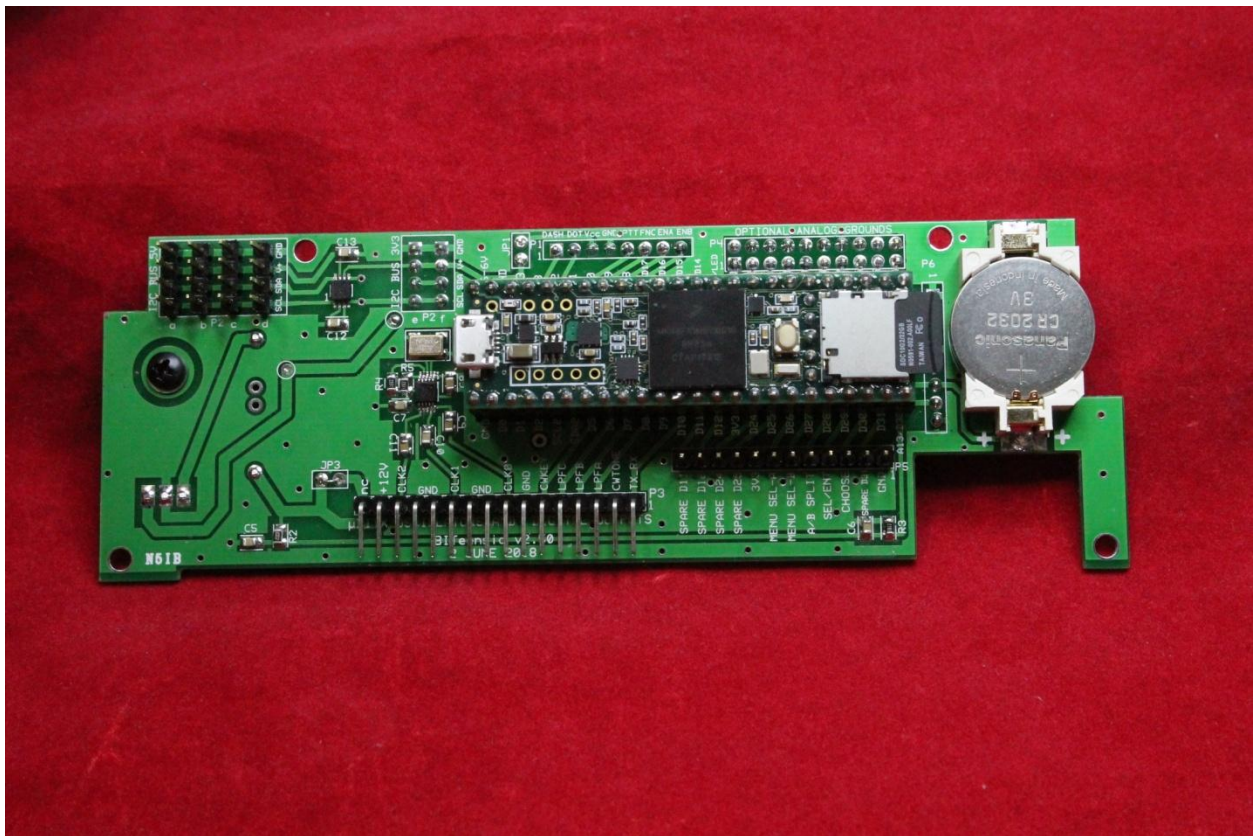


Construction Manual for the W0EB/N5IB “BITeensio” Card Using the Teensy 3.6 MPU in place of the NANO V1.6 - 11/12/2018 (BITeensio is Pronounced “Bit-EEN-sio”)

Copyright © W0EB, W2CTX and N5IB July 25, 2018, all rights reserved. This document may be freely copied and distributed as long as the contents are not changed and the authors are given full credits.



This replacement for the Raduino was designed and produced after having a successful run of our RadI2Cino card which used the original Arduino NANO MPU and later via an adapter card we called the TeensyDapter that allowed one to plug a Teensy 3.5 or Teensy 3.6 into the adapter and then plug the adapter into the RadI2Cino in place of the NANO. Since it gave us so much more computing power

and additional I/O (both digital and analog) it was decided to completely re-design the RadI2Cino into a card that only used the Teensy 3.5 or 3.6 as the onboard MPU (Micro-Processor Unit). An added benefit of the Teensy is that it has an on board micro SD card of up to 16 gigabytes – (32GB if you can format it to use the FAT32 rather than NTFS format) which can be used like an attached “Solid State Hard Drive” for storage of variables and backup of critical onboard parameters such as alignment information.

Overview:

This kit uses small surface mount components and you must use caution when installing these so you don't short adjacent pins of the very tiny devices. Also, when installing resistors and capacitors, be careful to position them properly and don't use too much solder. A very helpful item is Chip Quik™ “Tack” flux, especially for soldering the Si5351a and the TXS0102 IC's.

If you are not completely familiar with the techniques for soldering the tiny, fine pitch IC pins to extremely close-spaced pads on a PC board, we suggest you proceed very cautiously or maybe even enlist an experienced friend to at least solder the 10 pin clock chip the crystal and the 8 pin level translator in place.

Very small diameter 63/37 low temperature, flux core solder is recommended for installing all components. For the header pins, slightly larger 0.8mm 63/37 flux core solder can be used if desired.

Unless you have really good eyes, you will also need a good magnifying glass or better yet one of those headbands with several different powers of magnifying lenses. A fairly bright light to illuminate your work is also a necessity. For most people, it's very difficult to accurately solder the Si5351 and the TXS0102 level translator IC's in place without these.

Refer to the BOM (Bill of Materials) for component values and ordering information if you are building from the bare board or ordering replacements for accidentally lost parts during construction (it happens).

All the solder pads on the board have been “pre-tinned” by the PC fabricating house and have a very thin layer of solder already applied to them. If you use a flux similar to Chip Quik™ “Tack Flux”, there is usually enough solder already on the pads that the pins of these tiny chips will bond with only the application of heat from a very fine tipped soldering iron though with the resistors/capacitors it’s a good idea to apply a small amount of solder to one pad, and while holding the component in place with tweezers, heat that end again until the solder melts and anchors the capacitor or resistor in place. Soldering the other end completes the job.

Constructing your BiTeensio:

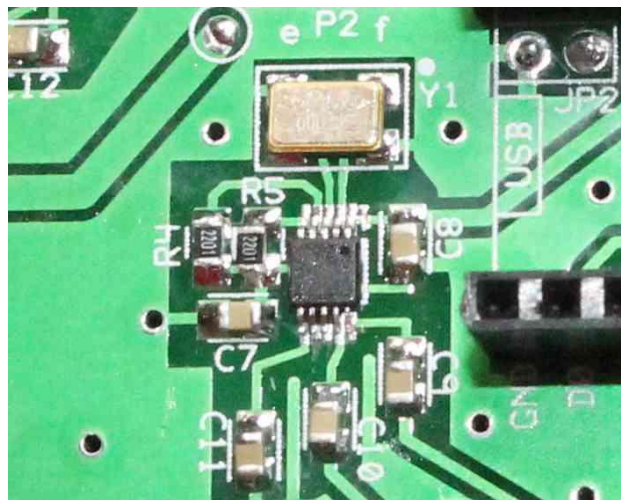
Let’s get the really hard stuff out of the way first. There are two integrated circuits, U2, the Silicon Labs Si5351a clock generator chip which creates the various RF oscillator signals used in the uBITX and U3, the TXS0102 Logic Level Translator chip which serves to protect the Si5351a from more than 3.3VDC logic levels placed on the I2C bus by any other I2C devices such as a display controller designed for 5 volt operation and finally there is a 25.000 MHz surface mount crystal which provides the master frequency from which all the other RF signals are derived. These are the most difficult parts to install correctly so we tackle them first.

Apply a thin layer of flux across all the PC board pads for U2, the Si5351a clock IC. Carefully observe the pin 1 indicator dot on the 5351 chip and place that pin over the indicated pin 1 pad on the board. (The Pin 1 designation may be hard to see but it is the upper right pin of the pad pattern when viewed from the front of the board with the upper edge horizontal.) Using a tweezers or a pointed wooden toothpick, carefully position U2 so each pin on both sides is in the center of its pad on the board.

Using a very fine point soldering iron, “Tack” pin 1 in place very carefully. Next “Tack” pin 5 and check to make sure all pins are still square and over their pads. If necessary apply a little heat to pin 1 or pin 5 while applying a very GENTLE twisting motion to the IC with a pair of tweezers until the pins are all properly centered on their pads. Now carefully “Tack” pins 10 and 6. Inspect for shorts

