W0EB's Modifications to the BITeensio card to enable use with a 5" Color Touch Screen Display. (Best done on a brand new card at build time).

W0EB's uBITX modified for a 5" color Display with MOUSE control.

BITeensio card modifications necessary to utilize a 5” RA8875 based TFT Color Touch Screen Display that will allow touch screen or mouse control of the uBITX. (Keyboard control as well, but as of this writing, it can be keyboard OR mouse, not both at the same time.)

All front panel controls except on/off/volume can be operated with the mouse. The mouse can be either a wired USB mouse or a wireless one (the wireless receiver is plugged into the upper left corner of the uBITX in the photo. (The little white dot just under the “Memory Table” box is the mouse cursor.

First off, no modifications are needed to use the 2.8” ILI9341 based color touch screen display other than to make up a special cable which is covered in the main BITeensio construction manual. Keyboard and mouse control with the 2.8” display is also possible but not simultaneously.

We have successfully interfaced a 5” RA8875 based TFT color touch screen display sold by the “Buy Display” company and others. The Buy Display model is the ER-TFTM050-3 and should be ordered with the 4 wire SPI control interface, Resistive Touch Screen and the ER3304-1 font chip. The SD card interface is not required as we do not use the one on the display.

The major advantage of the 5” color display with the BITeensio is with the USB host mode available on the Teensy 3.6, it’s possible to use a USB mouse or unlighted keyboard for control of the uBITX (all but the on/off/volume control).

Unfortunately due to constraints of the Teensy’s USB host interface the KB and mouse cannot be used at the same time (as of the time this manual was written, though we are trying to overcome this problem).

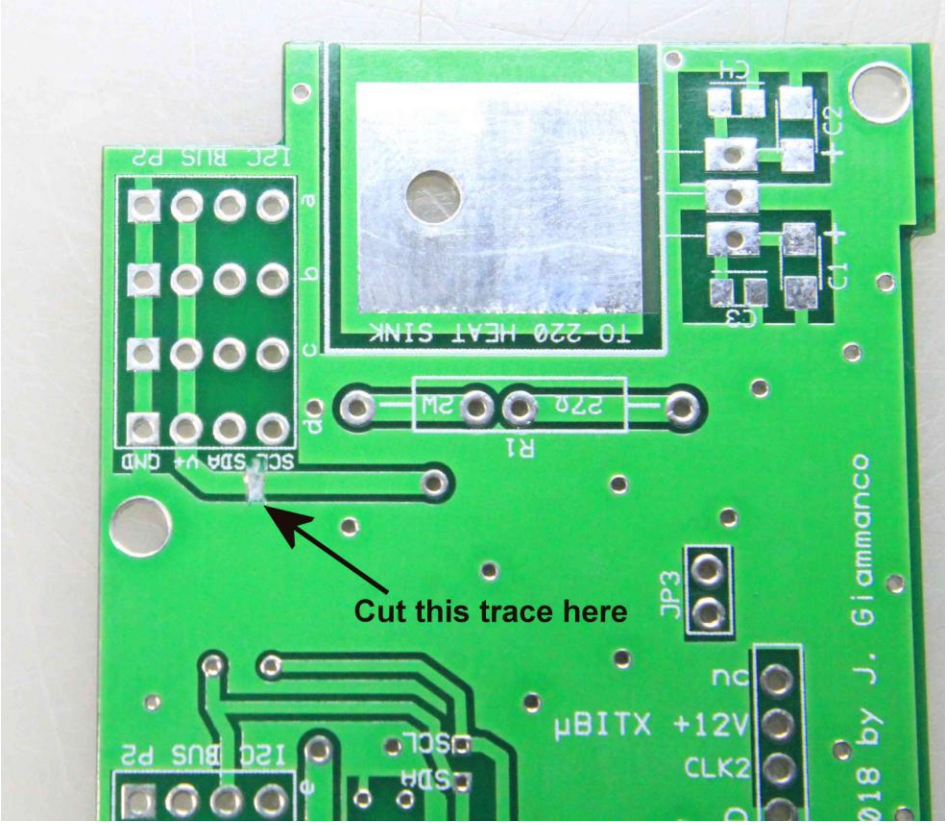
The major disadvantage of using the 5” display with the BITeensio card comes from the display requiring around 500 milliamps of current at 3.3 volts, mostly consumed by the LED backlight. In order to provide this much current, the card must be modified to have an on-board 1.5 amp LD1086V33 voltage regulator IC to provide the 3.3V to the display. The relationship between the Teensy’s SPI I/O signals to and from the display require that both the 3.3V power supply ground and the signal ground must basically be the same, low impedance ground and this

