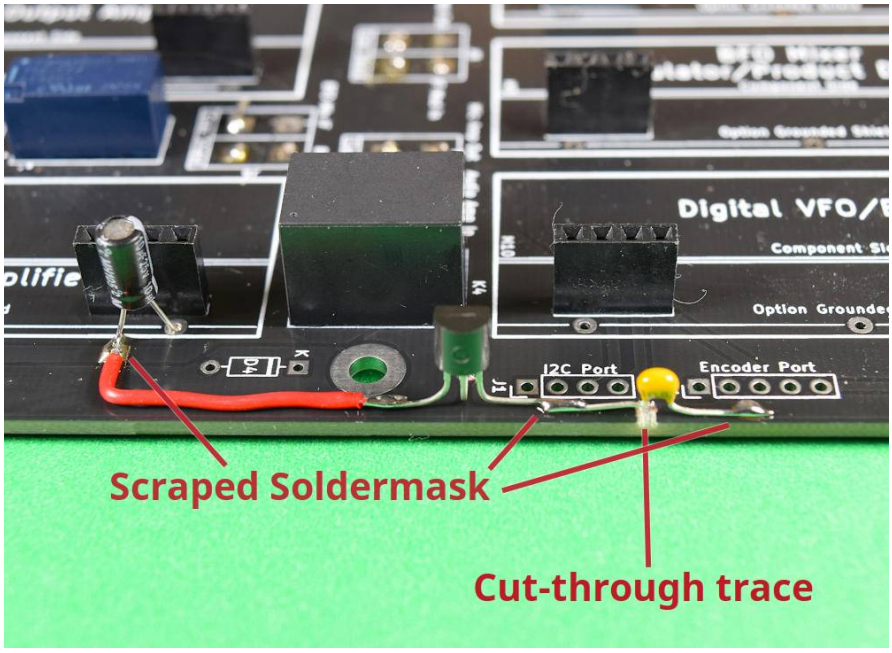
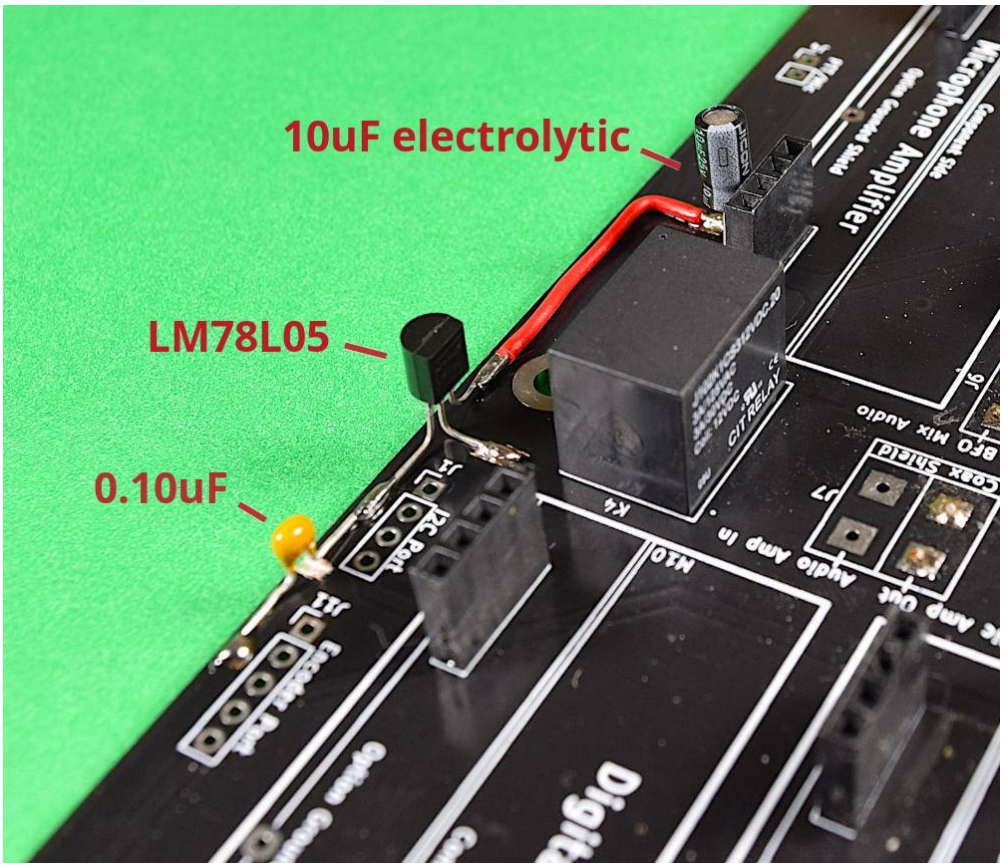
P3ST Kit