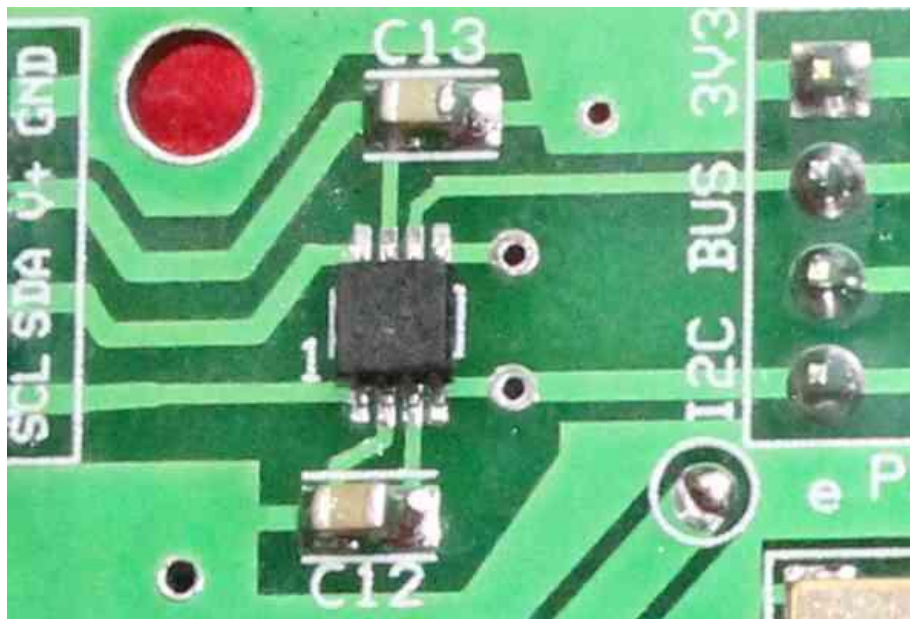
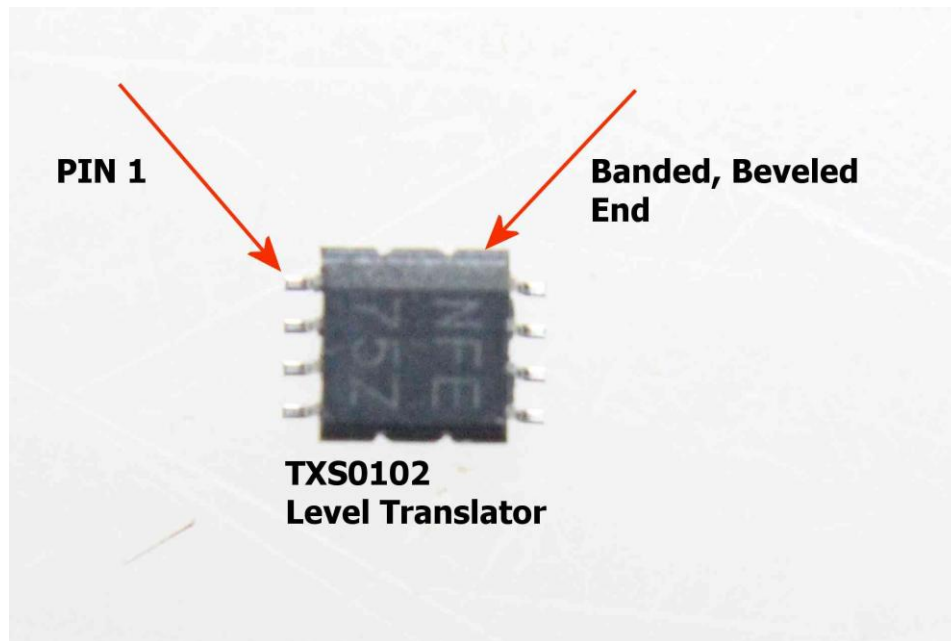
and if there are none, carefully tack the rest of the pins until they are all adhered to their respective pads. Before proceeding to the next step – using your DMM’s continuity check and referring to the trace and pad pattern near the end of this manual, perform a continuity check between the actual pins on U2 (not the trace or pad) and the capacitor or resistor pad they connect to. Use minimal pressure on U2’s pins so you don’t push it down to make contact in case the solder didn’t take. Clear any shorts with a little flux and solder braid. Once you are satisfied all pins are firmly attached to their pads and there are no shorts between them proceed to the next step.

It is NOT a good idea to use an older “analog” meter’s “Ohms” ranges to make these checks as some of them had fairly high voltage (22.5V or sometimes higher) batteries in them to power the resistance checks and if accidentally applied between the wrong point(s) and ground on the Si5351a or TXS0102 it could be enough to actually destroy the chips. Especially the 5351 which can tolerate no more than about 3.5 volts. A DMM usually has enough limiting resistance that its 9V battery doesn’t produce enough current through the probes to create a problem.



Using exactly the same procedure you used to install U2, install and check the TXS0102 Logic Level Translator IC, U3. The picture shows the location of PIN 1 on the device. On the PC board’s pattern for U2, pin 1 is located in the lower left

corner of the pad pattern with the horizontal upper edge of the card viewed from the front and the pads are immediately under the holes for 3.3V I2C header P2.



Just like with the Si5351, test it for shorts or opens before proceeding.

Install Y1, the oscillator crystal for U2. Apply a very small amount of solder to the 4 mounting pads for Y1, the 25.000 MHz SMD crystal. Look at the underside of the crystal and you will see 4 Gold plated pads. Apply a drop of flux (preferably

Chip Quik “Tack” flux) to these 4 pads. Carefully center the crystal between all 4 pads with the longest dimension oriented along the horizontal axis of the card. The crystal is designed so that it cannot be installed wrong as the upper right and lower left contacts (viewed with the long dimension oriented horizontally over the solder pads) connect to the internal crystal and the other two connect to the case for shielding.

Once you have the crystal positioned properly, you should actually see some of the solder for each of the 4 pads protruding beyond the edges of the crystal. While gently holding the crystal in place with a wooden toothpick or better yet, a sharpened bamboo skewer to keep it from moving, gently apply your soldering iron tip to the portion of one PC pad where solder is visible beyond the outline of the crystal until the solder flows and attaches to the unseen crystal pad. A good solder joint will be indicated if the crystal can't be moved after it's cooled. Once you have the one pad soldered, use the same technique and solder the remaining 3 pads. Only 2 pads are used to connect the crystal to the 5351, but the other two are connected to the shield can and grounded so you need to solder all 4.



When you have the Si5351, TSX0102 and the crystal in place, the really difficult part of the build is done. A short break to catch your breath might be in order here.

Continuing, solder the 2.2K 0805 SMD Resistors R4 and R5 near the bottom of the Si5351 chip.

Solder a 10K 0805 SMD Resistor at R2 and a 2.7K 0805 SMD resistor at R3.

Next, carefully solder the .1uF 50 volt, 0805 ceramic chip capacitors in place at locations C5, C6, C7, C8, C9, C10, C11, C12, and C13 on the front side of the board.

Now, install 10 uF 25V 1206 ceramic chip capacitor C15. You will have to scrape back the green solder mask on the ground side of C15 to allow it to fit properly. This is a (non critical) change from the original circuit and installing a 10 uF cap here helps cut down on the tuning clicks generated in the Si5351 clock chip.

The rest of the components will be installed on the rear of the board (with the exception of resistor R1 and, of course, the Teensy 3.6 MPU, which will be installed later) so flip it over – you might want to set it on a padded anti-static surface for ease of installing the final SMD's.

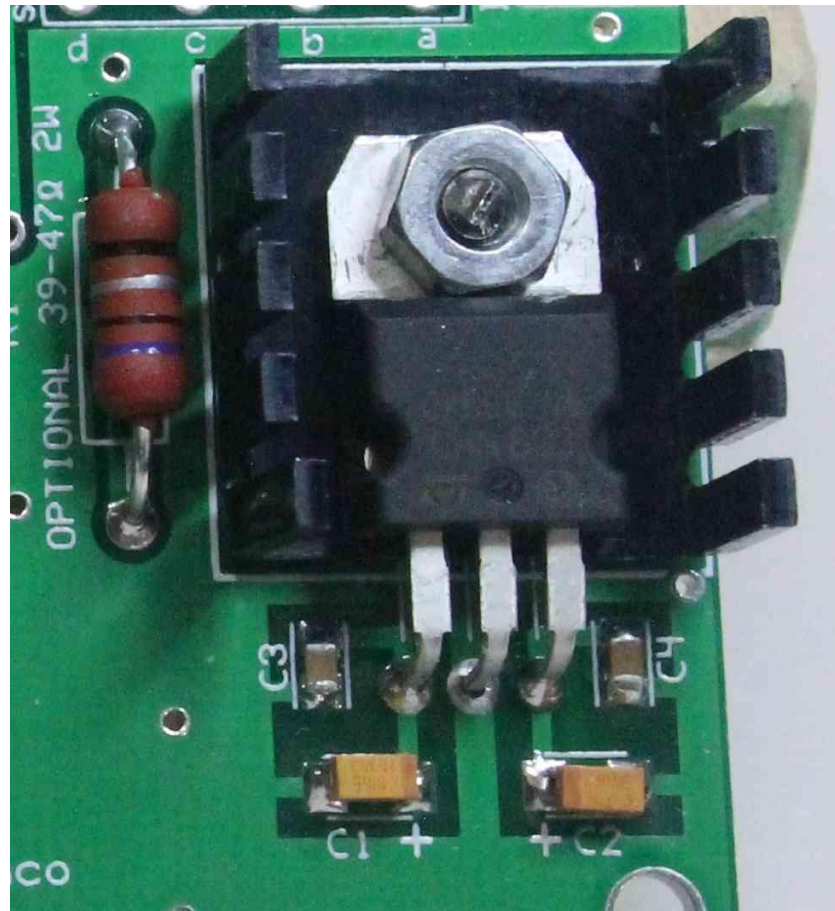
On the back side, install .1uF ceramic chip capacitors, size 0805 at C3 and C4 and C14.

Install 10uF, size 1206 or 1210 (either may be supplied) Tantalum capacitors at C1 and C2, carefully observing the polarity marked on the board. **The colored line on the capacitors is the + terminal and must be soldered to the pad marked + on the board. Size 1210 components are slightly larger than 1206 so if you have the larger ones, be careful that their terminals don't accidentally short to the ground plane when soldering.**

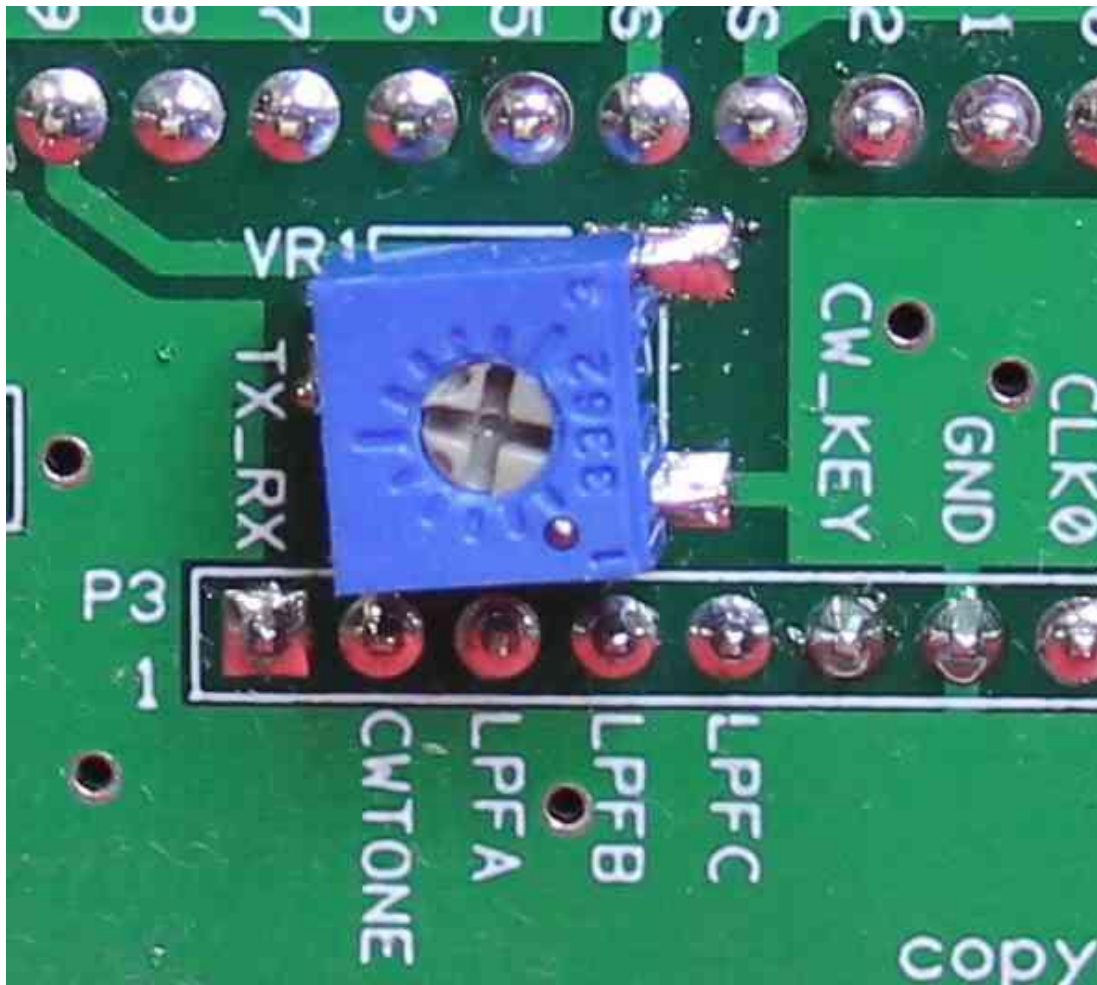
Make sure the terminals on both ends of these capacitors are properly soldered to their respective pads or U1 could break into oscillation with the possibility of a catastrophic failure that might destroy your Teensy and other critical parts on the board. Better safe than sorry here.