almost dictates that the regulator be mounted on the BITeensio itself to provide this. If not done this way, any ground loop at all will cause the display to be extremely distorted, blurry, and noisy or some other manifestation of poor readability and we do NOT want that. Fortunately, if we are going to use a Color display, most likely we won't need the 5 volt I2C header connections so the area used for the 4 headers of the 5 Volt I2C connections can be easily modified for use. An added advantage of this removes the requirement for the TXS0102 Level Shifter IC and then the only difficult item to solder on the board becomes the Si5351a.

Another advantage of using the color display (either 2.8" or 5") is the only front panel controls on the radio now are reduced to just the tuning encoder, its function switch and the on/off/volume control. In the case of the 5" display, all front panel controls except the on/off/volume control can be operated just by using a USB mouse (either wireless or plug-in).

Modification of the card is best done during the initial build and all the pictures associated with the mod in this manual will be geared toward that. A relatively experienced builder can also perform the mod on an already built BITeensio card with only a little more difficulty.

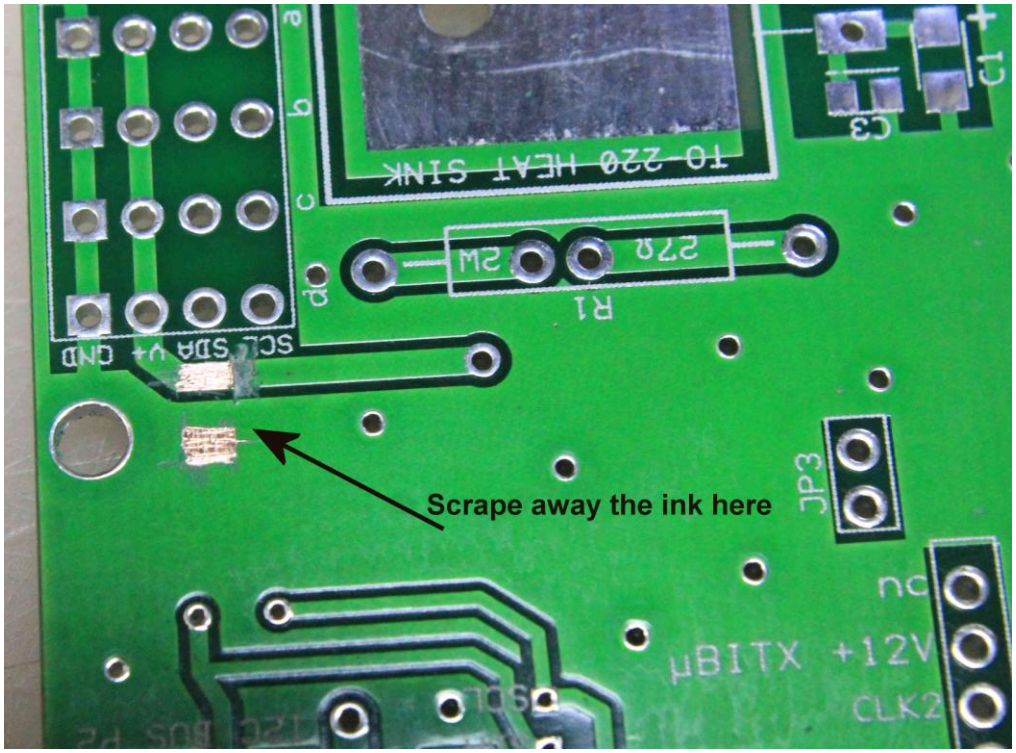
First we need to isolate the 5 volt I2C header area's +5 power connection so we can make it 3.3V. That's done by cutting the trace between the 7805's 5 volt output and the V+ line on the header bus.



