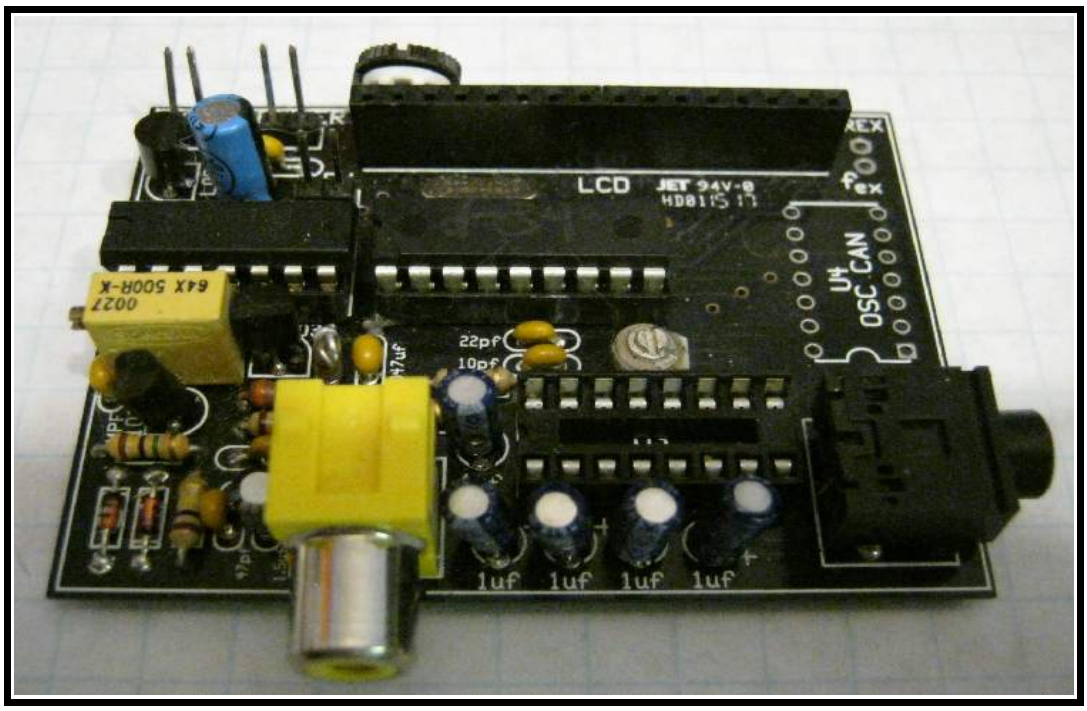


Builder's Guide

FDIM 2015 Buildathon
Frequency Counter Module

(PRELIMINARY)