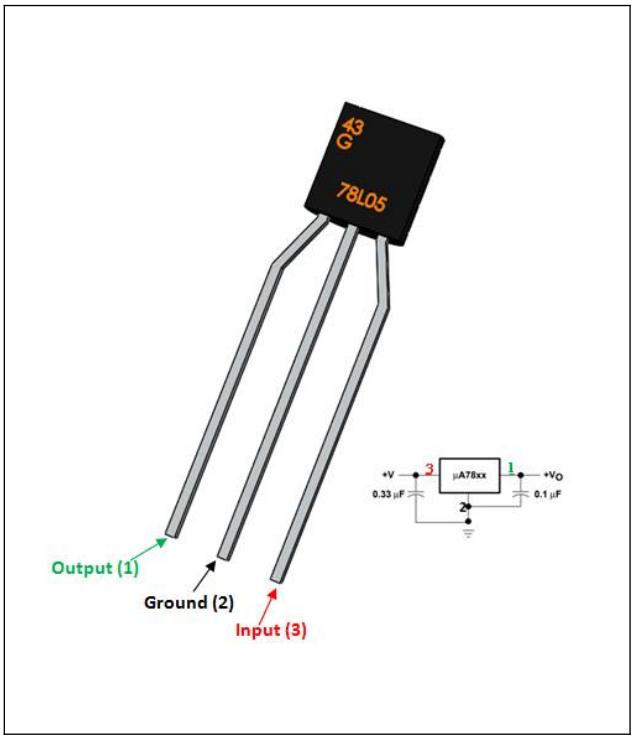
Field Service Bulletin #1: LCD Display Power

Problem:	Most 16x2 I2C-enabled LCD displays will not work to full brightness and contrast on the 3.3V supplied by the current P3ST motherboard. This bulletin will suggest a few ways this can be remedied.
Scope:	3.3VDC is provided to the I2C bus and the rotary encoder port available on the motherboard, and to the SPI and 2nd rotary encoder ports on the Digital VFO/BFO. Other display devices such as OLEDs and TFTs are available for 3.3V operation on whatever protocol they use (some I2C, some SPI). Nothing needs to be done with regard to voltage levels for those devices. It is only the common 16x2 backlighted LCD display (I2C enabled or not) that require 5V for satisfactory use.
Cautions:	The 5V supply suggested by this FSB should not be used for any device that inputs to the Xiao RP2040 MCU. For instance, it should not be used with a rotary encoder or for switch input. The pins of the 3.3V RP2040 chip are **NOT** 5V tolerant and damage will result. 5V supply should be used **ONLY** to power devices that require a separate voltage supply (such as an LCD display) and not for any device that might feed that voltage back to the MCU. Note: The PCF8574 I2C expander chip (the "backpack") used with the LCD display ***is*** 5V tolerant but will not drive the i2C bus with 5V itself.
Solution 1:	Add a 78L05 +5V regulator (TO-92 package) to the motherboard. This involves cutting an existing trace that feeds 3.3V to the I2C port nearest the Digital VFO/BFO, scraping away small areas of black solder mask, and soldering three components to the board.



