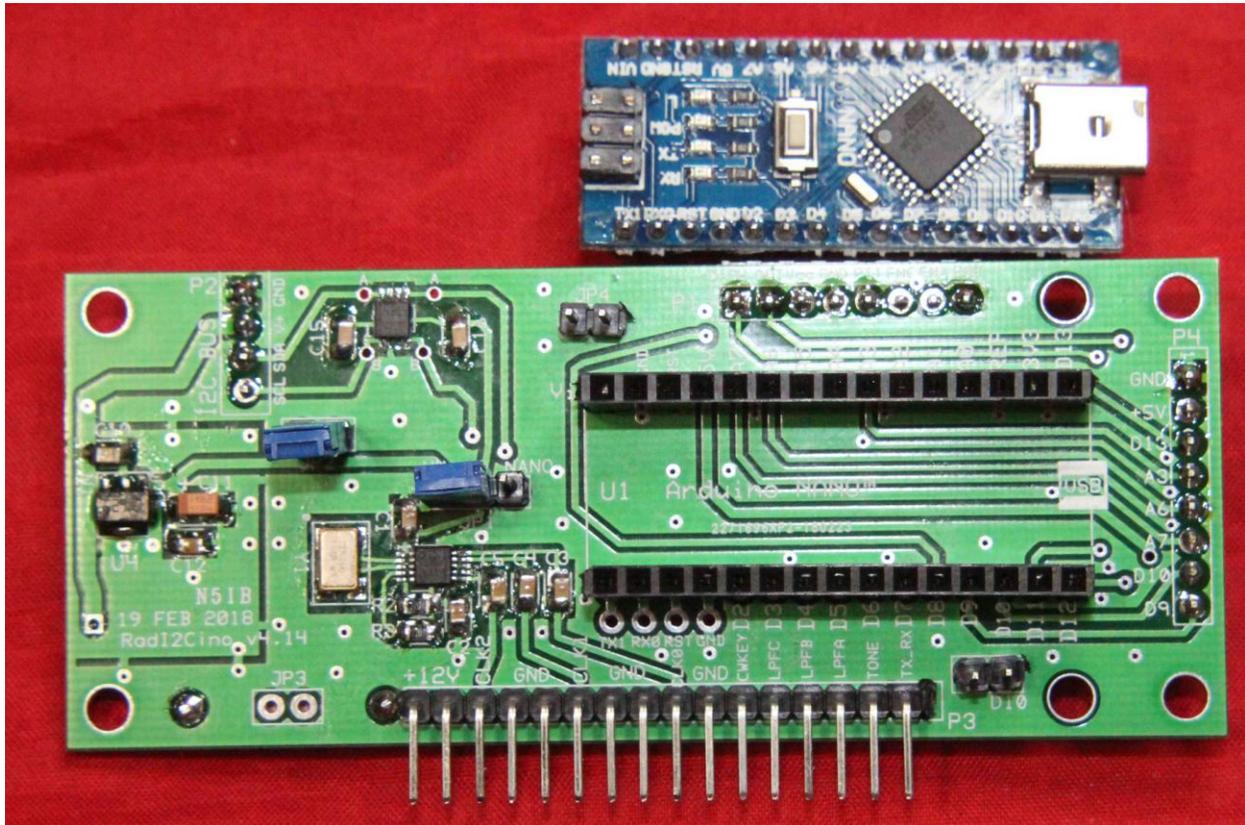


W0EB/N5IB RadI2Cino Construction

Manual Rev 1.18

Copyright © W0EB and N5IB May 04, 2018, all rights reserved.



The RadI2Cino (Pronounced Rad ee too CEE no) was designed to replace the “Radino” card for the Micro BITX (uBITX) transceiver to allow the use of the built in I2C data bus. Using the I2C bus to operate the LCD display allows us to utilize the digital I/O lines that previously were used only for display data & control.

The RadI2Cino card along with the software for it, makes use of these digital lines for a much improved CW Keyer capable of Iambic Modes A and B operation as well as “CAT” control of the transceiver from external programs.

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:

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.