Install the 7805 Regulator IC and its heatsink at U1. Carefully bend the leads of U1 so they clear the edge of the heat sink and the hole in the regulator's tab is centered over the hole in the heat sink. You can, if you like, use thermal compound (not supplied) between the 7805 and the heatsink but this is probably not necessary. The tab of the 7805 is grounded so use a short 4-40 screw, #4 lock washer and nut to secure the regulator and heatsink to the board using the available hole.

NOTE: The heatsink for U3 can have taller fins than the one called out, but the base of it can't be bigger than a .75" x .75" footprint or it will overlap the thermal barrier etched into the ground plane of the PC board.



Finally, install the 5K trimmer potentiometer VR1. Match its pin pattern to that on the PC board, just above and near the left end of P3. It's virtually impossible to install it wrong if you center the pins over the pads as they will fit only one way. CAREFULLY solder these pins to the pads without touching the soldering iron tip to the plastic case as it's easily melted. VR1 is used to set the CW side-tone volume to a level suitable to the user.



That completes the SMD component and voltage regulator installation. Final construction consists of installing the connection headers.

Install the multi-block I2C header P2, which consists of six, straight 4 pin headers. These are intended to be mounted on the front or back side of the board and how much room you have between the card and your chosen case's front panel will dictate where you put them. However, if space allows, we do recommend putting the two 3.3V headers (P2-e and P2-f) on the side of the board OPPOSITE from the side on which you decided to install the 5V headers P2-a through P2-d. That will help keep you from accidentally plugging a 3.3V I2C device into one of the 5V headers which will likely destroy the 3.3V device.

Starting with P2-f, insert a 4 pin DuPont header. Solder one pin first and then carefully heat that pin again while positioning the header separation strip flush

and flat to the board for proper orientation of the connection pins. Don't keep heat on it too long though as you might lift a PC pad or trace. Once you have the positioning to your liking solder the other 3 pins. Be careful not to use too much solder and don't short any of the adjacent pins together. Do the same with P2-e and then continue on toward the left side of the board installing P2-d, c, b and finally P2-a. Check that all connections are well soldered and there are no shorts between pins.

On the back side of the board, install P1 using the same technique, but solder the pins to their pads on the opposite side of the board. Make sure the separation strip is flush and flat to the board. *The big flat tab on P1 needs to be positioned on the side away from the board's top edge exactly like on the original Raduino. If you install it upside down the pins will be numbered wrong and won't match the "Digital Plug" from the uBITX. This will cause things like the tuning encoder not to work and could possibly damage the encoder and other front panel controls.*

Next, solder the 16 pin, right angle header at P3.

Depending on your personal preference, you may install P3 on either the front or back of the board. Installed on the front, the board will easily clear the back side of the audio plug on the uBITX audio connector. If P3 is installed on the back side, the wires from the audio plug will need to be routed around the left end of the BITeansio but they will fit. You may have slight clearance issues with the 2 front mounting screws for the uBITX main board but a little judicious positioning of the ends and it will fit just fine.

Use the technique of soldering 1 pin near the middle of the header and then while heating this pin, quickly position the separation strip flat and properly positioned with relation to the board. Solder the rest of the pins and check for shorts.

Install wire jumpers at JP1 and JP2 if desired. These jumpers give you the options of whether or not to connect the 3v3 output of the Teensy to pin 3 (green wire) of digital plug P1 (Pin 3 of P1 connects to 5V on the original Raduino and our earlier Radl2Cino card) and the option of connecting your Analog and Digital grounds in

common (JP2 installed) or keeping them separate (JP2 NOT installed) possibly to help alleviate ground loops.

Optionally, install a wire jumper at JP3 which connects P3 pin 15 to P3 pin 16 in keeping with placing the 12V rail on both of those pins for the original Raduino and our RadI2Cino card.

Install the voltage dropping resistor R1 (value to be determined between 27 and 35 ohms, 2-3 watts) on the back side and leave about 1/8" between the body of the resistor and the board. It was originally thought that it should be on the front, away from one of the toroid transformers to minimize stray RF pickup but R1 has been moved to a position far enough from the transformer that RF pickup should not be a problem and we need to minimize heat transfer to the Si5351 and its crystal to help maintain frequency stability.

Carefully install BH-1, the surface mount CR2032 3 volt lithium coin cell holder with the positive side to the +_+ marked pad on the front of the board. The + side of the holder will be the one with the two small hooks very close to the edge of the holder. The negative terminal consists of the longer fingers that contact the underside of the coin cell when it's installed. There are + and - markings on the holder but they are difficult to see so be careful that you get this one right.



P4 - There is also a dual row 10 pin male header to the right of P1 which (depending on your preference) can be installed either on the front or back of the

card. Use the same technique described previously for installing and soldering DuPont header pins and inspect your solder joints carefully to prevent shorted pins. If you don't have a dual row header, two 10 pin single row strips are perfectly acceptable.

P5 – Install a 12 pin male header which should be mounted on the front of the card. P5 contains the majority of the extra Digital I/O pins and here is where you will hook up the menu encoder and the majority of the push button connections.

There is a 3.3V pin labeled 3v3 on P5 that has a pad on the back side of the board which may be bridged with solder to provide 3.3V from the Teensy's regulator to this pin. We recommend leaving it un-bridged for now to keep from accidentally connecting 3.3V to something while you are plugging the encoder & push button connectors to P5 initially.

There will be several new push button switches connected to P5 using 3 of the 5 "Spare" pins. See the BITeensio Software instruction manual for the function and connections for these buttons. They had not yet been determined when this manual was written.

Install P6 ON THE BACK side of the card as it needs to be kept clear of the battery holder to prevent accidental grounding of the + terminal of the CR2032 coin cell.

There are two "VIA's" on power carrying traces that are circled on the PC board. It is necessary to ensure sufficient power handling capacity from one side of the board to the other to fill these circled holes with solder. Or, better yet, solder a piece of bare hookup wire through each of these VIA's.

Install MPU1, the user supplied Teensy 3.6 You can either solder the Teensy's pins to the PC board or preferably use the provided female DuPont socket strips for easy removal of the Teensy if it becomes necessary.

Make sure to install the 5 pin cross socket near the micro SD connector end of the Teensy or the voltage from the CR2032 coin cell will not be connected to the Teensy and the Real Time Clock (if utilized) will quit running when main power is removed from the board.

The supplied female header strips assumes the user has either purchased their Teensy 3.6 with pins installed (you still have to install the 5 pins that go across the Teensy near the end closest to the SD card holder) or has the proper pins to allow the Teensy to plug in to the completed BITeensio board.

Finally, install B1, the CR2032 Lithium coin cell battery in the holder with the positive “+” side up (see the picture on page 11).

This completes the construction phase of your new board. Utilization and additional wiring instructions are contained in the Firmware manuals supplied with whichever version of the firmware you chose to use with this “BITeensio”.

There is a power jumper on the backside of the Teensy 3.6, shown on the card that came with your Teensy. It says this should be cut to separate VIN (Teensy power) from VUSB (power supplied through the USB port) if you are using external power which we are by plugging the BITeensio card into the uBITX and if not cut, this could destroy your teensy. You MUST cut this jumper before first powering up your BITeensio card!!!!

Jim Sheldon, WOEB

Jim Giammanco, N5IB

Appendix A:

Additional information:

You will need an I2C enabled LCD display of, at least 4 lines by 20 characters, and a 4 wire connecting cable (not supplied) in order to use this card with your uBITX transceiver. Our software uses I2C address 0x27 as standard, but this can be changed in the Arduino sketch. Instructions for doing this are contained in the appropriate software manuals which are included with each software release (to be included with the software). If only the HEX (Binary) files are available, they will be labeled with the I2C address of the display in the file name (either 27 or 3F) so when uploading the HEX file to your Teensy 3.6, be sure to pick the one that matches your display address.

WARNING!

When you plug this card into your uBITX and connect the original uBITX Digital plug to P1 on the BITEensio, you **MUST** remove the 4.7K resistor between the green and blue wires that were originally required for CW operation using the supplied "Raduino" card. We use the same Blue wire for the CW keyer's "Dot" paddle input, but it's now connected to a digital input pin on the Teensy and does not require the 4.7K resistor any longer. In fact, if you do have the 4.7K connected it could cause the CW keyer not to work properly so please remove the resistor. The green wire will no longer have 5 volts DC on it either, it will now be 3.3Volts max from the Teensy's on board 3.3 volt regulator – **DO NOT USE** it for connecting extra 5volt items. You will have to supply them from an external 5VDC source.

You **MUST** also cut the tiny jumper on the back of the Teensy (called out on the back of the Teensy's pin-out card and labeled "Cut to separate VIN from VUSB, if using a battery charger or external power. Since we are using external power this jumper **MUST** be cut. If it's not cut we've found that under certain circumstances simultaneous power applied to the Teensy from both the BITEensio board and the USB connection has the possibility of destroying something in the Teensy. Better safe than sorry and cut the jumper with a sharp hobby knife.