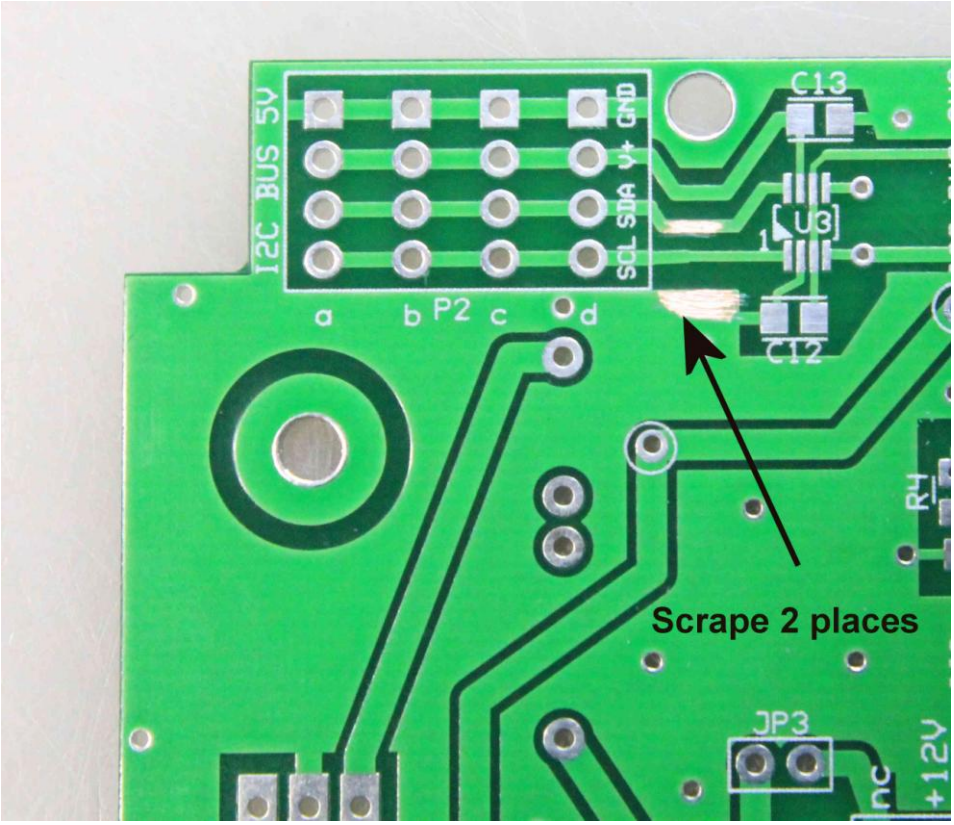
Cutting the indicated trace removes the original 5 volts from the I2C bus. Measure continuity with an ohm meter to make sure there is NO connection across the cut.

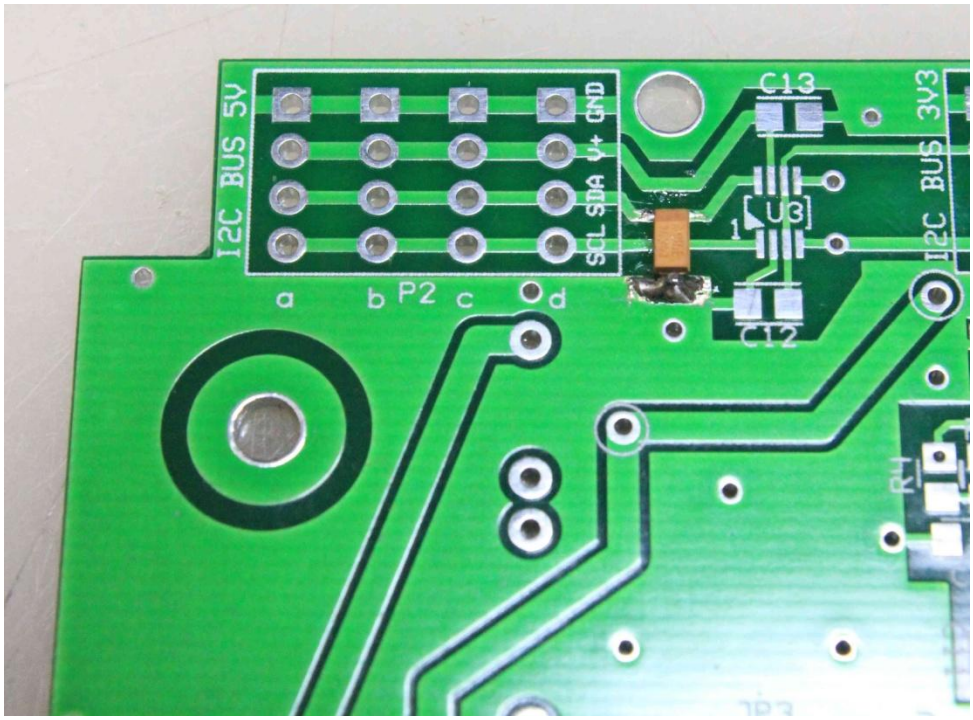
The LD1086V33 regulator requires 10uF stabilization capacitors on both the input and output so we now must create mounting pads for a size 1206, 10uF Tantalum electrolytic capacitor to be mounted to the soon to be 3.3V power line on the bus.