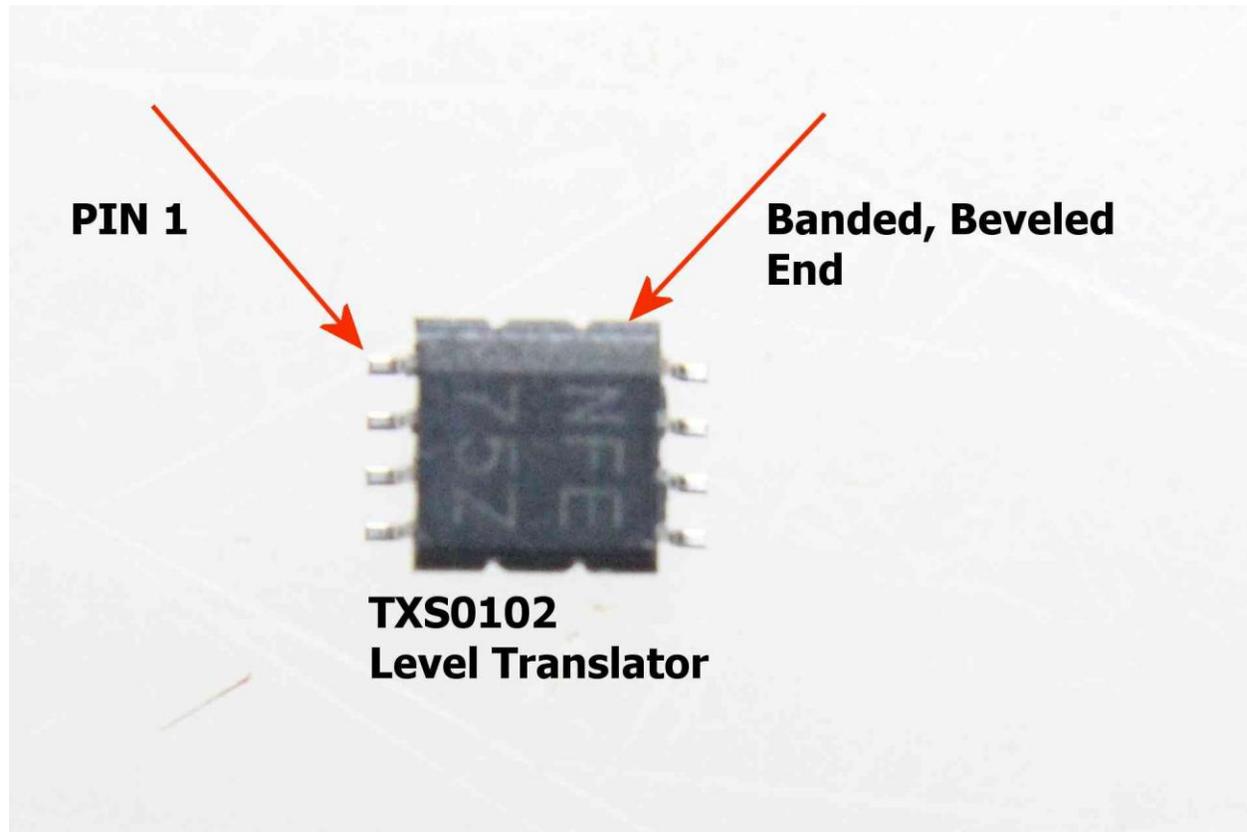
Constructing your RadI2Cino:

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 U5, 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 chip and place that pin over the pin 1 pad on the board. 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.

Using exactly the same procedure you used to install U2, install and check the TXS0102 Logic Level Translator IC, U5. The picture shows the orientation of PIN1.



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. If you look at the picture on the first page of this manual and orient the crystal like you see it there, it will be installed correctly. The crystal is designed so that it cannot be installed wrong as the upper left and lower right contacts connect to the internal crystal and the other two connect to the case for shielding. If you orient it as in the picture, no matter which end is up it will be okay. 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 carefully 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 so you need to solder all 4. Look at the picture on the first page.

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.

Solder the 2.2K 0805 SMD Resistors R2 and R3 near the bottom of the Si5351 chip.

Next, carefully solder the .1uF 50 volt, 0805 ceramic chip capacitors in place at locations C1, C2, C3, C4, C5, C10, C12, C15 and C16 on the front side of the board.

Solder the LD2981CU33TR 3.3 volt regulator IC in place at U4 make sure to solder the tab to the ground pad under the regulator use a little extra solder on the tab AND the 3 pins.