CONSTRUCTION NOTES:

COMPONENT CHANGE NOTE:

Capacitor C-15's value has changed from .1uF chip ceramic, size 0805 to 10uF non-polarized chip ceramic size 1206. The proper capacitor has been included in all kits since the change. This is NOT a critical item and for those who received kits that still call for C-15 to be .1uF do NOT have to change it unless they want to. The new 10uF component will have to be sourced locally as it would be prohibitively expensive for us to send this one tiny component to all early kit purchasers, especially since it isn't a critical change and the BITeensio will work OK without the change. – C-15 is a bypass capacitor on the 3.3V rail and helps eliminate any clicking noise generated by the Si5351a clock chip.

Concerning the I2C Headers,

If the devices are, as usual, powered via the V+ line on the I2C bus, then no harm can be done to the Teensy by connecting a 5V device onto the 3.3V I2C bus. The connected device just might not work with the 3.3V supply. Now, if a 3.3V device is connected onto the 5 V I2C bus, that might be bad for the device, but I don't think the Teensy will suffer. We try to protect the \$35 part.

There are a few I/O lines not already on headers that have uncommitted pads connected to them for access. A header could be installed or wires soldered there to access these if the user desires.

The front and back ground planes along the bottom edge have been relieved enough so that when the board is plugged into the uBITX it won't accidentally short out components that may wind up under that edge.

Forming a socket for the Teensy 3.6 MPU:

Two 25 and one 5 pin female header sockets are provided. Pick an end and pull ONLY ONE pin from each of the 25 pin female headers to make 24 pin sockets. You MUST also, if it hasn't already been done, install a 5 pin male header on the Teensy itself. This header is to be installed across the narrow part in the 5 holes between the pushbutton switch and the SD card socket with the long pins on the same side as the long part of the 24 pin headers on the MPU. If you leave this

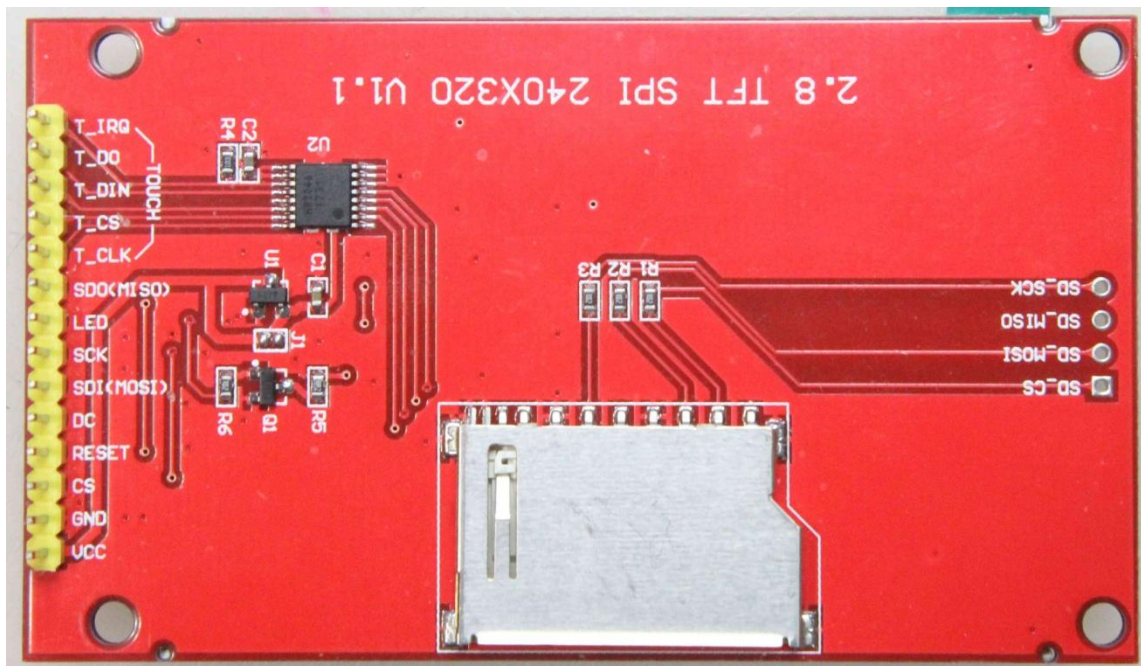
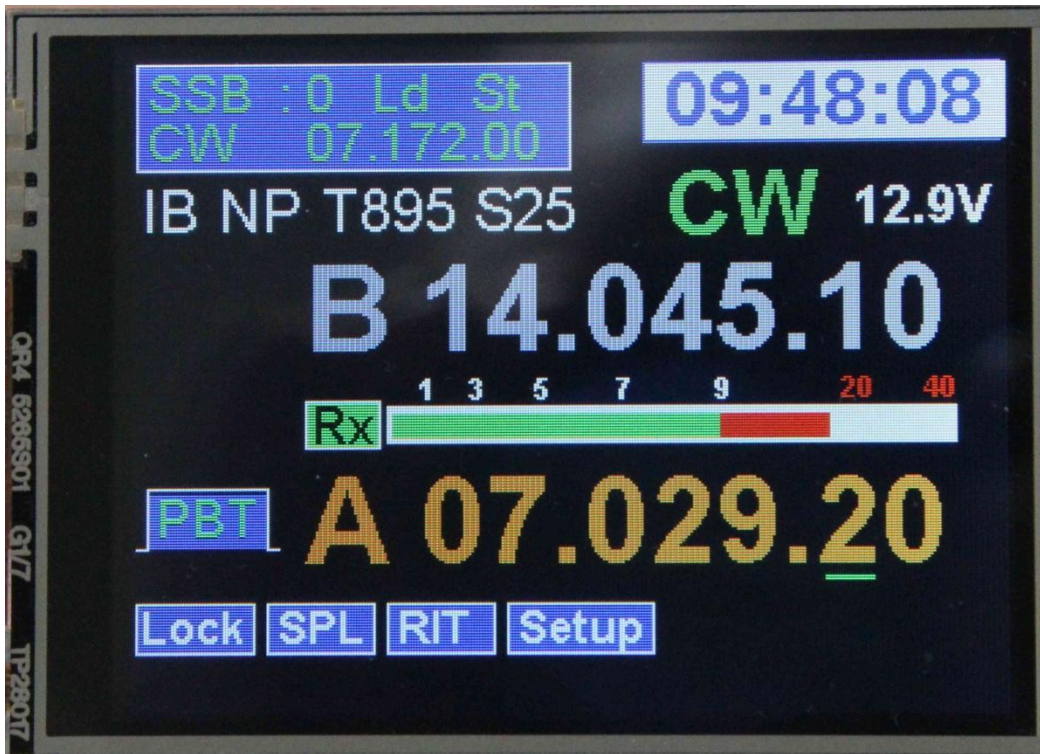
header out, the battery voltage will not reach the Teensy to keep the RTC running when main power is removed.

Carefully file or sand down both ends of the 5 pin female header so that it fits in the 5 cross holes on the board (labeled RESET, PGM, GND, 3V3 and Vbatt) without causing the 24 pin headers to bulge outward when it is inserted.

Dry fit everything, carefully plug the 24 pin and 5 pin female header sockets onto a Teensy 3.5 to position them properly – insert the pins into the BITeensio board on the front side, turn the board over and CAREFULLY solder all 49 pins. Carefully remove the Teensy and check the alignment. If it's OK, your board is complete.

Appendix B

Future optional Color Touch Screen Connections:



Back side of the display showing the 14 pin connector.

On the display board's front side (away from the connector) you will need to jumper the following pins with soldered jumpers or, better yet when you construct your cable, jumper the necessary pins on the connector itself. In this way you will stand less chance of damaging the display.

T_DO, jumper to SDO(MISO)

T_DIN, jumper to SDI(MOSI)

