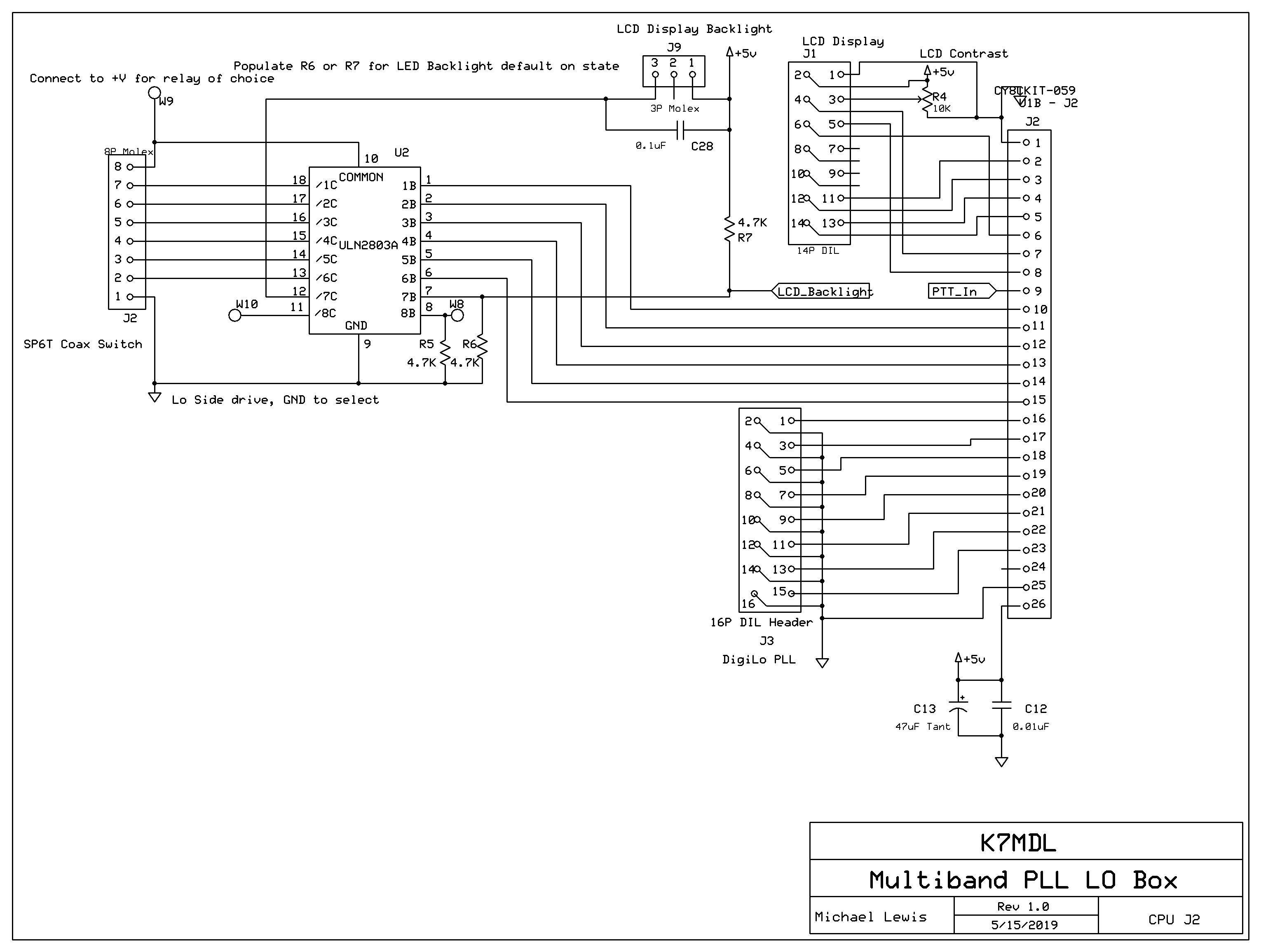


Figure : Page 2 of PCB Schematic

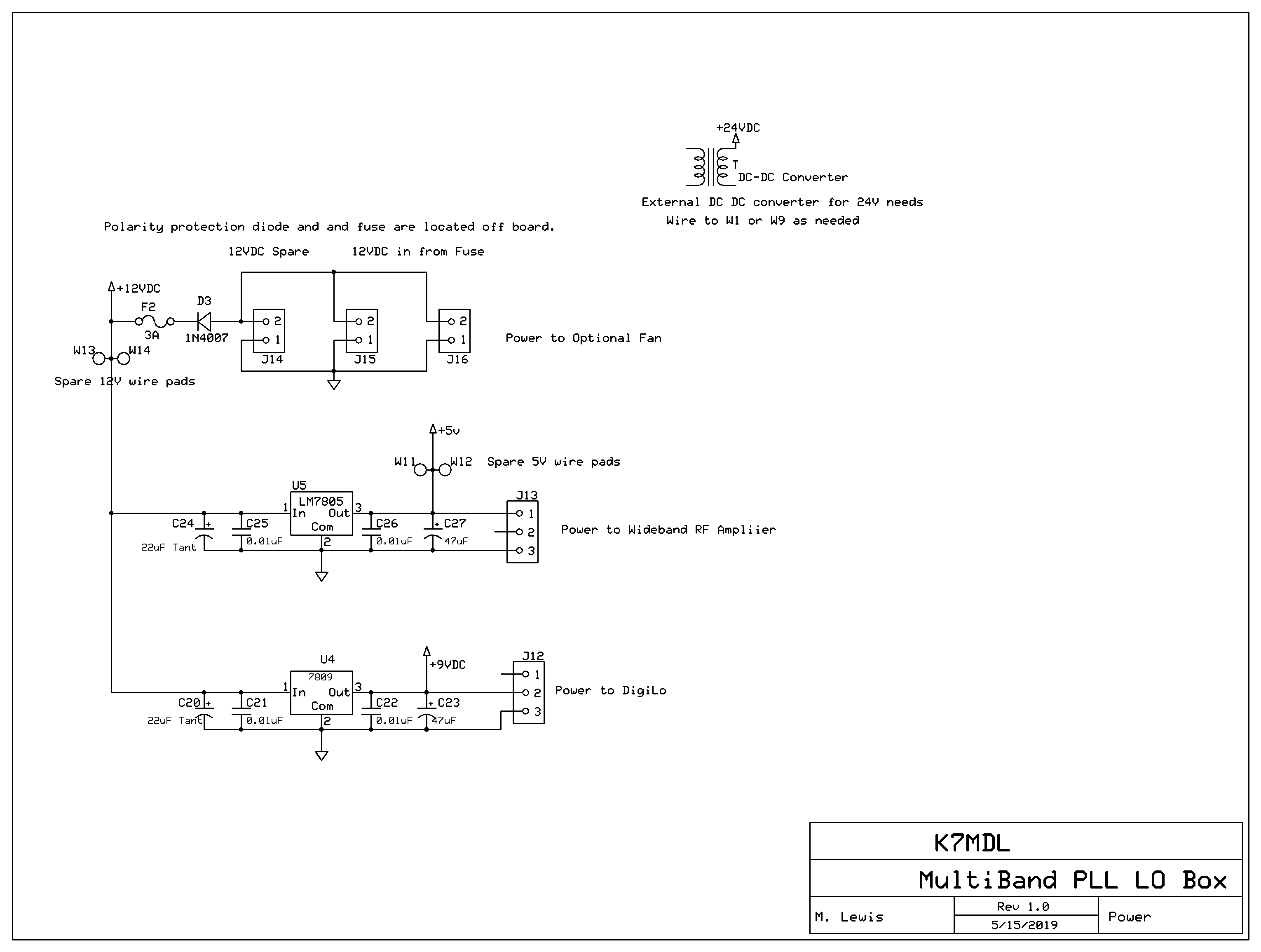


Figure : Page 3 of PCB Schematic

**Appendix 11 – Feature Changes since first release June 27 2019**

9/25/2019 – Added code for GPSDO functionality and an ADC on port 3:2 for analog voltage measurement. The GPSDO code is ported form another chip platform and is considered mostly done but not yet debugged and is not active code.

The ADC was needed for adding FT-817 and similar radio analog band select output reading. This caused a resource shortage in the hardware side so the GPSDO was branch off and mostly removed, some bits will still show up for later work. Soon after it was decided to use the ADC for reading the output of an inexpensive Log Power Detector based on the AD8318 chip. Code was created adding a power measurement in dBm in the Frequency and Attenuation config menus. These menus update the setting real time and so you see the final output power from the LO chain if you attach the detector to one of the RF ports. The power readings will only appear if the Bandname assigned a band slot is “POWER” (Case sensitive) or the power detected is > -20dBm. Be sure to assign the correct RF switch port for that band slot. I used port 6. If you use the power meter with other bands then the code checking the name matching “POWER” can be bypassed and displayed always. The header file has some constants defined for calibration purposes. Later these should appear in a config menu.