Finally, install and solder C11, a 10uF 25 volt Tantalum capacitor size 1206 or 1210. If a size 1210 is supplied, it's slightly wider but will fit the pads. **Be careful when soldering the 1210's to not short the solder tabs to close-by components.** Carefully observe the capacitor's polarity. The dark colored bar on one end is the + terminal, and should be soldered to the pad marked + on the board. **Make absolutely sure the capacitor's contacts are bonded to the tab it sits over. Bad connections here may allow the regulator to break into spurious oscillation. This could cause the Si5351a clock chip to have the 3 oscillators modulated at an audio rate which could manifest itself in a loud squeal in your headphones or speaker do to that audio modulation being applied to the BFO frequency!**

The rest of the SMD components will be installed on the rear of the board 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 C7 and C9.

Optional but recommended: Install 10K resistor R4, the optional .01 uF ceramic chip capacitors at C13 and C14 and finally the last 10K resistor R5. These components are used in de-bouncing the encoder signals.

Install the SMD 7805 DPAK-3 Regulator IC at U3 use a little extra solder on the pins and enough heat and solder to insure the tab is solidly connected to the ground plane. A little flux under the chip will definitely help here.

Install 10uF size 1206 or 1210 Tantalum capacitors at C6 and C8, carefully observing the polarity marked on the board. As with the other polarized Tantalum capacitor, C11, the colored line on the capacitor is the + terminal and must be soldered to the pad marked + on the board. **Make sure the terminals on both ends of these capacitors are properly soldered to their respective pads – see the caution concerning the 3.3 volt regulator IC and it's 10uF tantalum capacitor. Same conditions apply here.**

That completes the SMD component installation. Final construction consists of installing the connection headers, the (optional) voltage dropping resistor or shorting jumper.

I2C header P2 – If you are planning on using standoffs (not supplied) and mounting your display to the RadI2Cino card, you will likely need to mount this header on the rear of the board and we recommend you place the connecting pins pointing toward the “Digital Connector”, P1. This will keep the connector clear of the display mounting screw on that end of the board. Otherwise it's your choice which side you mount it on and you should determine the most convenient side and positioning for your proposed installation.

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.

On the back side of the board, install P1 using the same technique, but solder the pins to their pads on the front 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 could damage the board and the radio's encoder.* CAUTION! There is a tiny ground "VIA" between the pin 6 and pin 7 pads for P1. Do not use too much solder here or you could short either the function switch or the encoder phase A to ground. Too much solder might bridge both pins to the VIA and to each other so care is required in soldering these pins on P1.

Next, solder the 16 pin, right angle header at P3. It's okay to put the header on either the front or back of the board but on the original Raduino supplied with my uBITX, it's on the back. If you have enough room between the installed card and the front panel of your uBITX, then put it on the back. If you need about ¼ inch more room, solder it on the front of the board. 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.

Install a 3 pin header at JP1. Then install 2 pin headers at JP2, and JP4. These will be jumpered or not depending on the power option selected. Your board may have another set of 2 header pins located just to the right of P3. These are connected to Digital I/O pin D10 and ground and are for ease of connecting the A/B – SPLIT push button switch. As there may be adapter boards offered at a later date that will plug into the NANO's socket, JP1, JP2 and JP4 could interfere with the mounting of such an adapter. Therefore you may want to mount these jumpers on the back side of the RadI2Cino card rather than on the front, contrary to the way they are shown in the picture on the front page.

P4 - There is also an optional 8 pin header that can be installed to the right of U1. This allows connections to several of the extra digital and analog pins as well as a low current source of +5 and ground. It's up to the user whether or not to install this header. (This header could also interfere with any future adapter cards.)

Install (optional) 27 ohm 2 watt resistor R1 on the front side of the board to keep it away from the toroidal transformer on the main board very close to its position. If you are not installing R1, solder a wire jumper across JP3 at the bottom left of the board.

Finally, install U1, the user supplied Arduino Nano which controls the RadI2Cino card and ultimately the uBITX transceiver. You can either solder the Nano's pins to the RadI2Cino card or if desired use female DuPont socket strips (not provided) to allow the Nano to be easily removed and re-inserted as necessary.

Several powering options are provided, selected by shorting jumpers.

JP1 selects whether or not the Si5351 is powered from the onboard 3.3V regulator or from the Nano's 3.3 volt rail.

JP2 selects the power source for the Nano – if installed, the Nano is powered from the uBITX 12 volt rail ***

JP4 selects the Nano's 5 volt pin for powering the display controller from the Nano's 5 volt rail**

** If both JP2 and 5 volt regulator U3 are installed, open jumper JP4. If JP4 is left in while powering the Nano from the 12 volt rail, the Nano's on board 5 volt regulator may be destroyed which will let the rest of the "magic smoke" out of the Nano in short order.

The recommended settings are JP1 set to power the 5351 from U4, the 3.3V regulator, JP2 installed powering the Nano from the 12V rail, and JP4 open, powering the display from the RadI2Cino's on board 5 volt 1 amp regulator. To test the card & display with the NANO hooked to the USB port, without plugging it into the uBITX, you will need JP4 jumpered and JP2 open. Remember to put them back the other way (JP4 open and JP2 jumpered) before use.

***If you chose to omit the optional 39 to 47 ohm 2 watt resistor R1, DO NOT EXCEED 12 volts on the input to the uBITX or you risk destroying the Nano.

Another very helpful modification would be the addition of an extra SPST pushbutton switch in parallel with or even in place of the tuning encoder's built in switch. The reason for this is when using the encoder's pushbutton for menu and other selections, it is very easy to accidentally turn the encoder while pressing on the knob. That COULD cause the frequency to move from where you want it by several MHz (depends on where the frequency digit select cursor happens to be at the time.) Adding this switch may save you some frustration.

This completes the construction phase of your RadI2Cino board.

You will need an I2C enabled LCD display and 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 Nano sketch. Instructions for doing this are contained in the appropriate software manual (included with the software).

Jim, W0EB w0eb@cox.net

Ron, W2CTX w2ctx@yahoo.com

Jim, N5IB n5ib_2@juno.com

Bill of Materials:

U2 Si5351a clock generator in 10-MSOP package

Mouser 634-SI5351A-B-GT

U5 TXS0102 logic level translator

Mouser 595-TXS0102DCTR

Y1 25 MHz parallel mode 10 pF SMC crystal

Mouser 559-419B-25-7

R2, R3 2.2K 5% 1/10W SMD 0805 resistor

Mouser 652-CR0805-JW-222GLF

C1, C2, C3, C4, C5, C7, C9, C10, C12, C15, C16 0.1 uF 50V MLCC SMD 0805

Mouser 581-08055E104MAT4A C6,

C6, C8, C11 10 uF Tantalum Electrolytic SMD 1206 or 1210

Mouser 581-TCJA106M025R0150 Digi-Key 478-9408-1-ND (for size 1210's)

U4 LD2981CU33TR 3.3 V regulator

Mouser 511-LD2981CU33

R4, R5 Optional but recommended 10K 5% 1/10W SMD 0805 resistor

Mouser 667-ERJ-6GEYK186V

C13, C14 0.01uF 50V MLCC SMD 0805

Mouser 80-C0805C103K5R

U3 LM7805 5V linear regulator in DPAK-3 package

Mouser 863-MC7805CDTRKG

U1 Arduino Nano – user supplied

R1 27 ohm, 2 Watt 10% Metal Film Resistor, Non-Inductive

Mouser 594-PR02FS0202709KR5

JP1, JP2, JP3 and JP4 Selection Jumpers DuPont male header pins
Tayda SKU-A-197, Tayda SKU-A-1324 (shorting bars)

P1 KF2510-8P 8 pin male polarized header for uBITX CONTROLS
Tayda SKU-A-810

P2 4 pin single row male header strip – straight or right angle for I2C bus
Tayda SKU-A-197 (straight) SKU-A-199 (Right Angle)

P3 16 pin male, right angle header strip, Oscillators and control outputs
Tayda SKU-A-199

P4 8 pin single row male straight or right angle header strip, for uncommitted I/O
Tayda SKU-A-197 (straight) Tayda SKU-A-199 (Right Angle)

----User Supplied Parts Not Included In Kit----

U1 Arduino “NANO” available from many sources, eBay, Amazon, Adafruit,
etc.

----Off Board Optional Parts----

SW1 SPST pushbutton (PTT switch part of whichever microphone is used)

SW2 SPST pushbutton (A/B-Split switch connects to D10 header)

SW3 SPST pushbutton (function switch, wire in place of encoder switch)
SW3 is not shown on the schematic – it connects between pin 6 of the “Digital”
plug P1 and ground.

Hand key jack 1/8 or 1/4 inch (user selected)

LCD display with I2C interface backpack you should try for address 0x27 (27 HEX)
but it isn't critical as it can be changed when you compile the software.