T_CLK, jumper to SCK

Jumper LED and RESET together and also to VCC

The rest of the connections will be as follows

SDI(MOSI) connects to BITEensio P5, D11

SDO(MISO) connects to BITEensio P5, D12

SCK connects to BITEensio P(TFT) D13

CS connects to BITEensio P(TFT) D14

DC connects to BITEensio P(TFT) D15

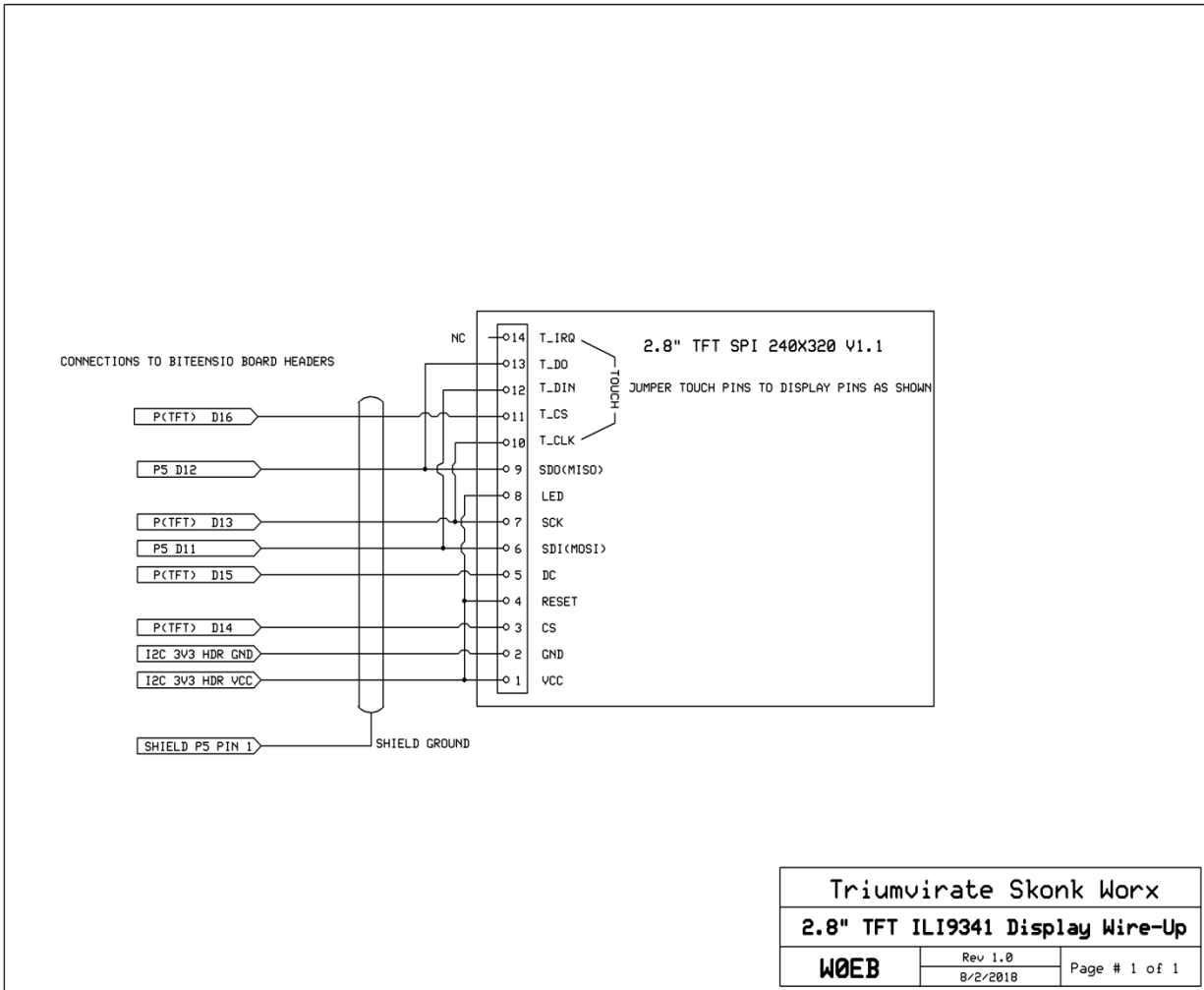
T_CS connects to BITEensio P(TFT) D16

VCC connects to BITEensio 3V3 I2C header (either one) VCC 3.3V pin

GND connects to BITEensio 3V3 I2C header (either one) GND pin

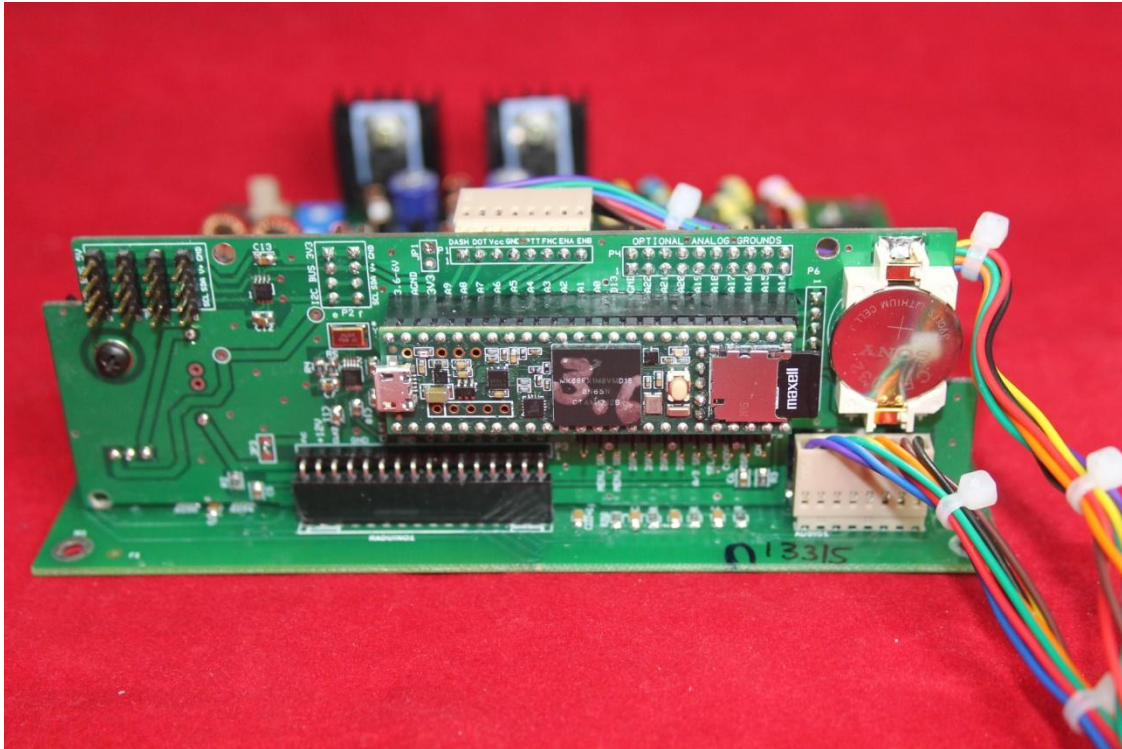
With the proper software this will allow full control of the uBITX from the color touch screen and possibly eliminating all push button switches except the main encoder function switch. The main tuning encoder will still be needed as well as its integral (or optional external) function push button.

Schematic of the shielded cable/jumpers necessary to hook up the display.

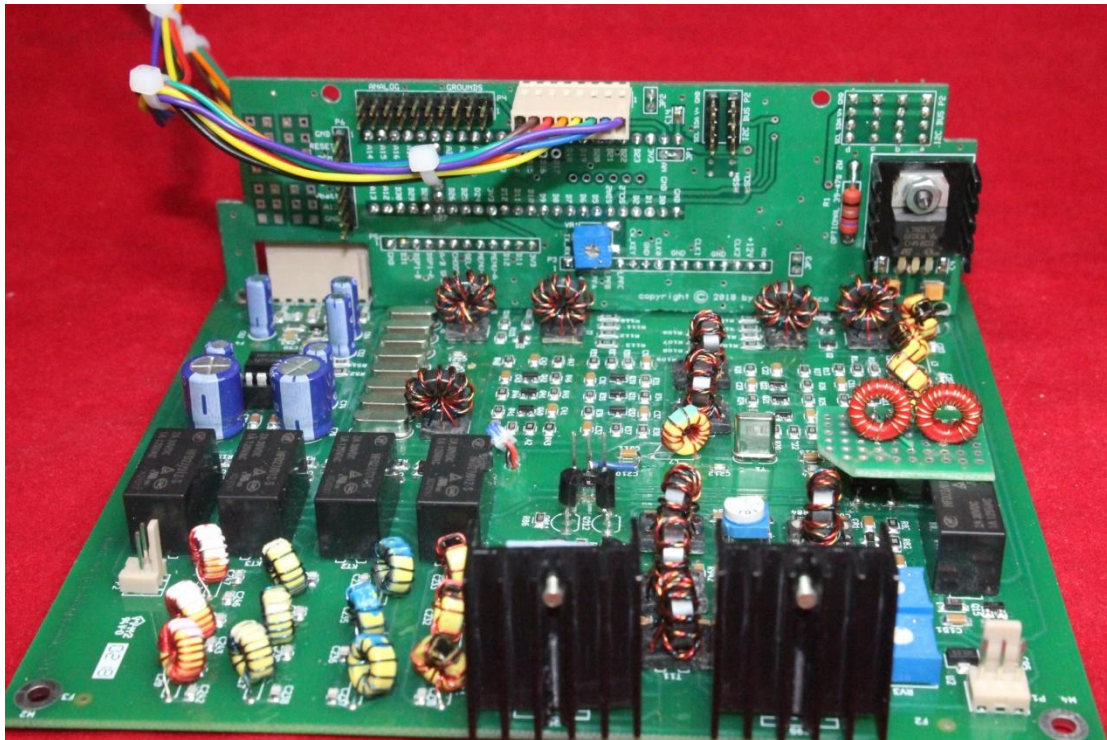


This cable will need to be constructed by the user as it is not commercially available.

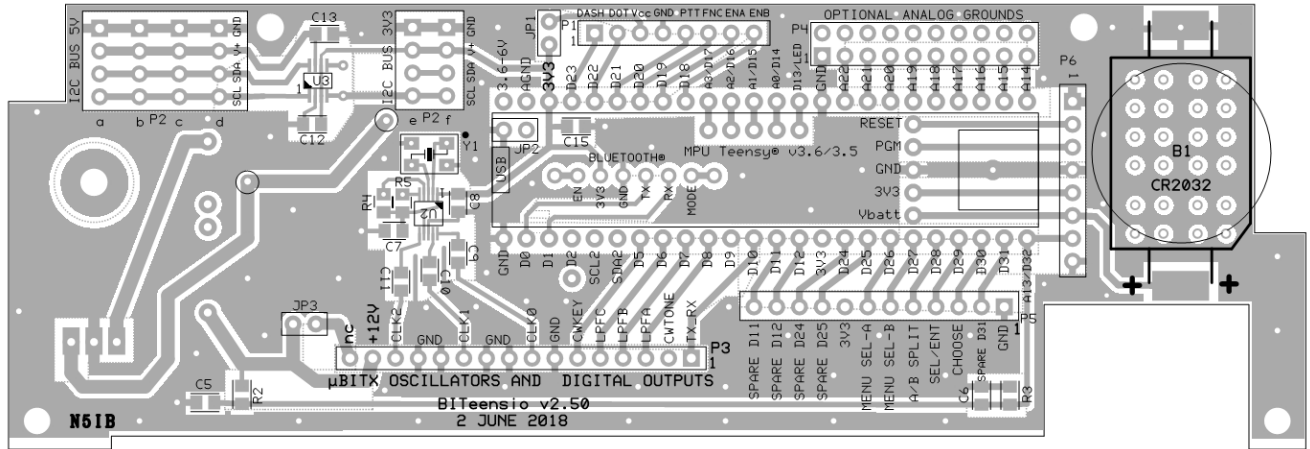
Appendix C:



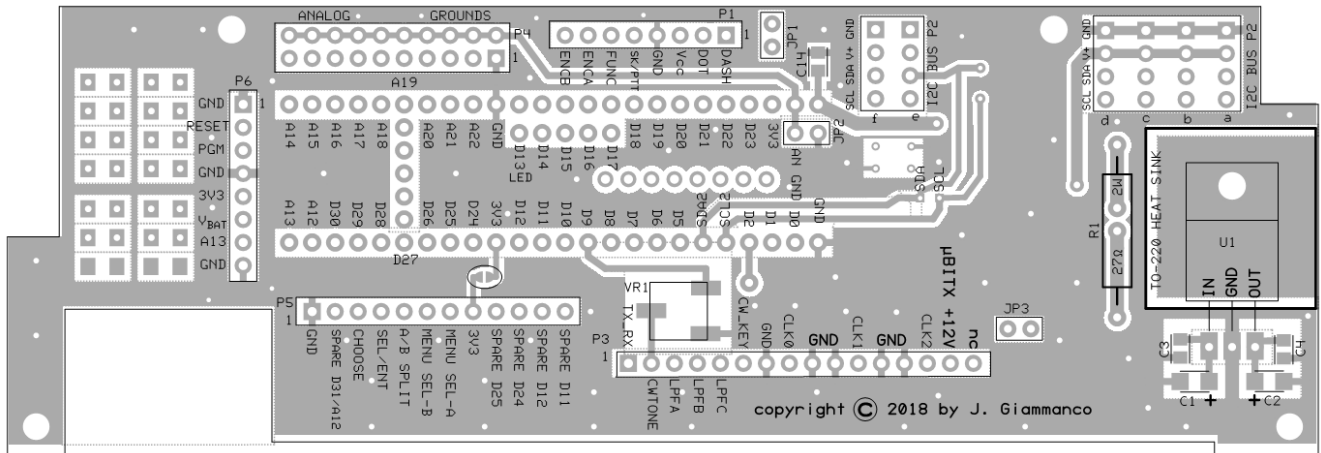
Front & Back reference views of the Completed BITEensio mounted on a uBITX main board.