Carefully scrape away some of the silkscreen ink on the +V side of the cut trace and also scrape a ground pad directly below on the ground plane of the board to make soldering the capacitor possible.



On the other side of the card you will need to create a set of solder pads for the input side to the LD1086V33 regulator as well. No trace cutting is needed for that if the TXS0102 Level shifter is not mounted so if you are modifying an already built BITeensio, it is suggested that the level shifter be removed. That will make the modification much easier.

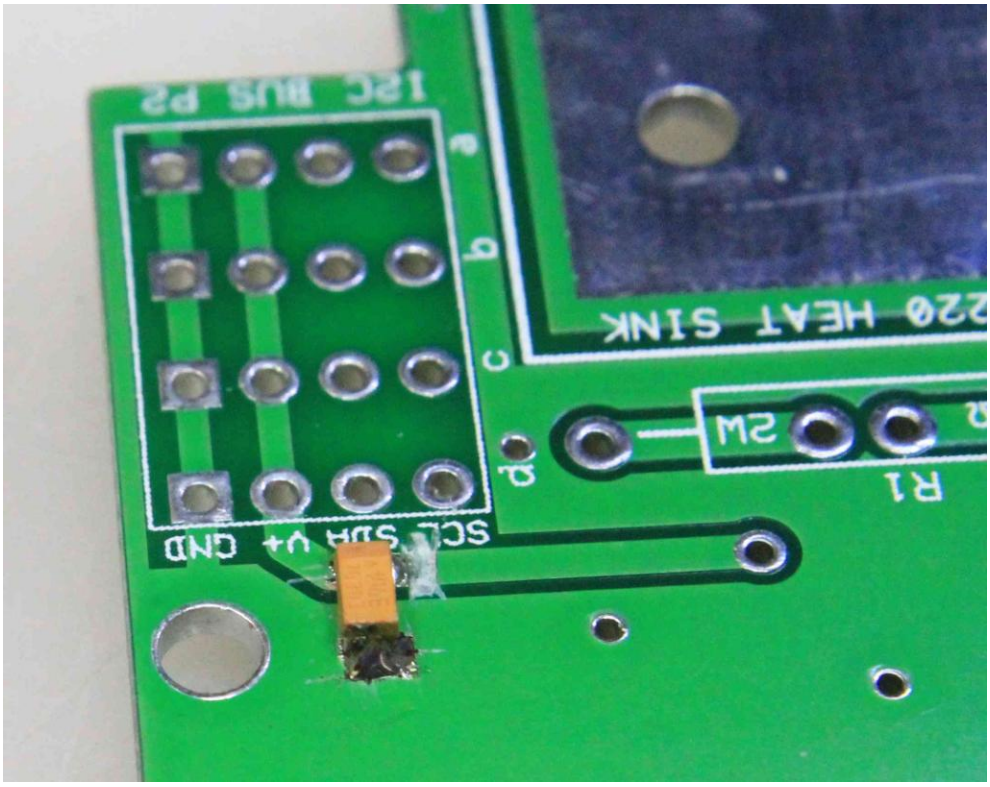




Now solder a 1206 size 10uF Tantalum capacitor to the pads you just scraped clean of silk screen ink. The positive side of this capacitor is indicated by a dark

line (unlike many other electrolytic capacitors which have the negative side indicated by a dark line). Make SURE you solder the + side of the capacitor to the scraped pad on the line marked SDA (this will become the input to the 3.3V regulator. Make sure you get a good solder connection on both ends of the cap or the regulator might break into oscillation and create all sorts of havoc.

Turn the board over and solder the other 10uF Tantalum across the pads you first scraped there. Again, the dark line on the capacitor indicates the + side and must be soldered t0 the +V trace. The other side of the cap gets soldered to the ground plane where you scraped a pad for it.

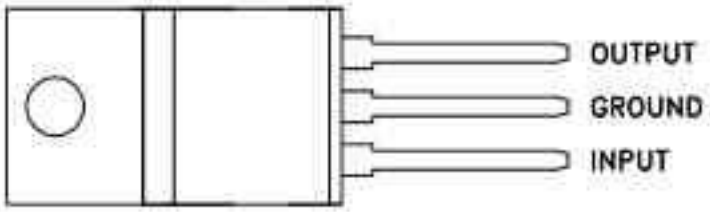


This completes the most difficult part of the modification. Next, you need to decide which side of the board you want the LD1086V33 regulator to stand out from. There will have to be a heat sink on this IC and it will depend on whether or not you have enough room between the front of the BITeensio card and the front panel area (where the display will be mounted) if you intend to mount it on the front side of the card. The back side gives more room to work with but puts the heat sink in close proximity to the 7805 5Volt regulator's heat sink. The tab of the

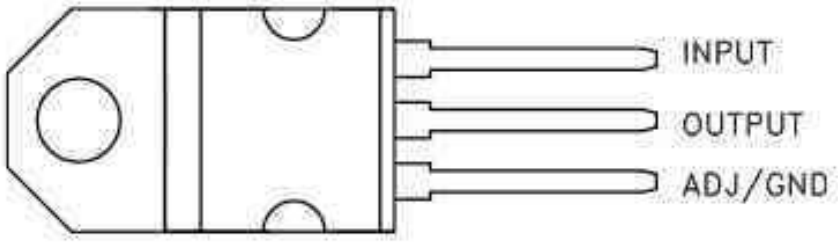
LD1086V33 is hot to 3.3V so you MUST insulate the heat sink from the regulator with a thermal pad and an insulating bushing for the screw to keep the heat sink from causing a short if it comes in contact with ground or something else. The 7805's heat sink is grounded and contact between the two heat sinks IS possible. Insulate the LD1086 and avoid possible trouble.

The LD1086V33 looks like the standard 3 terminal (7805) regulator, but that's where the similarity ends.

Below is the pinout for a standard 3 terminal regulator like the 7805 for comparison only. Looking at the front of the 7805 you can see the lead pattern.

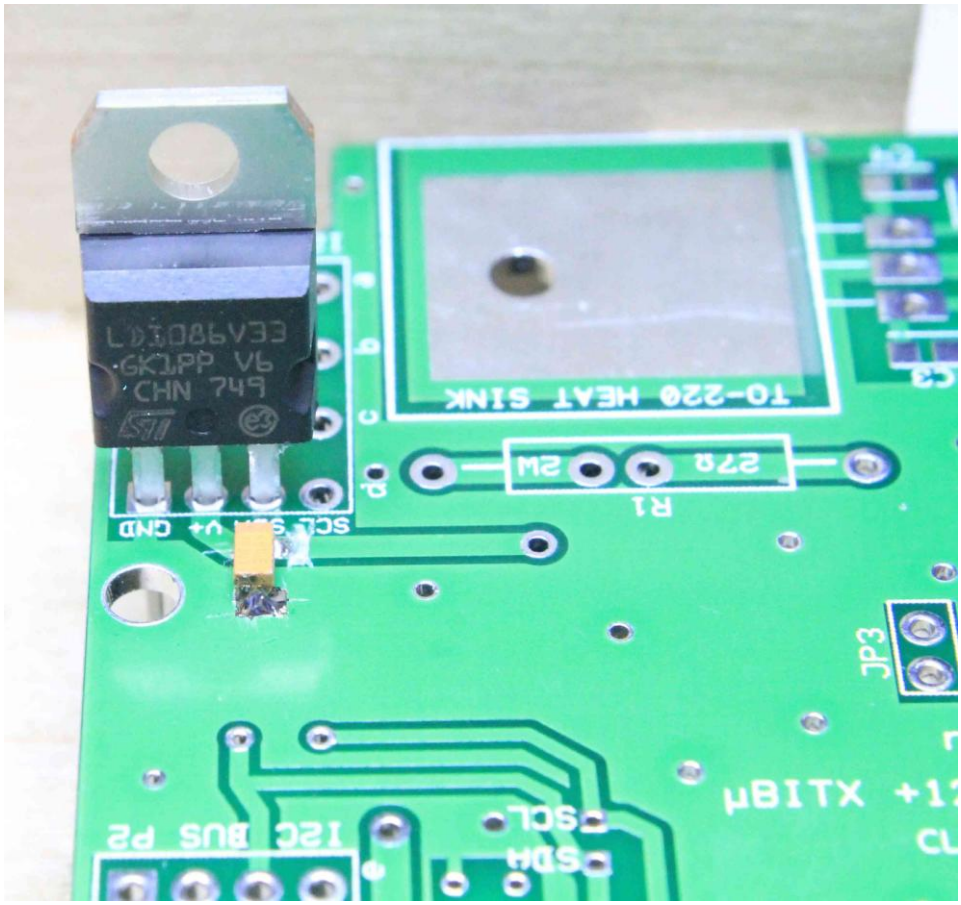


Now the one we are using for the 1.5A 3.3V regulator, the LD1086V33 has a completely different pinout which just happens to be to our advantage for this modification.



Note that the ground is on the bottom lead and the output on the center lead. The output of the LD1086V33 is also connected to the tab and is the reason the heat sink MUST be insulated to prevent shorting the 3.3V output to ground or something else on the BITeensio board.

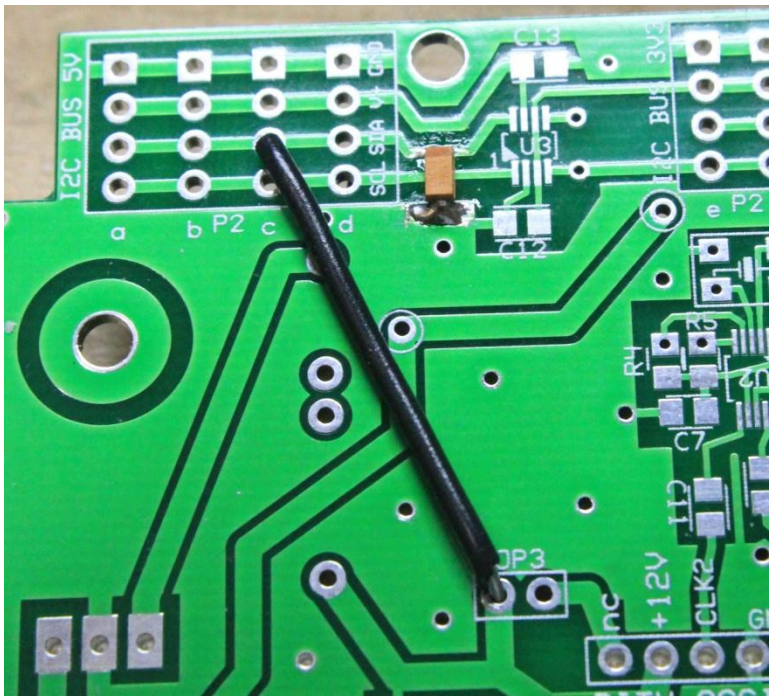
Looking at the layout of our modified 5V I2C header area, reading from the top down we have GROUND, +V and SDA (we are co-opting the SDA trace as input to the regulator. Note that if we put the LD1086V33 on the back side of the BITeensio, the ADJ/GND (in this case GND) goes into the top hole, the output goes in the middle (+V) hole and the input goes in the SDA hole. Works perfectly for our modification this way as you can see in the picture.



If you decide to install the regulator on the other side of the board, just remember to turn it 180 degrees so the pins still fit in their appropriate holes on the board.

Whichever way you decide to mount the regulator, now is the time to solder it in place.

We have to provide input to the regulator and that takes a short wire jumper from the 12V input of the card to the former SDA trace on the old 5V I2C header pads.

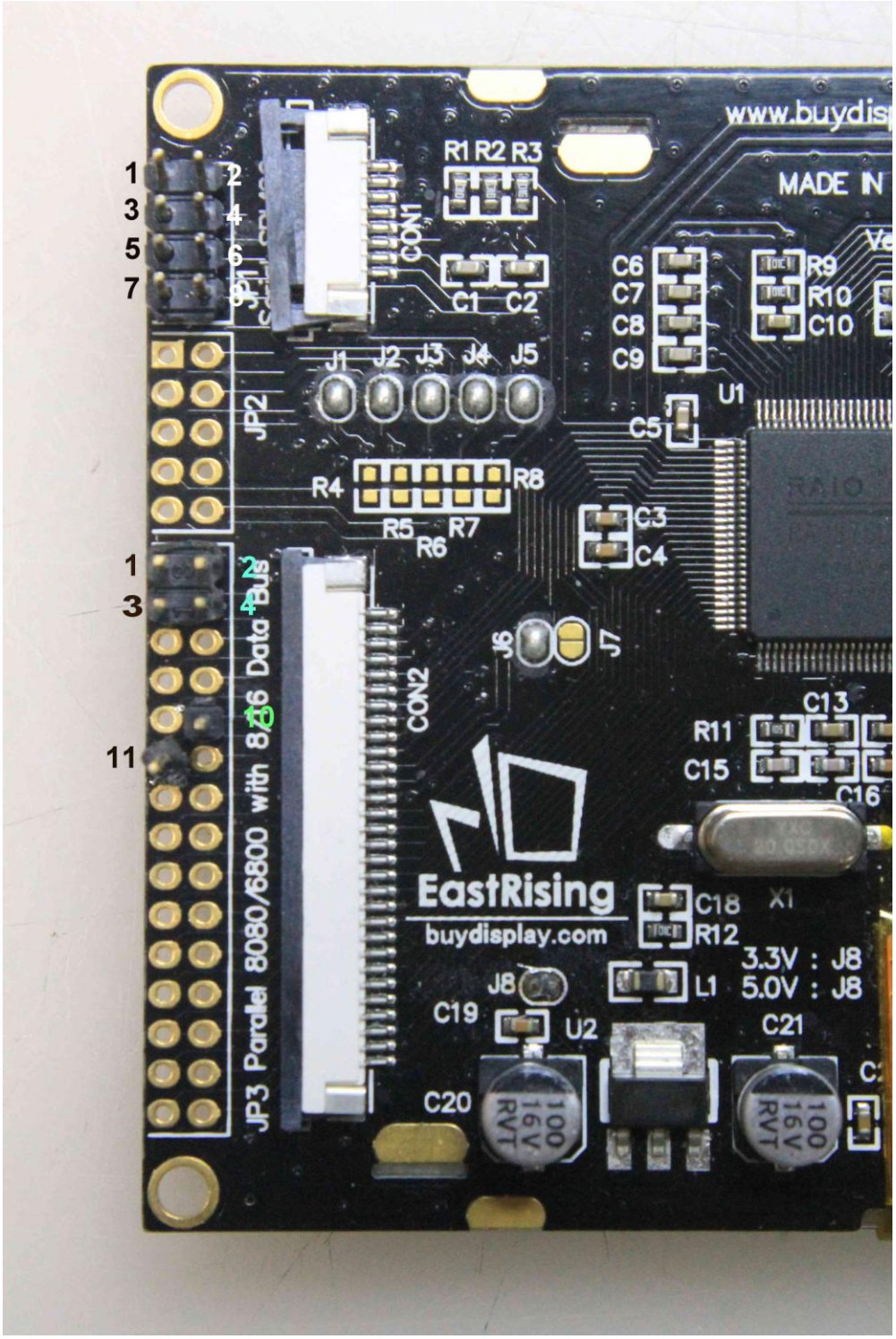


In this picture, shown modifying a new board at build time, you can see that the TXS0102 Level Shifter, U3 is not installed. If you are modifying an already built board, you must either removed U3 or cut the SDA trace between pin 8 of the TXS0103 and the + side of the 10uF tantalum capacitor to prevent destruction of the level shifter by conflicting power supplies. Removal of the chip is preferable since you won't need the 5V I2C bus for driving the display anyway.

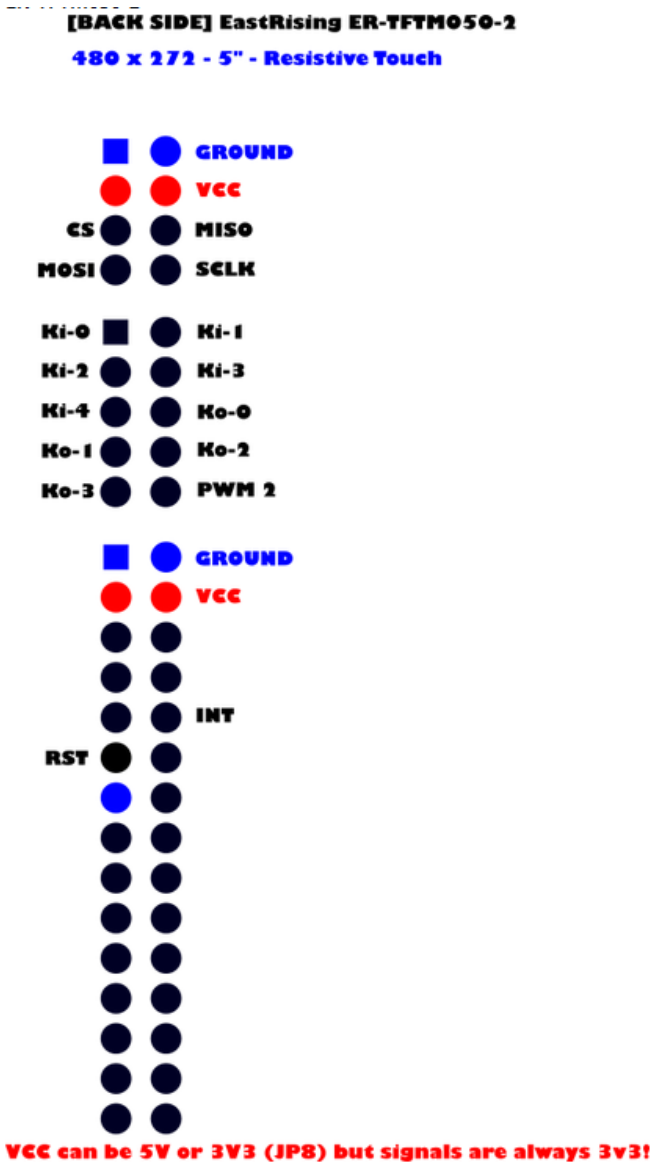
We DO leave the two 3.3V I2C headers and the 3.3V I2C bus intact and it can be used for peripherals like ADC boards for S meters, Power output/SWR meters and the like that are capable of running on a 3V3 I2C bus.

If you are doing the mod on a new board, at build time, you might still want to install the two .1uf SMD capacitors at C12 and C13 as these provide additional RFI bypassing for the 3.3V line on the Teensy itself and also the 3.3V output line of the new power supply.

Connections to the 5” Color Display:



There are 3 header areas on the back side of the display labeled JP1, JP2 and JP3. JP1 is the main connector, JP2 is not used and we use only Pin 10 of JP3.



Pin connections are as follows:

Display JP1

Pin 1 and 2: +3.3V @ .5 amp (Above diagram says VCC can be either 5V or 3V3 but this is not the case. Use ONLY 3.3Volts to the display power and all signals must also be 3.3 volts or the display will be damaged.

Pin 3 and 4: Display power Ground (also signal ground)

Pin 5: CS (display Chip Select) Connects to BITeensio header P(TFT) pin D14

Pin 6: MISO (SDI output) connects to BITeensio header P5 pin D12

Pin 7: MOSI (SDI input) connects to BITeensio header P5 pin D11

Pin 8: SCLK (SDI clock) connects to BITeensio header P(TFT) pin D13

Display JP2 No connection all pins.

Display JP3

Pin 1 and 2: Display Power (paralleled with JP1 pins 1 and 2, not used here)

Pin 3 and 4: Display Ground (paralleled with JP1 pins 1 and 2, not used here)

Pins 5 through 9: not used

Pin 10: INT (touch screen interrupt) connects to BITeensio header P(TFT) pin D17

When making your cable:

Twist together power & ground leads – connect to the new 3.3V 1.5A regulator output & ground on the BITeensio card.

Twist together MISO and MOSI (twist in a ground wire as well – connect this ground to BITeensio header P5 Ground) All three (MISO, MOSI and this ground) connect to P5.

Twist together the other 3 wires, SCLK, CS and INT and connect these to P(TFT) as indicated above. The cable MUST be shielded with the shield connected to ground on both ends. The ground end going to the BITeensio should be connected to P5 pin 1 (Ground) for convenience as well as providing additional grounding for the display. This really helps keep RFI to a minimum and noise out of the display.

A small PC board has been designed by the author to make connecting the BITeensio's headers to the display's headers easy and one of these boards will be included with each BITeensio kit if the purchaser indicates their intention to use the 5" color display instead of a 2.8" color display or a 4 line X 20 character I2C LCD display.

This manual is intended to be additional to the main BITeensio construction manual and is needed only if the user is planning to use a 5” Color TFT touch screen display instead of I2C displays originally called out in the main manual.

Once this modification has been performed it will be difficult to return the BITeensio card to its original configuration.

Jim Sheldon, W0EB

Design & testing, N5IB/W0EB
PC Design and layout, N5IB,
Manual & photography, W0EB