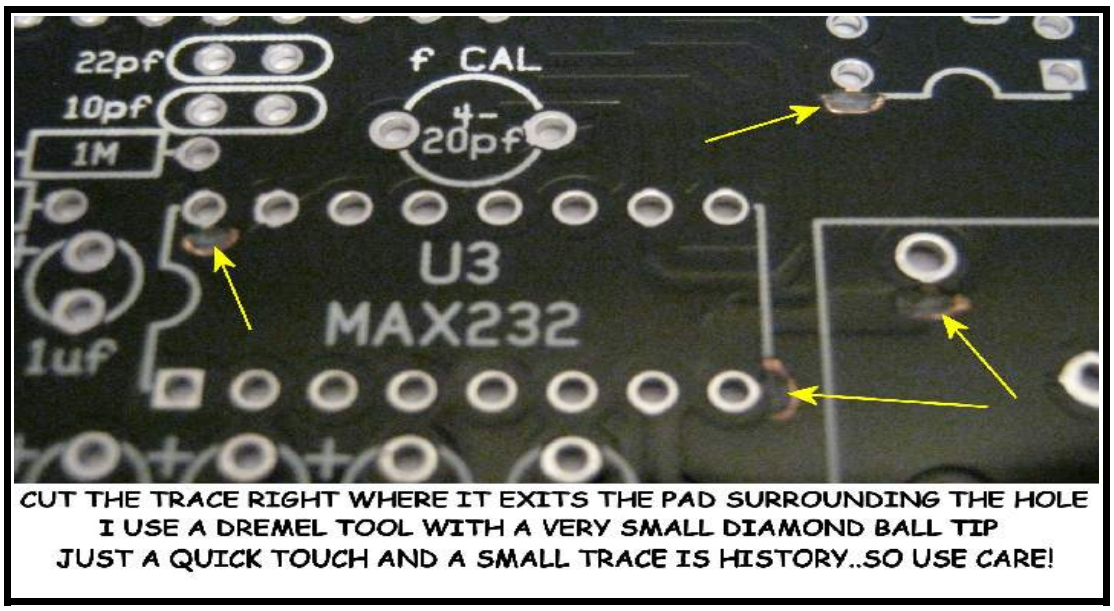
September 20, 2015

Rex Harper W1REX

FIRST off, there are 3 printed circuit board errors that need to be corrected. I accidentally ran 3 traces at the last minute before sending the files off to the FAB house and forgot that the top and bottom flow layers were turned off. So 3 traces ran off their respective pads right into the top and bottom ground planes.

For those of you who got their boards in Dayton at the Buildathon, you will need to make some pcb trace cuts to make the frequency counter module work. For those who received their kits in the mail, I have already made the pcb cuts, but you will still have to add the mods on the back side of the board to make it work properly.

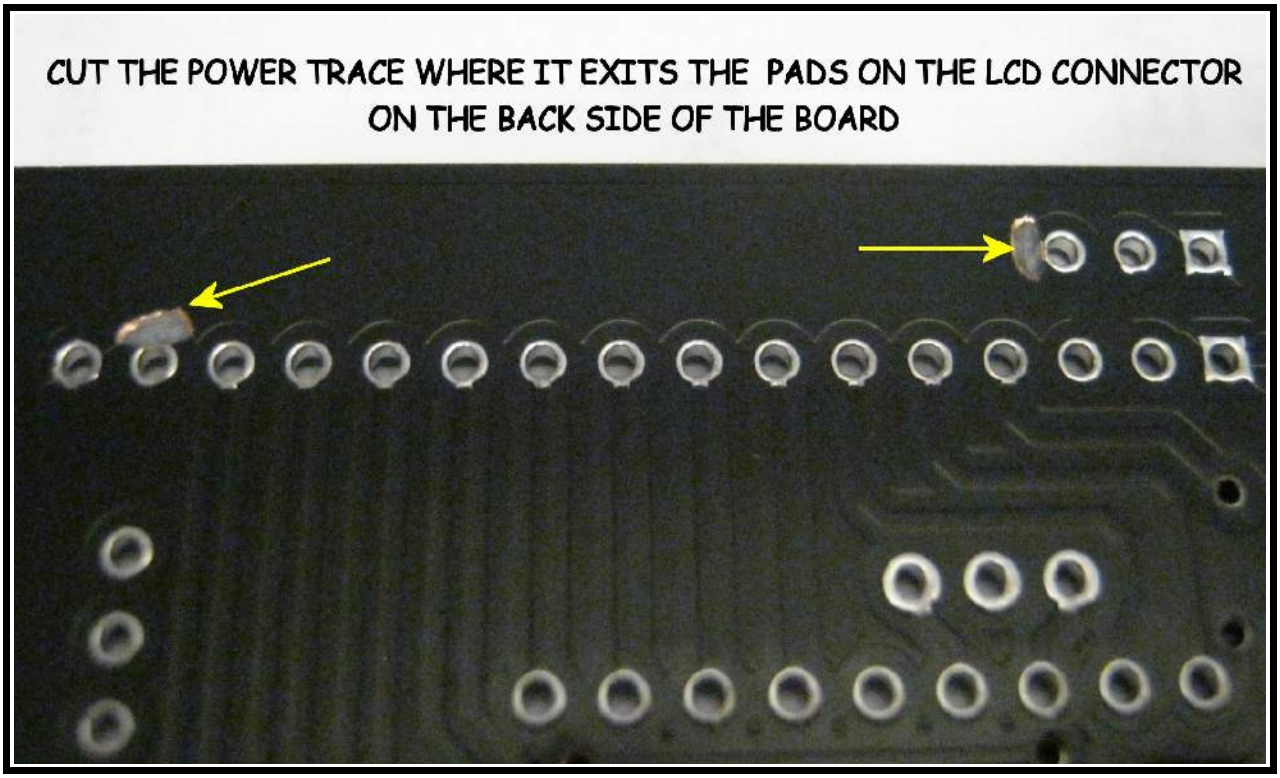
I use a Dremel with a remote hand wand attachment and a very small diamond ball cutting bit. This is what the PROs use to make pcb mods. They might use a smaller more specialized bit, but I have found this readily available bit to work just fine. On the frequency counter pcb, you need to cut traces as they pass through the unflowed area around the pads, just before they enter the copper flow area which happens to be the ground plane which will short the 3 signal lines to ground. Which, of course, is a bad thing....



The first pair of trace cuts are on the TOP side of the circuit board where the +5 volt power trace leaves the MAX232 chip, on Pin 16 and enters the OSC CAN socket on Pin 14. From the picture above, you can see the grind marks where I ground off the small trace just as it exits the actual solder pad.

You will have to make a replacement trace on the back side of the circuit board using hookup wire.....BUT.... only if you intend to replace the 4 Mhz crystal with a more accurate oscillator can in the socket. I haven't found the need so I haven't added the replacement wire on my board. As my young daughter used to say: "But maybe...!" In the future, you might want to used the oscillator can.

The second pair of trace cuts are where a RS232 signal trace leaves the MAX232 chip, on Pin 8 and enters the upper pin on the stereo jack pad. To use the module in the RS32 mode, you do have to add a replacement trace on the back side of the circuit board, using hookup wire, for the module to work in the RS232 mode.



Likewise, the +5 volt power trace from the contrast pot to the LCD backlight pad is shorted to the ground plane on the BOTTOM side of the circuit board. Cut the 2 traces just before they enter the ground plane and solder in a short piece of hookup wire to connect the two pads together....after you complete the assembly and the pads have been populated.

The pcb mods should be made before anything is soldered onto the board! Once parts are installed, TOP side pcb mods are either problematic or impossible to make. BOTTOM side mods are usually much easier to make so no big deal with those.... AGAIN, if you received your kit in the mail, I ***SHOULD*** have already made the cuts. Please check you board before you start building.

If you don't have access to a Dremel, then this would be a GOOD EXCUSE to buy one and pin the blame on QRPme and this kit that you already bought! But there is a second method of making mods on circuit boards that you can use...

Another way of making circuit board trace mods is to use a VERY sharp Xacto blade.... as in a NEW blade. Make a pair of cuts across the trace at either end of the area that you want to remove.

PROBLEMS:
Don't cut into the GOOD areas of the board.
Don't stab yourself or cut you finger off.

When making tiny cuts in delicate areas, don't try to cut thru the entire trace with one cut. Make MANY small light cuts that eventually make it all the way through the trace. Once both ends of the problem trace have been cut through, you can place the hot tip of your soldering iron on the trace and in a second or two, it will lift right off the board.

BE EXTRA CAREFULL, as flesh cuts way better than copper!
Wear whatever protective wear you have that you think might prevent an accidental slip of the blade from becoming a accidental slice of skin.....

So on to the assembly!

You will note that all the parts designators on the silk screen display actual component values instead of the typical R1, R2, R# etc. You should be able to assemble the module without much in the way of instructions.... so I will try to provide the necessary instruction here in my 1st draft builder's guide.