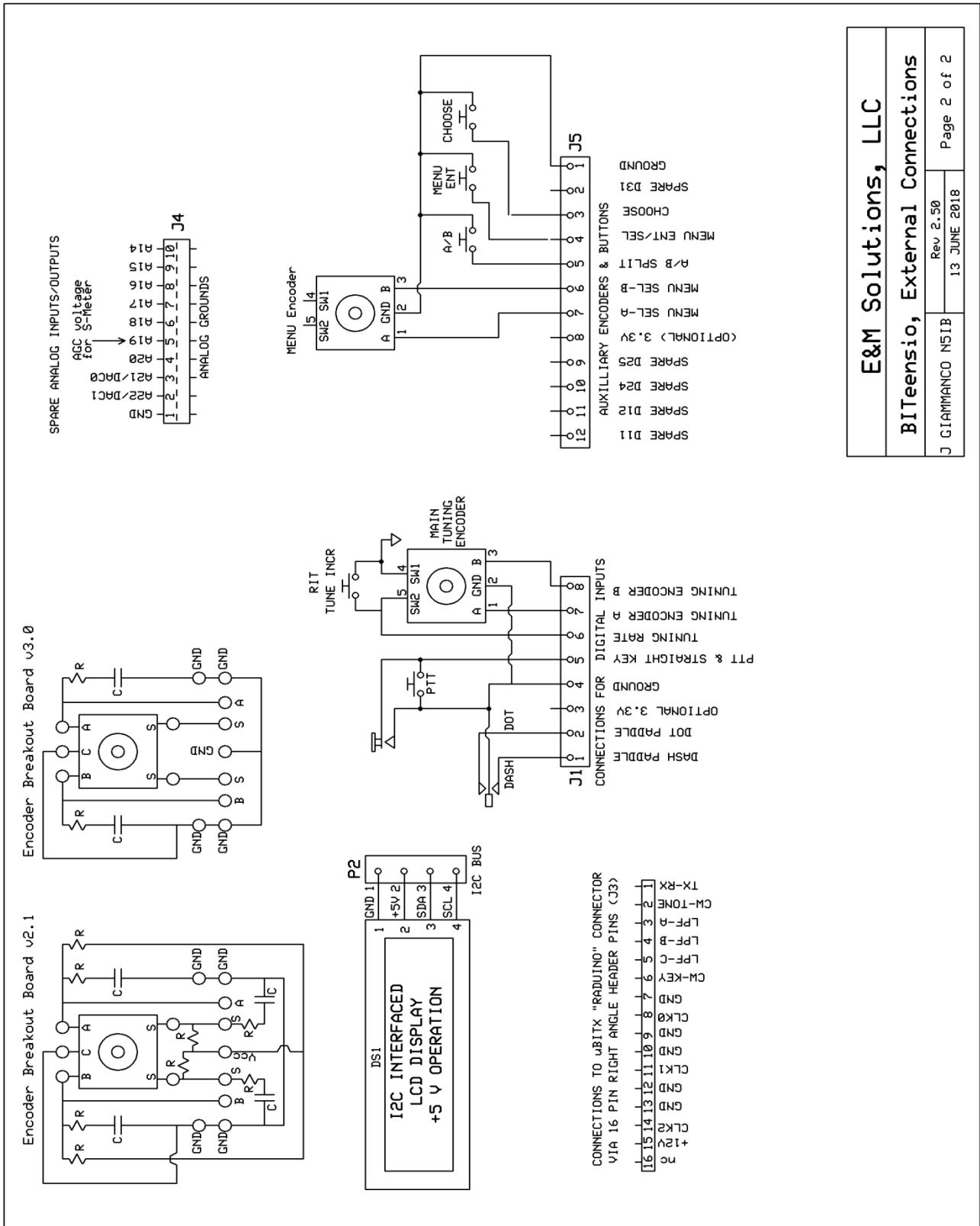


Board Layout Front:



Board Layout Rear:





Appendix F: Special Modifications needed to use 5" TFT Color Display

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 has been covered earlier in this manual.

We also 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.

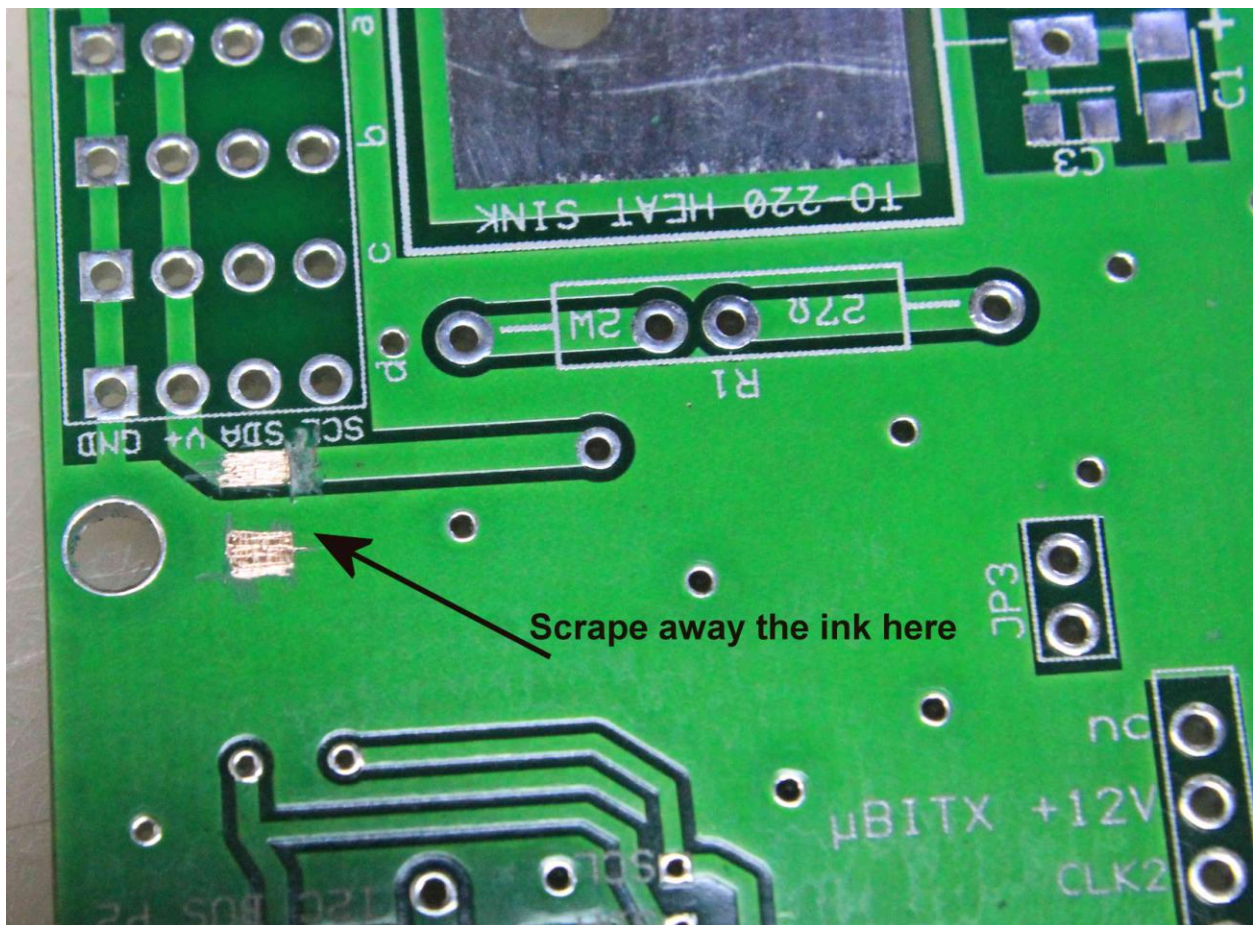
The major advantage of the 5" color display with the BITeensio is with the USB host mode available on the Teensy 3.6, it's possible to use a USB mouse for almost total control of the uBITX. An unlighted USB keyboard can also be used but unfortunately due to constraints of the Teensy's USB host interface 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, 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.