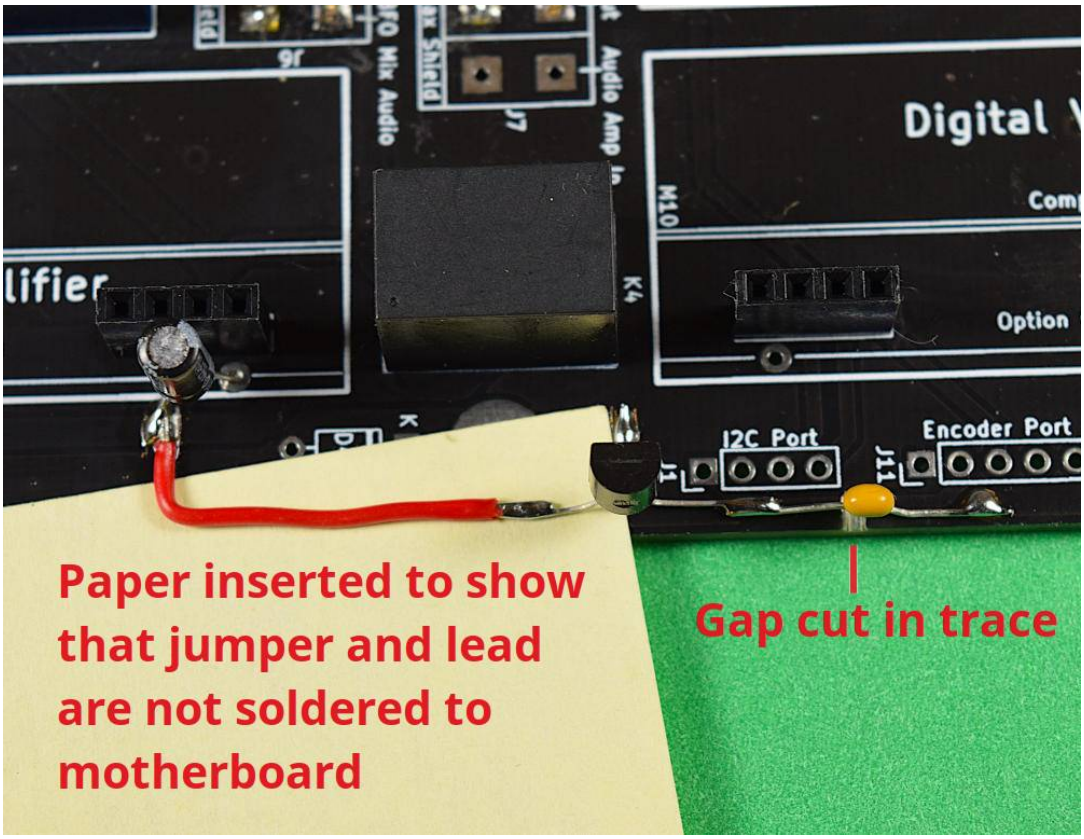


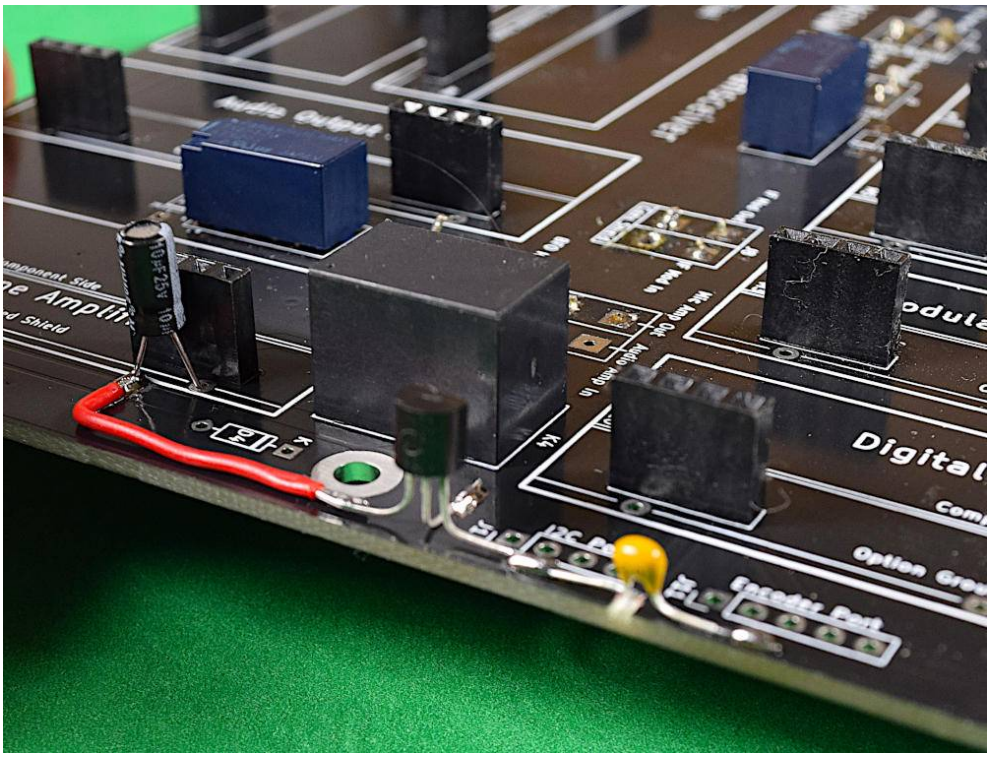
The 78L05 regulator and two bypass capacitors soldered in place