1. Make sure the 6 trace cuts are made. Ohm out the adjacent pad and a nearby ground pin to make sure that the pads are free of shorts.
2. Install the 3 DIP IC sockets at U1, U2 & U3
3. Install the 2 8-pin SIP connectors for the LCD. Insert BOTH connectors into the holes and make sure that they are in line before soldering them in.
4. Install the flat close to the board components such as the resistors, diodes and monolithic caps. Check resistor values with an ohmmeter if in doubt as it is easier to measure a resistor with an ohmmeter than to remove it from a board after soldering it in. Diodes have teeny tiny lettering which you can read using a magnifier. I use a nice lighter magnifier with built in white LEDs for maximum clarity. For the caps, I included .1" and .2" spacing caps for the appropriate sets of pads.

100K	[BRN-BLK-YEL]	
1M	[BRN-BLK-GRN]	(qty 2)
100	[BRN-BLK-BRN]	
300	[ORG-BLK-BRN]	
820K	[GRY-RED-YEL]	
1N4148	[1N4148]	(qty 2)

.1uf	[104]	w/.1" spaced leads (qty 2)
.1uf	[104]	w/.2" spaced leads
47pf	[470]	
470pf	[471]	
.047pf	[473]	
10pf	[100]	
22pf	[220]	

5. Now install some taller items. The MPF102 and 2N4403 transistors and 500 ohm pot spacing is a little tight so install them all in the pads to make sure that they fit together before soldering. Install the 78L05 regulator.
6. Bend and solder a piece of cut off resistor lead into a small loop solder. Solder it into pad A to form a test point for later calibration.
7. Solder in the electrolytic caps next. Double check the polarity when installing the caps.
   1.5uf
   1uf (qty 5)
   10uf
8. Install the small 4-50pf trimmer cap between the micro and the MAX 232 chip.
9. Cut the multi-pin Molex header into some 2 pin smaller headers. Install 1 at the pads marked SW and another at 5V. A small pin jumper is supplied to use as a mark-5 switch. If you want to use a 'real' power on/off switch, solder the switch leads into the SW pads instead of the 2 pin Molex header.
10. Install the 20K pot with the thumbwheel towards the outside edge of the board.

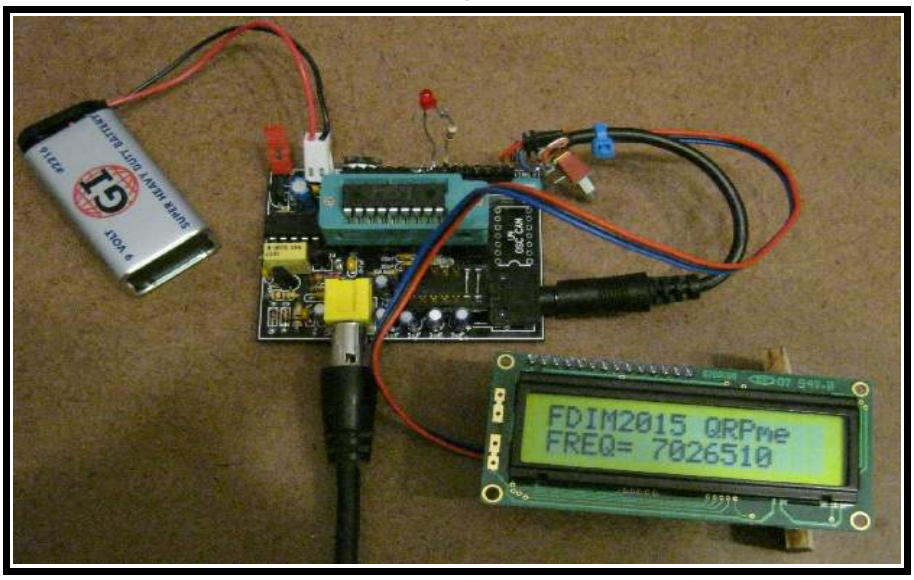
11. Solder in the 9 volt battery snap at the pads marked POWER. The + RED lead should be closest to the SW connector.
12. Double check that the trace mods have been made before soldering in the stereo jack at RS32.
13. Install the RCA input jack at SIGNAL.
14. You can now insert the chips into sockets:
    1. SN74HC132 into U2
    2. PIC16C622A into U1

You only need to install the MAX232 chip in U3 if you want to send an EIA-RS232 signal out to a true RS232 device. If you are going to use it primarily with a parallel LCD, then leaving the MAX232 chip out of the socket will increase battery life. Likewise, if you will be sending the frequency data to a serial backpack display, you won't need EIA voltage levels. Serial backpacks use TTL signal levels so the MAX232 chip is NOT required. Also, for TTL operation, pins 7 & 10 need to be shorted together. Pins 8 and 9 also need to be shorted together. These 2 shorts send the TTL level signals from the micro directly to the serial output jack.

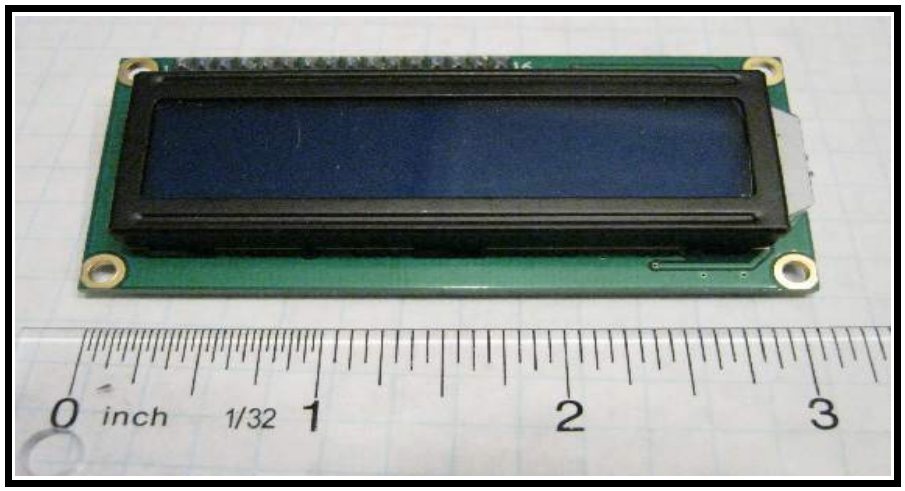
You should now be ready to power up the frequency counter and calibrate the counter. With the power applied, you need to adjust the signal at the calibration point A to 5 volts. Power it back off.

Now install a compatible output device.

I like the Sparkfun LCD modules. I use a white on black display (5 volt version) with built in white backlight.



I also use the serial backpack version (at times) which requires TTL level signals, so a jumper plug is installed at U3. The display requires +5v, ground and signal. The crazy wiring above picks up the +5v and ground from the last 2 pins of the LCD connector which normally powers the LCD backlight and sends it out to power the backpack electronics and display.



A COMPATIBLE LCD SCREEN IS ABOUT 3" LONG AND USES THE KS0066U OR EQUIVILENT HITACHI CONTROLLER. IT HAS A 16 PIN INLINE MALE HEADER FOR PLUGGING INTO THE PCB. I USED A GDM1602K LCD MODULE FROM SPARKFUN.

You can use other LCD displays with this module but you will have to make some special cables to connect it up to the board. I found some LCD modules for less the 5 bucks each. They work but since they are about 4 times bigger than the Sparkfun module and have a different input connector configuration, they are much more clumsy to use. Buy the Sparkfun LCD module and simply plug it into the board!

(I have no financial interest in Sparkfun. I just LIKE and buy like their products...)

Rex W1REX