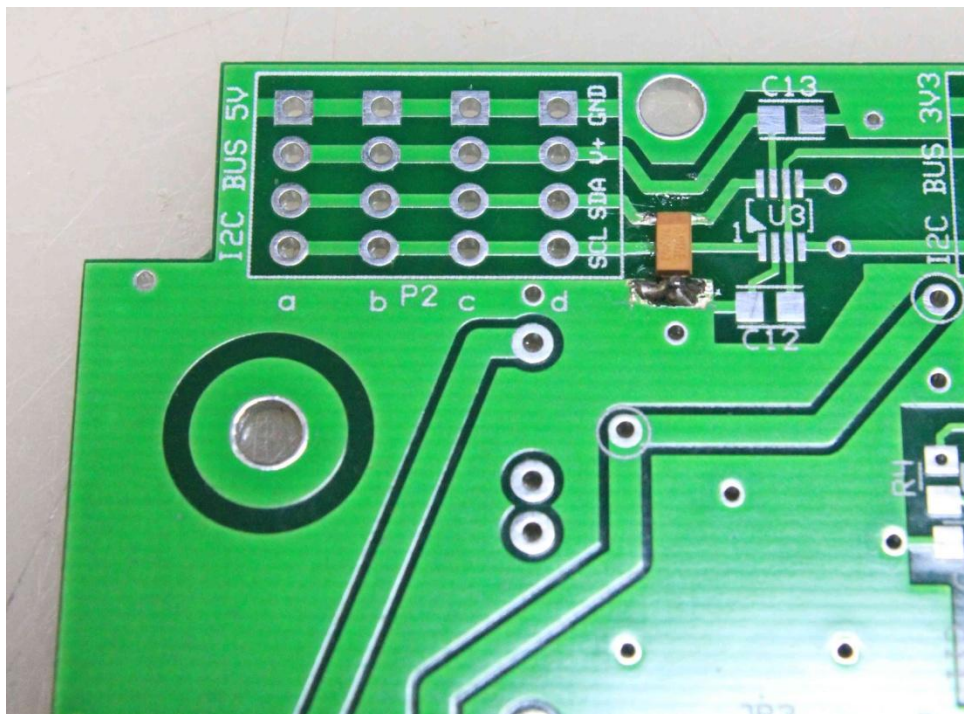
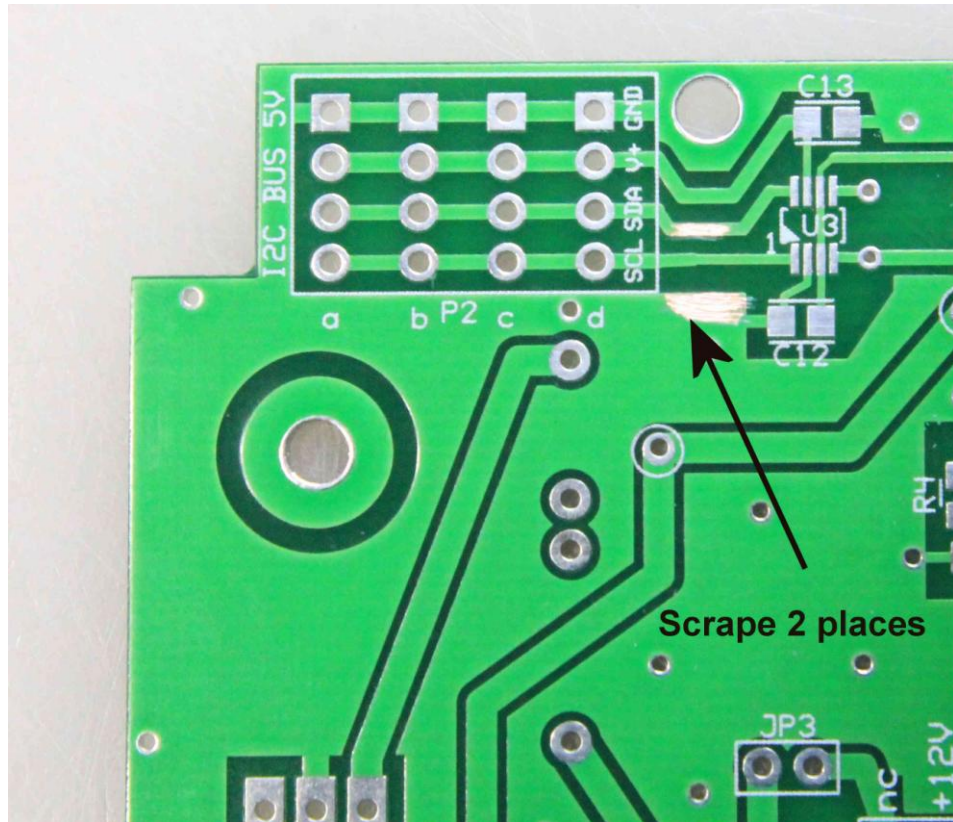
Cutting the indicated trace removes the original 5 volts from the I2C bus. Measure continuity with an ohm meter to make sure there is NO connection across the cut.

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.



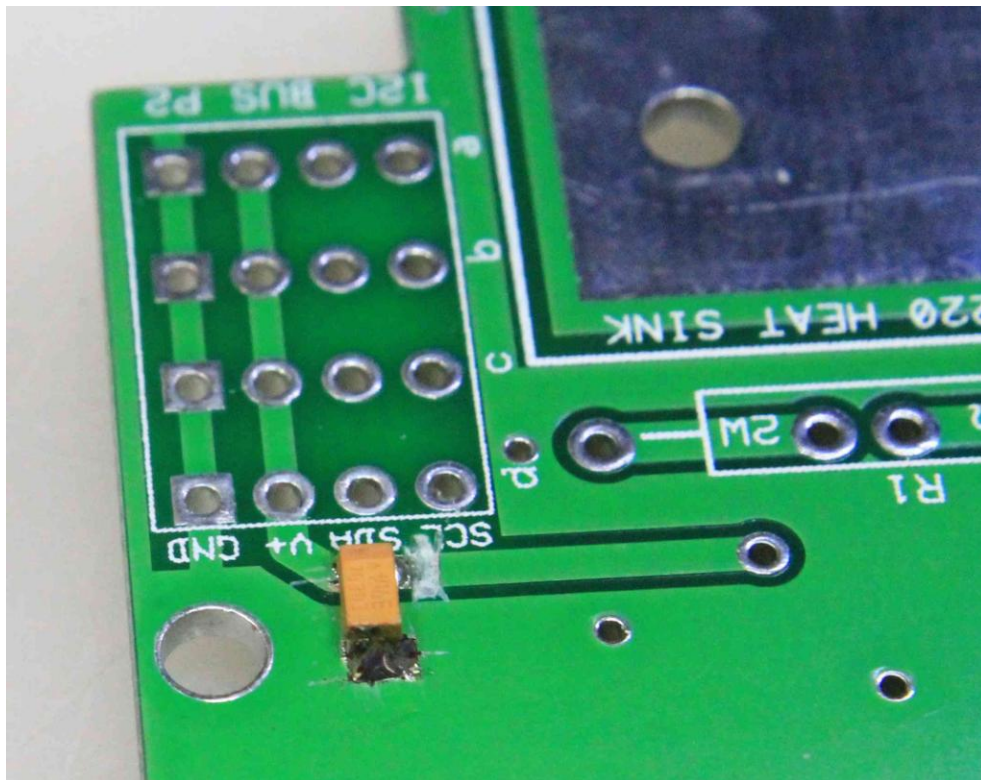
On the other side of the card you will need to create a set of solder pads for the input side to the LD1086V33 regulator as well. No trace cutting is needed for that if the TXS0102 Level shifter is not mounted so if you are modifying an already built BITEensio, it is suggested that the level shifter 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 to the +V trace. The other side of the cap gets soldered to the ground plane where you scraped a pad for it.

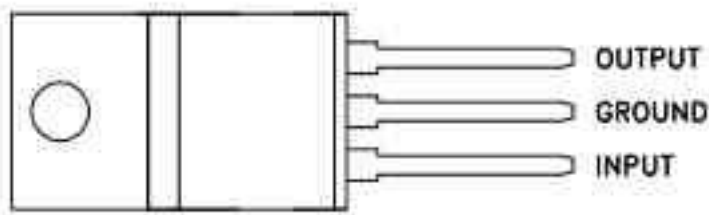


This completes the most difficult part of the modification. Next, you need to decide which side of the board you want the LD1086V33 regulator to stand out from. There will have to be a heat sink on this IC and it will depend on whether or not you have enough room between the front of the BITEensio card 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

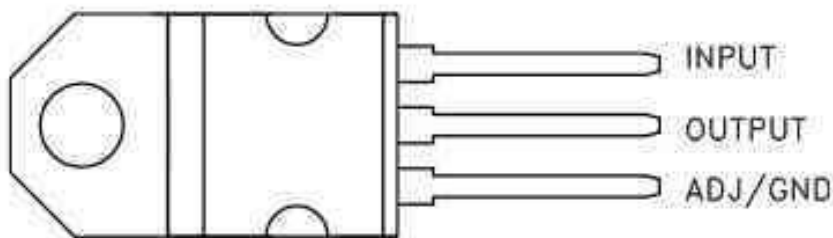
LD1086V33 is hot to 3.3V so you MUST insulate the heat sink from the regulator with a thermal pad and an insulating bushing for the screw to keep the heat sink from causing a short if it comes in contact with ground or something else. The 7805's heat sink is grounded and contact between the two heat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

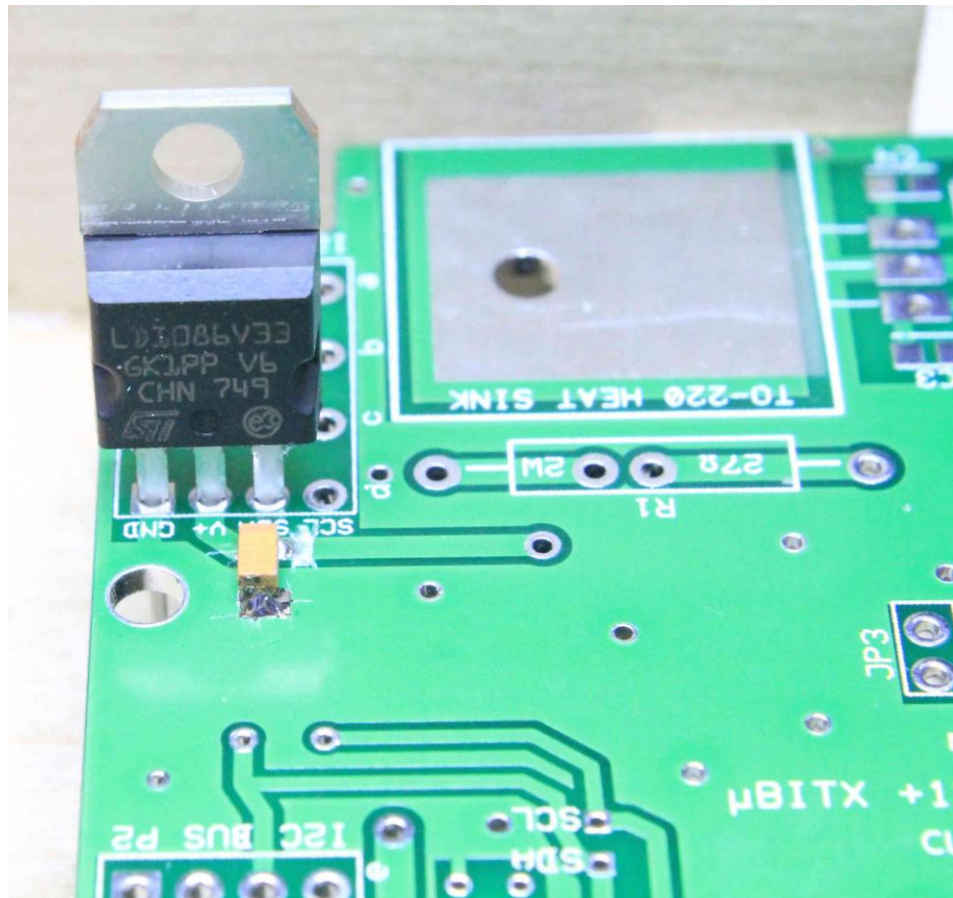


Now the one we are using for the 1.5A 3.3V regulator, the LD1086V33 has a completely different pinout which just happens to be to our advantage for this modification.



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.