Instructions are in the software manual. Available from Amazon, eBay and other
places. Standard 2 line by 16 character or 4 line by 20 character displays can be
used. I2C controllers are also available (eBay, Amazon and other places) and as
long as they are designed to work with whatever display you choose, they should

work with the RadI2Cino card. You may have to change the address of the display in the software but instructions explaining this are in the manual for all I2C enabled software we make available.

Mouser Electronics URL: www.mouser.com

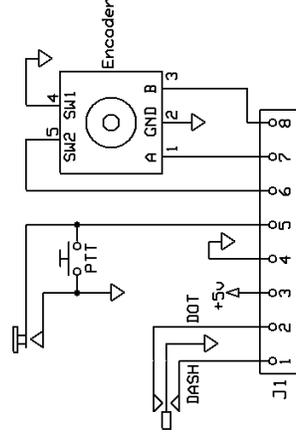
Digi-Key Electronics URL: www.digikey.com

Tayda Electronics URL: www.taydaelectronics.com

RadI2Cino Arduino I/O pin assignments

RadI2Cino pin	Arduino pin	Type	Function
P3-1	D7	Digital OUT	uBITX TX_RX status
P3-2	D6	Digital OUT	uBITX CW Sidetone
P3-3	D5	Digital OUT	uBITX LPF-A select
P3-4	D4	Digital OUT	uBITX LPF-B select
P3-5	D3	Digital OUT	uBITX LPF-C select
P3-6	D2	Digital OUT	uBITX CW keying
P3-7 to P3-16		Clock signals to uBITX, Grounds, and +12V from uBITX	
P1-1	D11	Digital IN	DASH paddle
P1-2	D8	Digital IN	DOT paddle
P1-3	+5	VCC	
P1-4	GND	GND	
P1-5	D12	Digital IN	Hand key / Keyer / PTT
P1-6	A2	Digital IN	Function switch
P1-7	A1	Digital IN	Encoder phase A
P1-8	A0	Digital IN	Encoder phase B
P2-1	GND	GND	
P2-2	+5	VCC	
P2-3	A4	I2C Bus	SDA - SERIAL DATA
P2-4	A5	I2C Bus	SCL - SERIAL CLOCK
SCL AND SDA ARE OPEN DRAIN, SHARED BUS LINES - PULLUPS REQUIRED			
P4-1	GND		
P4-2	Vcc	+5 V	
P4-3	D13	DIGITAL OUT/PWM	- - NANO ON-BOARD LED
P4-4	A3	DIGITAL IN/OUT or ANALOG IN	
P4-5	A6	ANALOG IN ONLY	
P4-6	A7	ANALOG IN ONLY	
P4-7	D10	DIGITAL IN/OUT/PWM	
P4-8	D9	DIGITAL IN/OUT/PWM	

D9, D10, D13, A3, A6, A7 ARE UNCOMMITTED I/O
 D13 is DIFFICULT TO USE AS AN INPUT DUE TO ITS CONNECTION TO NANO'S LED



CONNECTIONS TO RADUINO / RadI2Cino DIGITAL CONNECTOR, P1

NOT TO BE REPRODUCED
 FOR COMMERCIAL PURPOSES

COPYRIGHT © 2018 by James Giammarco

E&M Solutions, LLC	
RadI2Cino	
J Giammarco NS1B	Rev 4.14
18 February 2018	
Page 2 of 2	

Adding the optional uBITX input voltage monitor:

This simple modification might be very useful especially if you are operating the uBITX from a battery and you need to keep an eye on it so you don't discharge the battery too far.

Make up a simple voltage divider using a 10K 1/8 watt 1% and a 2.7K 1/8 watt 1% resistor in series with each other. One end of the 10K resistor connects to the uBITX 12 volt rail (pins 1 or 2 – the 12 volt input to the RadI2Cino) and the other end goes in series with the 2.7K resistor to ground. The center junction of the two resistors connects to the A6 analog input on the RadI2Cino (or Raduino) NANO's board. This allows the software to monitor the power supply voltage to the uBITX by using the analog to digital converter built into the Arduino NANO. If you use 1 percent resistors, the readout will be quite accurate, usually within one or two tenths of a volt. A calibration routine for the voltage monitor has been built into the Alignment setup (see software instruction manual for details on this). The schematic is shown below and also in the software instruction manual as Appendix C.