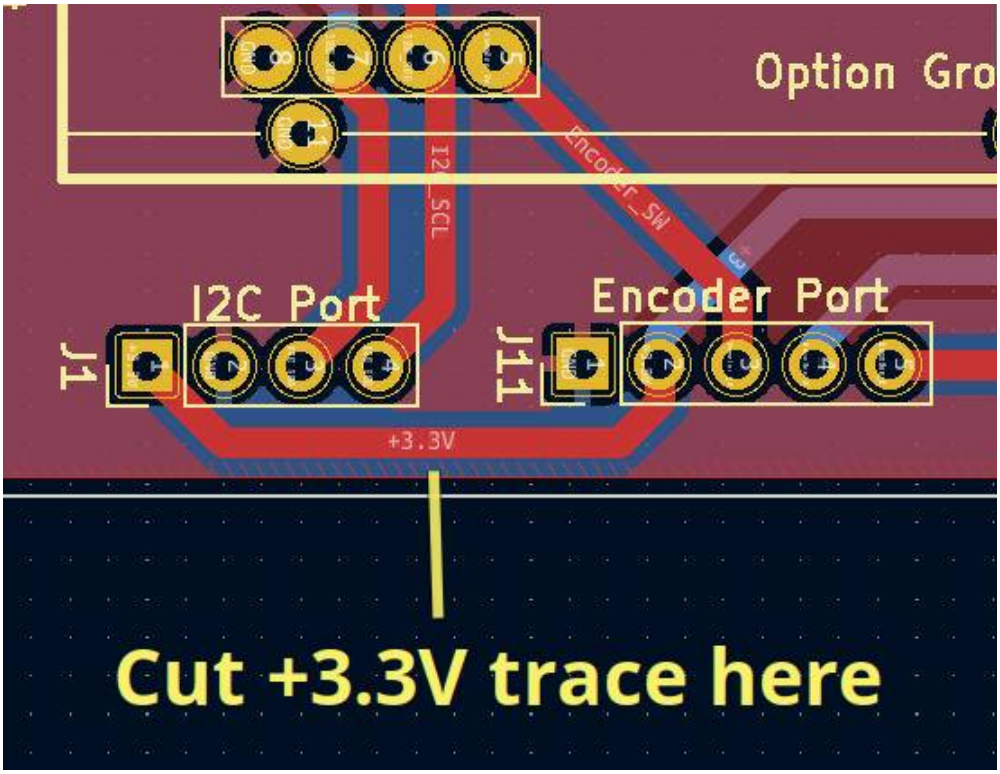
Pin 1 goes to the I2C-port side of the severed 3.3V trace, Pin 2 to the ground plane, and Pin 3 to the 12VDC trace.


Digital VFO/BF
Component Side
Option Grounded Shie
I2C Port
Encoder Port
Jumper soldered to +12V trace
LM78L05 GND soldered to motherboard ground-plane