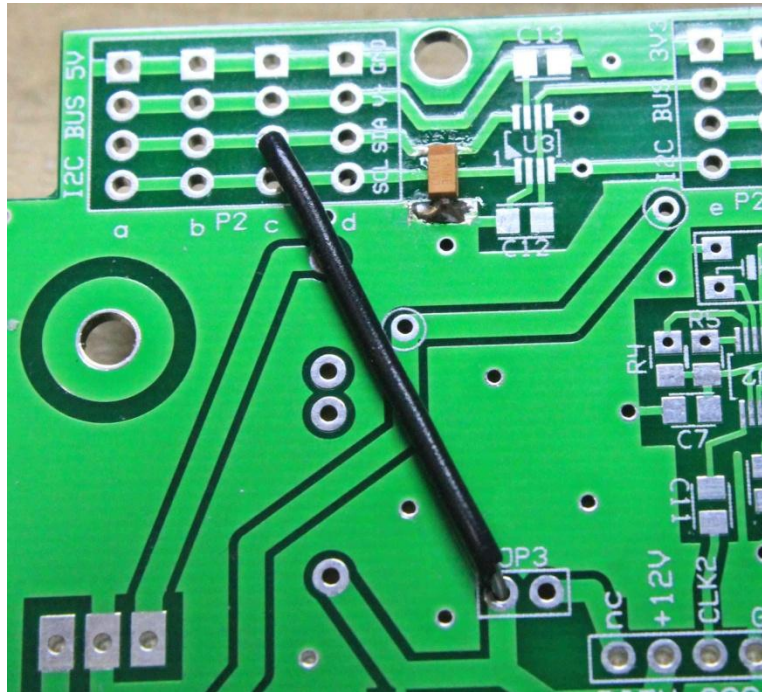
Looking at the layout of our modified 5V I2C header area, reading from the top down we have GROUND, +V and SDA (we are co-opting the SDA trace as input to the regulator. Note that if we put the LD1086V33 on the back side of the BITensio, the ADJ/GND (in this case GND) goes into the top hole, the output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

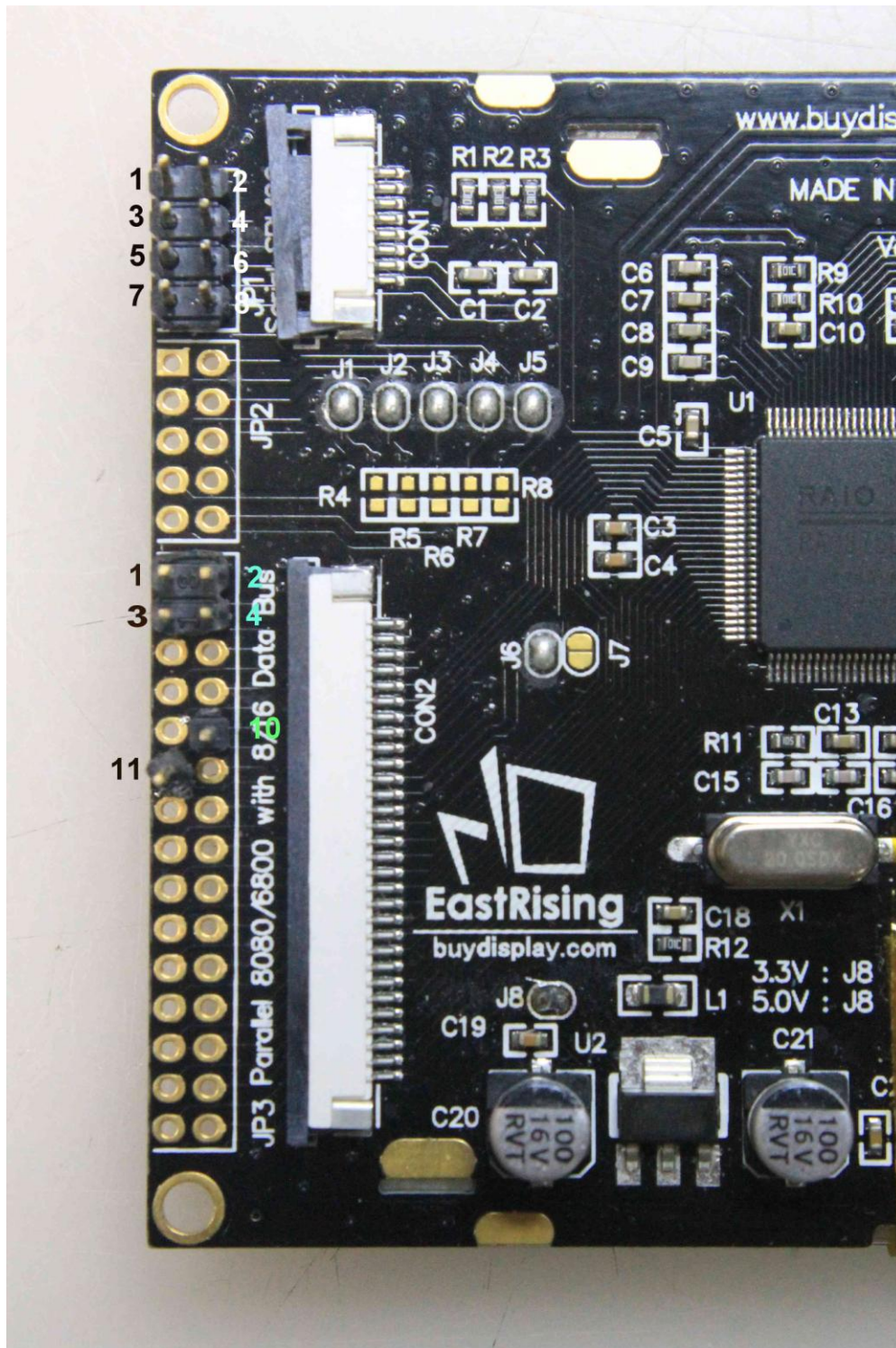


In this picture, shown modifying a new board at build time, you can see that the TXS0102 Level Shifter, U3 is not installed. If you are modifying an already built board, you must either removed U3 or cut the SDA trace between pin 8 of the TXS0103 and the + side of the 10uF tantalum capacitor to prevent 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.

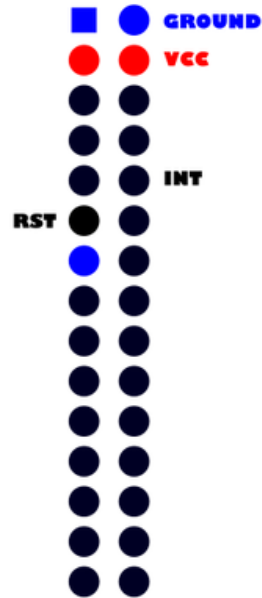
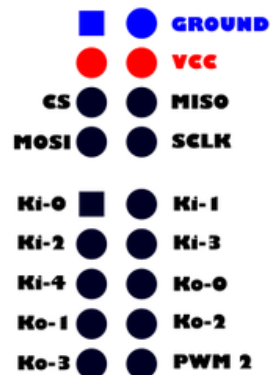
If you are doing the mod on a new board, at build time, you might still want to install the two .1uf SMD capacitors at C12 and C13 as these provide additional RFI bypassing for the 3.3V line on the Teensy itself and also the 3.3V output line of the new power supply.

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.

[BACK SIDE] EastRising ER-TFTM050-2
480 x 272 - 5" - Resistive Touch



VCC can be 5V or 3V3 (JP8) but signals are always 3v3!

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 BITensio 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.

Appendix F, Bill Of Materials (BOM):

BH1	SMD battery holder, for 1 CR2032 coin cell eBay #23161149483
C1, C2	10 uF Tantalum Electrolytic SMD 1206 or 1210 size Mouser 581-TCJA106M025R0150 (for size 1206) Digi-Key 478-9408-1-ND (for size 1210)
C3-C14	0.1 uF 50V MLCC SMD 0805 Mouser 77-VJ0805Y104KXAAT or Tayda A-3511
C15	10 uF 25V MLCC SMD 1206 Mouser 963-TMK316BJ106KL-T
JP1-JP3	wire jumpers recommended.
P1	8-pin, polarized “wafer” connector Tayda SKU-A-810
P2	6 pieces, 4 pin, vertical male header strip Tayda SKU-A-197 (20 pin break apart strips)
P3	16 pin, right angle, male header strip Tayda SKU-A-199 (20 pin break apart strip)
P4	20 pin (2 x 10) vertical, male header strip (optionally single 10 pin strips can be used.) Tayda SKU-A-197 (20 pin break apart strips)
P5	12 pin, vertical, male header strip Tayda SKU-A-197 (20 pin break apart strip)
P6	8 pin, vertical, male header strip Tayda SKU-A-197 (20 pin break apart strip)
P(TFT)	5 pin, vertical, male header strip (optional for ILI9341 Color Touch Screen if used)

R1	27 ohm, 2 Watt 10% Metal Film Resistor, Non-Inductive Mouser 594-PR02FS0202709KR5
R2	10K 0805 5% 1/8 W SMD Mouser 754-RS2012P-103-DT53
R3	2.7K 0805 5% 1/8 W SMD Tayda A-0375
R4, R5	2.2K 0805 5% 1/8 W SMD Mouser 652-CR0805-JW-222GLF
VR1	5K trimmer potentiometer Mouser 652-3361P-1-502GLF
U1	LM7805 5V series regulator, TO-220 Mouser 512-LM7805CT
U2	Si5351a clock generator Mouser 634-SI5351A-B-GT
U3	TXS0102 logic level translator Mouser 595-TXS0102DCTR
Y1	25 MHz crystal, parallel mode, 10 pF, Mouser 559-419B-25-7
HS1	heat sink, TO-220 .75" x .75" x .3" Mouser 532-507302B00
PCB	N5IB printed circuit board V2.30 (design copyright© N5IB 2018)

Off Board and User Supplied Parts:

B1	CR2032 3 V Lithium coin cell (user supplied)
MPU1	Teensy 3.6 microcontroller
ENC1-ENC3	Rotary encoder (user supplied) ENC1 is the one originally supplied with the uBITX kit)

Digi-Key PEC11R-4020F-S0012-ND (No Detent, 12 PPR)

SW1-5 SPST momentary pushbutton switch
(user supplied to their preference)

Teensy mounting socket, Female headers – two 25 pin (carefully cut one pin off each to make 24 pins) and one 5 pin female header. These can be made from 40 pin header strips cut to size, which may be a bit cheaper.

Tayda SKU: A-196 (40 pin female strips)

Tayda SKU: A-1311 (25 pin female strips)

Tayda SKU: A-1304 (5 pin female strips)

Hardware #4-40 x 1/2" machine screw and nut
#4 flat washer
Available from most hardware stores

Mouser Electronics URL: <https://www.mouser.com>

Digi-Key Electronics URL: <https://www.digikey.com>

Tayda Electronics URL: <https://www.taydaelectronics.com>

Design & testing, N5IB/W0EB

PC Design and layout, N5IB, Copyright © N5IB, 2018

Manual & photography, W0EB