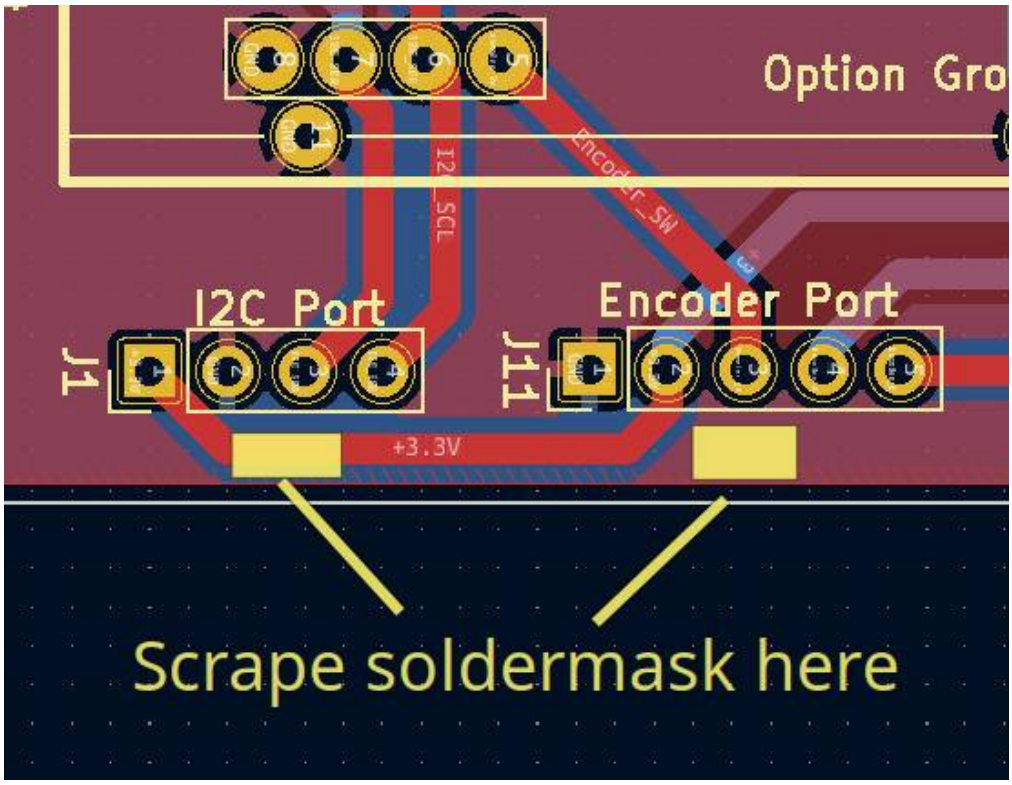

Digital
Option
I2C Port
Encoder Port
Paper inserted to show that jumper and lead are not soldered to motherboard
Gap cut in trace

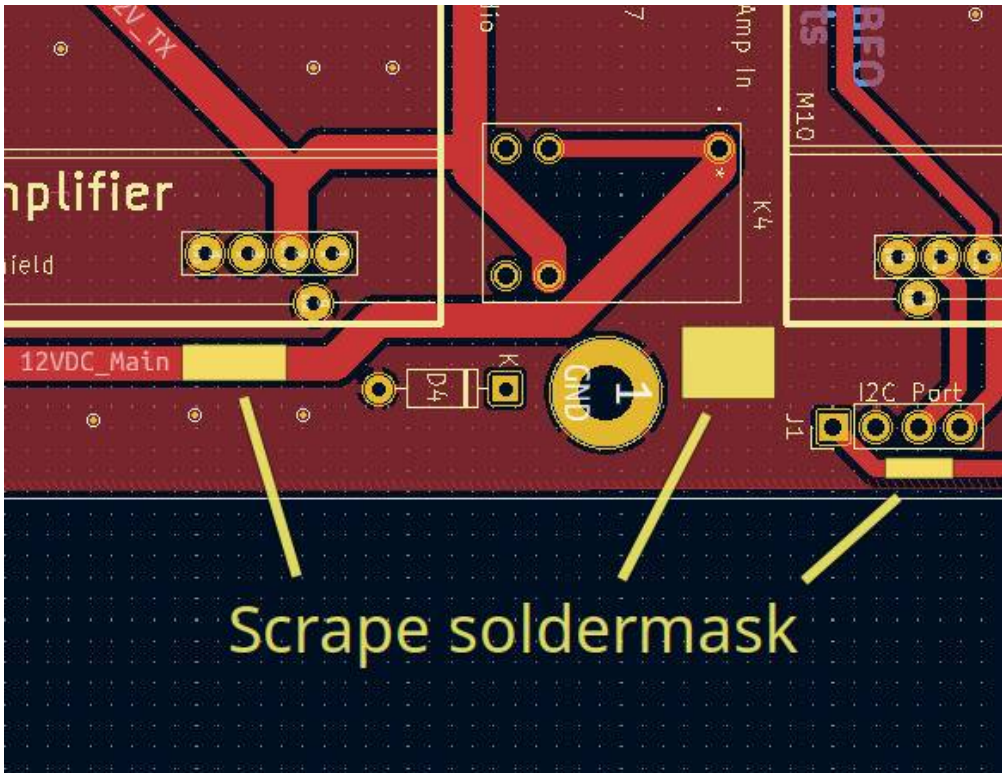


The modification doesn't have to be done this neatly (unless you're retentive too). This is ham radio, not brain surgery.



You can see the traces even with the black soldermask over them. Make sure to create a bit of a gap so no continuity is possible.





Scrape black soldermask to expose bare copper on the 12VDC trace, the solid ground plane, and the severed 3.3V trace. If you won't be using the nearby mounting hole, you can use its pad (grounded) in lieu of scraping soldermask next to it.

Install the 10uF capacitor from the 12VDC trace to the nearby pad for the

	optional shield (for the mic amp module), observing proper polarity, and install the 0.10uF cap from Pin 1 of the regulator to the groundplane patch you prepared earlier to the right of it.
Solution 2	Use an external source of 5VDC. This would also involve severing the existing 3.3V trace as for Solution 1, and either soldering +5V to exposed trace as shown above, or running a wire directly to the LCD display. No bypass capacitors are required, but not a bad idea if you have them on hand. The negative side of the external 5V supply must be grounded to the motherboard (where ever convenient). Any 5V supply will work. For instance, a small buck-type DC-DC converter may be installed to the motherboard traces indicated for Solution 1. Here are a few very-cheap (< $2) examples: