Pi-PicoRx (VK2ARH Version)

Construction Manual

Version 1.3

By Richard W5ARH / VK2ARH

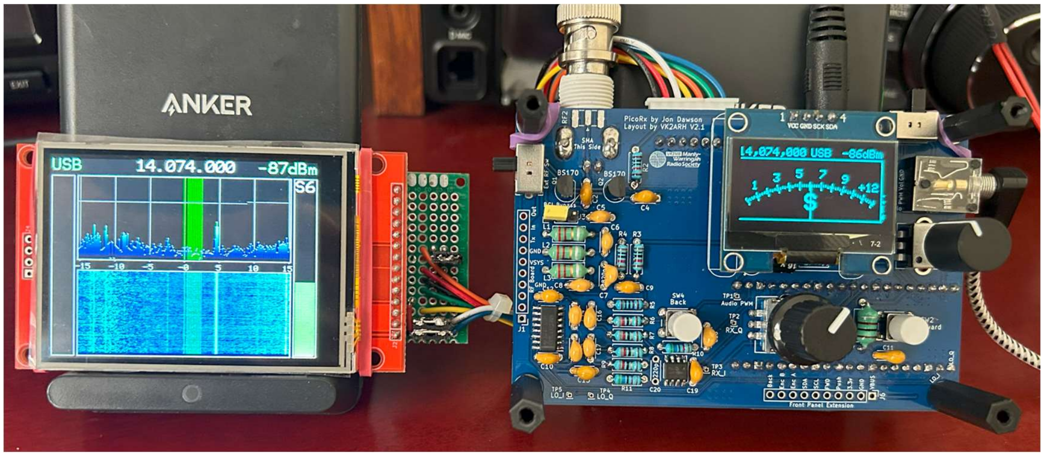
Version 1.0	5th July 2025	Initial Release (Version 1.1 Minor Changes)
Version 1.2	25th July 2025	Correction to External TFT Connection Diagram p33
Version 1.3	12th Aug 2025	Included mounting option for C1 on the underside of the PCB

Table of Contents

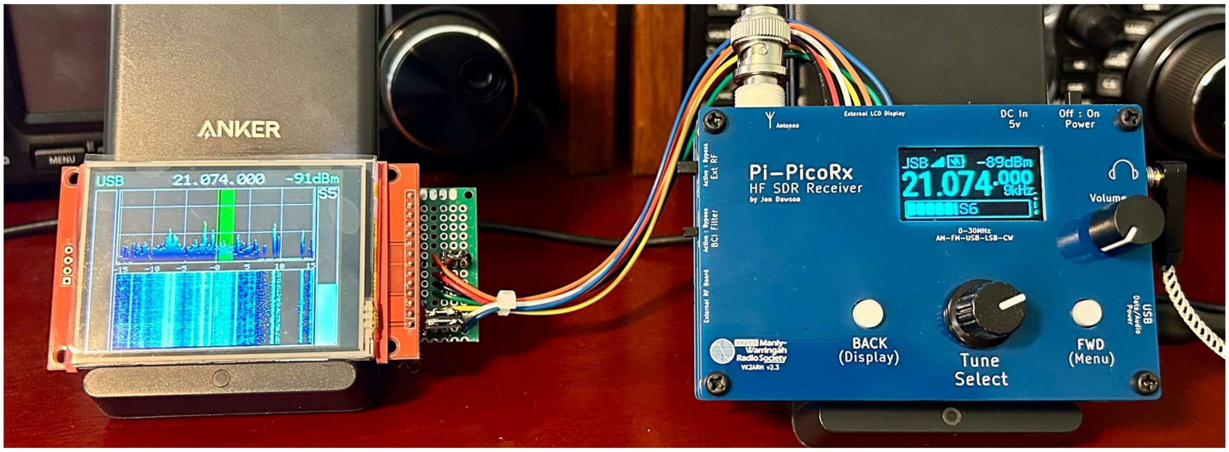
Introduction

The PicoRx was chosen for the Cowtown Amateur Radio Club (Fort Worth, Tx) and the Manly Warringah Radio Society (Sydney Australia) Buildathon programs as one of a series of projects to create interest and build practical skills amongst their Club Members in 2025. The key components of the selection were:

- Low cost using readily available components.
- Use of through hole devices wherever practical – to enable construction by novice/intermediate builders.
- Provide a practical piece of equipment that can be used in the shack or work bench to facilitate building skills, learning, experimentation and have fun both building and using the radio.



The radio can be built standalone or with an optional external TFT waterfall spectrum scope. The PicoRx can be used in conjunction with a companion transmitter and mounted in a larger housing. This project provides a front panel and back cover to enable mounting in a PCB sandwich style of housing as shown below.



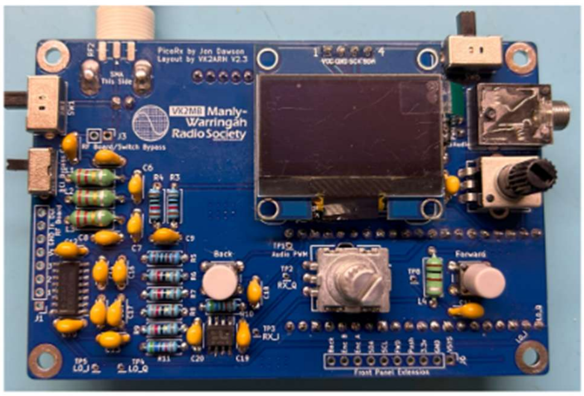
This project is based upon the Tayloe Quadrature Sampling Receiver and the Pi-PicoRx design and software developed by Jon Dawson. A full description of Jon's project, history and theory of operation can be found on his webpage. The three articles on the project shown on the left-hand side menu of his web page make good reading.

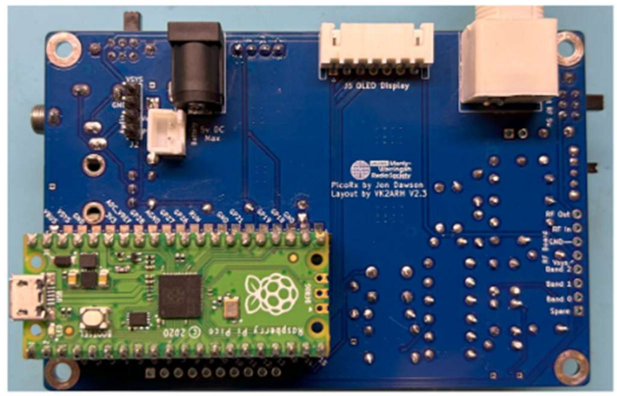
https://101-things.readthedocs.io/en/latest/breadboard_radio.html

The 'Pi-PicoRx VK2ARH Version' is my own implementation of Jon Dawson's Breadboard design together with a BCI filter (Peter parker VK3YE design) and connections for an external LCD Colour Display. This format provides a fully functional PicoRx with all its capabilities as outlined in Jon Dawson's breadboard radio. The layout simplifies the construction process and has been designed to provide a fun, easily built but capable receiver, which can also be used as a foundation quadrature sampling receiver for use as a building block for experimentation with external components to enhance the performance and/or functionality of the PicoRx radio. External components could include a Pre-Amplifier, Band Pass Filters, Audio Amplifier, Battery and Power Management Systems, external transmitters etc.

The design provides several test points to support fault finding and opportunities for learning, experimenting and development. An external user interface connector was incorporated in the layout to facilitate easy remote mounting of the radio controls and screen if mounted in a larger enclosure.

This layout was designed primarily with through hole components and a self-contained PCB sandwich construction to allow it to be built by beginners in Radio Club Buildathon style of events or by individuals. There are only two SMD IC's, however, for those new to SMD component soldering, the form factor of these IC's enables carefully soldered by hand with assistance from experienced kit builders.





This is not a kit to be contemplated unaided by a complete novice to kit building, electronics or radio. However, the Buildathon is designed to enable club members irrespective of experience or ability to come together and with guidance and support from each other, enjoy a mutual learning and fun environment in which to build their project and enjoy our hobby. I would classify the project as being of intermediate built complexity and should take no more than 4 hours for someone with previous building experience to complete.

The manual was written with a novice constructor in mind and is focused on a step-by-step instructional build process. Seasoned kit builders please ignore/excuse the detail in parts. The manual 'speeds' up as you get to later stages of the project as the construction techniques and details have already been covered earlier in the manual.

I recommend that you read the entire manual before you commence construction to gain an understanding of the process to be followed, the contents of this manual and familiarize yourself with the journey ahead.

This manual outlines the building process using the MWRS PicoRx kit which was put together as a 'Group Buy', but all components are readily available from electronics supply companies. The Gerber files to produce the PCB's used in this project together with other additional information can be found at my GitHub site:

Acknowledgements:

A huge vote of thanks to Jon Dawson (for his original PicoRX design), Peter Parker VK3YE for his BCI filter design and those involved in the continued development of the PicoRx software.

Richard
VK2ARH / W5ARH

Kit Assembly

Equipment Needed:

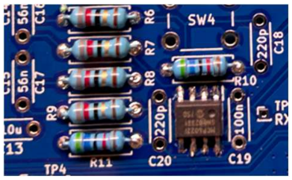
The following equipment is recommended to build the kit:

- Soldering Iron / Station (preferably ESD protected with a fine tip, together with a method of venting fumes away from the soldering area). Soldering the SMD IC's in this project will require a very fine soldering tip.
- Solder: 0.8mm or smaller diameter 63% Sn / 37% Pb with 'no clean' flux is recommended, if you are comfortable soldering with Pb based solder.
- Solder flux can be an advantage in soldering some components but is not essential if you have a good quality solder with embedded flux.
- Wire cutters (fine precision type) and long nose pliers
- Multimeter
- Phillips head screwdriver
- Modeling knife
- Solder sucker or solder wick if needed.
- A magnifying glass or higher power reading glasses can assist with checking the quality of your soldering.

Take the time to prepare your work area and layout your tools, keep it free of clutter. An organized work area helps you to focus on the build and not be distracted by searching for tools and components.

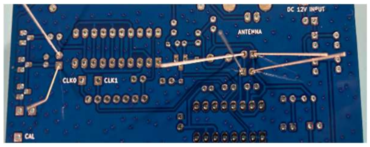
An example of a cleared and ordered work bench ahead of the kit build. Individual equipment will vary.
(My workbench isn't always that tidy 😊)

Construction Tips

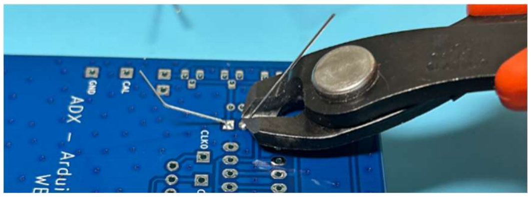


Whilst it makes no difference electronically which way the resistors (in this case) are mounted, good practice is to mount each device with the same orientation. By doing so it makes checking the value of components much easier.

For capacitors or devices with identification marks on them, I try to orient them all in the same direction unless their view is being obstructed by another component. When this happens, I orient them where possible, in a direction that enables them to be read easily.



When you place the through hole components on the PCB hold the component down against the PCB and spread the component leads so that when the board is turned over to solder, the components remain in position. Restrict the number of components you place at any one time to avoid a 'rats nest' of leads that will interfere with your soldering iron tip and the solder whilst soldering.



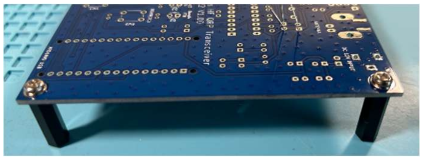
Once you have soldered the components – remove the leads as close as possible to the solder joint using a small pair of wire cutters. This process should be performed after installation of each component group.

I place my finger on the end of the lead when snipping the lead to absorb the 'snap' shock of the cut – reducing stress on the recently soldered join (**but not on static sensitive devices**). It also stops the lead 'flying' away when you cut it. If the lead is soft and you cut slowly this technique is probably not necessary, but it's something to consider.

Don't use excessive heat whilst soldering – this has the potential to damage components and destroy the PCB tracks. I like to tin the solder iron tip and then use the 3 second rule ensuring that the soldering iron is not in contact with the device for more than 3 seconds at a time when soldering each lead. If you need longer, allow the lead to cool and return to complete or improve the solder joint.

Flux or flux paste is not usually needed if you are using a good quality solder with incorporated flux, but it does help if you have trouble with solder flowing. Choose a 'no clean' flux if you have the opportunity as this makes for a cleaner result. If necessary, boards can be cleaned with an old toothbrush and isopropyl alcohol or for those looking for a 'Rolls Royce Solution' access to and cleaning with isopropyl alcohol in an ultrasonic cleaner produces great results.

Hint: Isopropyl alcohol can be purchased from the pharmacy or hardware store. The higher the alcohol content the better. Scrubbing and rinsing with a mild dishwashing detergent and water and then drying with a hair dryer produces good results removing the sticky residue that sometimes is present after cleaning with the isopropyl alcohol, but do this before adding any component that you think will in impacted by the washing or drying process. The rinse is important to ensure that any corrosive detergent residue is removed. If you have access to an ultrasonic cleaner even better, but if that's all too hard – if you used a 'no clean' flux, cleaning is only cosmetic.



You may wish to install the 15mm standoffs onto your main PCB before you start to solder components onto the PCB. This provides a nice stable platform for soldering once you have turned the board over. This same approach can be used in reverse when you have finished the build and want to use the UI to test the PicoRx before you mount the project in the housing.

In this manual each time a component is to be identified and soldered to the PCB, the task is listed separately proceeded with a square check box or listed in a table of similar components. You can 'tick' off the components as you progress, and this keeps a record of your progress through the build. You can also check off each page as you complete the activity listed on that page:

- ☑ Component 1
- ☑ Component 2 etc.

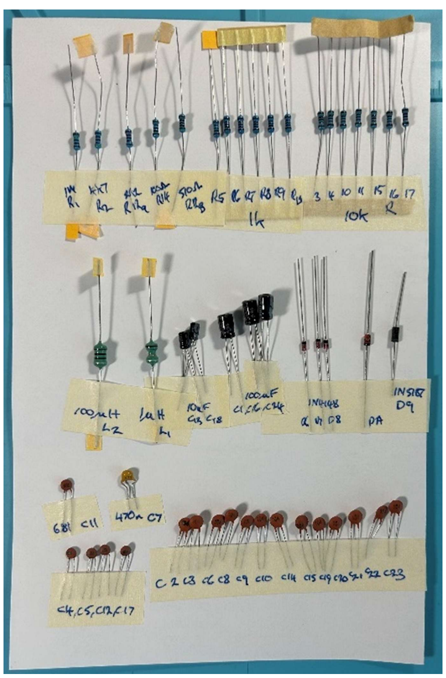
Page | 15 ☑

Bill of Materials (BOM)

Component Group	Designator	Part	Qty
PCB's	PCB Main	VK2ARH Design	1
	PCB Front Cover		1
	PCB Back Cover		1
Hardware	Standoff	7mm M-F	4
	Standoff	15mm F-F	4
	M3 Plastic Screw	6mm	8
	Encoder /. Volume Knob	WH148	2
	Misc	Self Adhesive Silicon Feet	4
	Pin Headers Male	Use 40 Pin Header and snap as required	2
	Ext Disp Headers	1 x 40 Pin Female cut as required	1
	Pin Headers Female x 20	Mounting Pi Pico	2
Capacitors	C1	100u 10v Electrolytic	1
	C2,C9,C10,C19	100n	4
	C3	0.47u	1
	C5,C8	2.2n	2
	C6,C7	820p	2
	C11,C12,C13	10u	3
	C14,C15,C16,C17	56n	4
	C18,C20	220p	2
IC's & Microporcessor	U1	Pi-Pico 2	1
	IC1	SN74CBTLV3253DR	1
	IC2	MCP6022-I_SN	1
Jumpers	J1	8 Pin Header	1
	J2	4 Pin Header	1
	J3	2 Pin Header	1
	J6	10 Pin Header	1
	J7	Solder Jumper on PCB	
Connectors	J4	3.5mm Audio Socket	1
(J5 Not Use)	J8	5.5mm x 2.1mm DC Socket	1
	~	5.5mm x 2.1mm DC Plug	1
	J9	2 Pin JST Connector - Battery	1
	J10	USB_C_Receptacle_PowerOnly_2P	1
	RF1	BNC Socket	1
	RF2	SMA Socket	1
Inductors & Diode	L1,L2,L3	2.2uH	3
	L4	100uH	1
	D1	1N5817	1
Display Hardware	OLED1	1.3" OLED SH1106	1
	Ext Display	2,8" SPI TFT Screen	1
	J5	7 pin plug + cable	1
	Small Prototype PCB Strip	To mount LDC Pin Headers if needed	1
Resistors	R1	100R	1
	R3,R4	10K	2
	R5	1K	1
	R6,R7,R8,R9	82R	4
	R10,R11	56K	2
	RV1	10K Potentiometer (Volume Control)	1
Switches, Knobs and RF Connectors	SW1,SW6	SW_Slide_DPDT	2
	SW2	Micro SPST BCI Bypass Sw	1
	SW3,SW4	6mm x 6mm Push Switch 9.5mm Shaft	2
	SW5	RotaryEncoder	1
	Misc	6mm Push Button Switch Covers	2

☐ Start by unpacking your kit, identifying each component, and checking the components against the BOM.

Whilst every builder will have their own style and approach, identifying, sorting, and labelling smaller components can save time during the build and minimize search effort and errors.

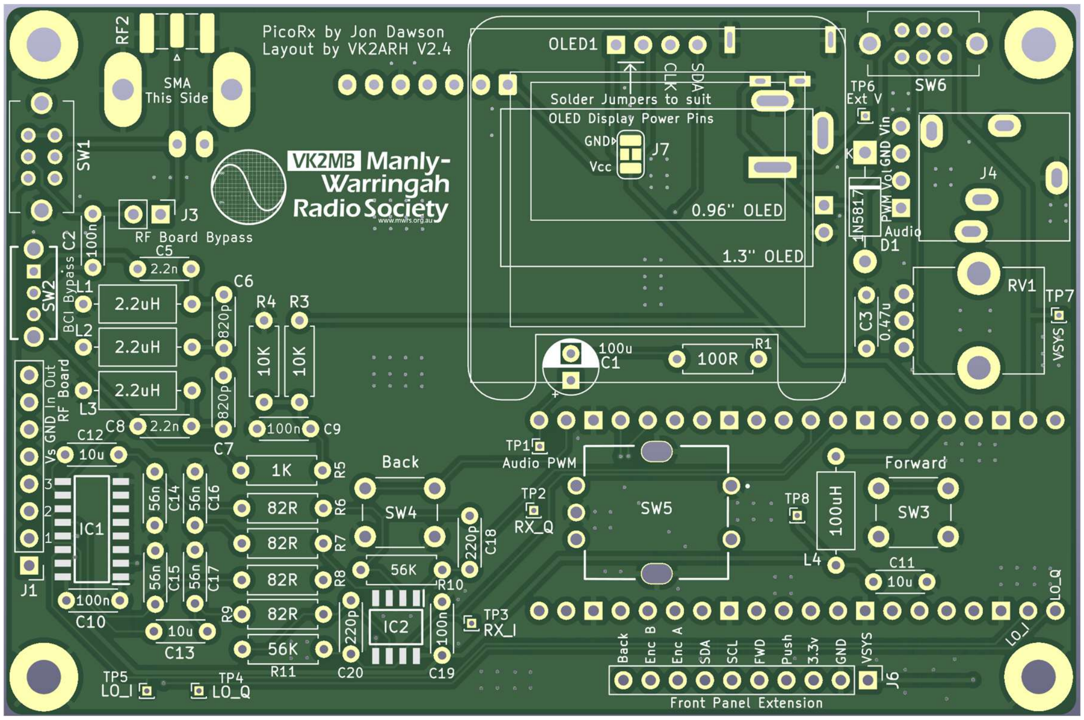


In this example (from a previous Buildathon project) the smaller through hole components have been checked and stuck onto a piece of paper with masking tape before being labelled to speed identification and use during the build. Testing of each component not only assists in identification but ensures that any faulty or out-of-spec component is not used during the build. Don't worry about any adhesive from the tape contaminating the leads, they are soldered up close to the device and not where the tape is used.

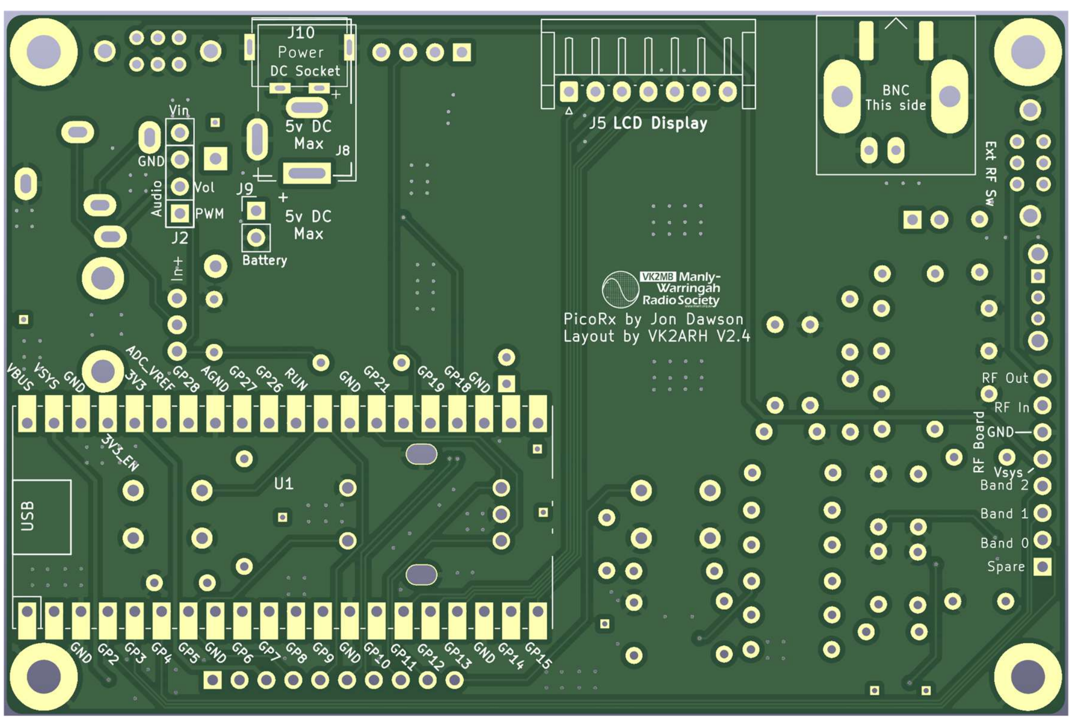
Resistors can easily be tested with a Multimeter; however, capacitors and inductors may need an LCR meter which is not essential but handy if you want to check the tolerances of components. It is not necessary to obtain a high precision meter but be aware and allow for the meter tolerance when measuring components. Most components do not need a high level of accuracy, but you may like to check to ensure that they are within their manufactured tolerance.

PCB and Schematic Diagram

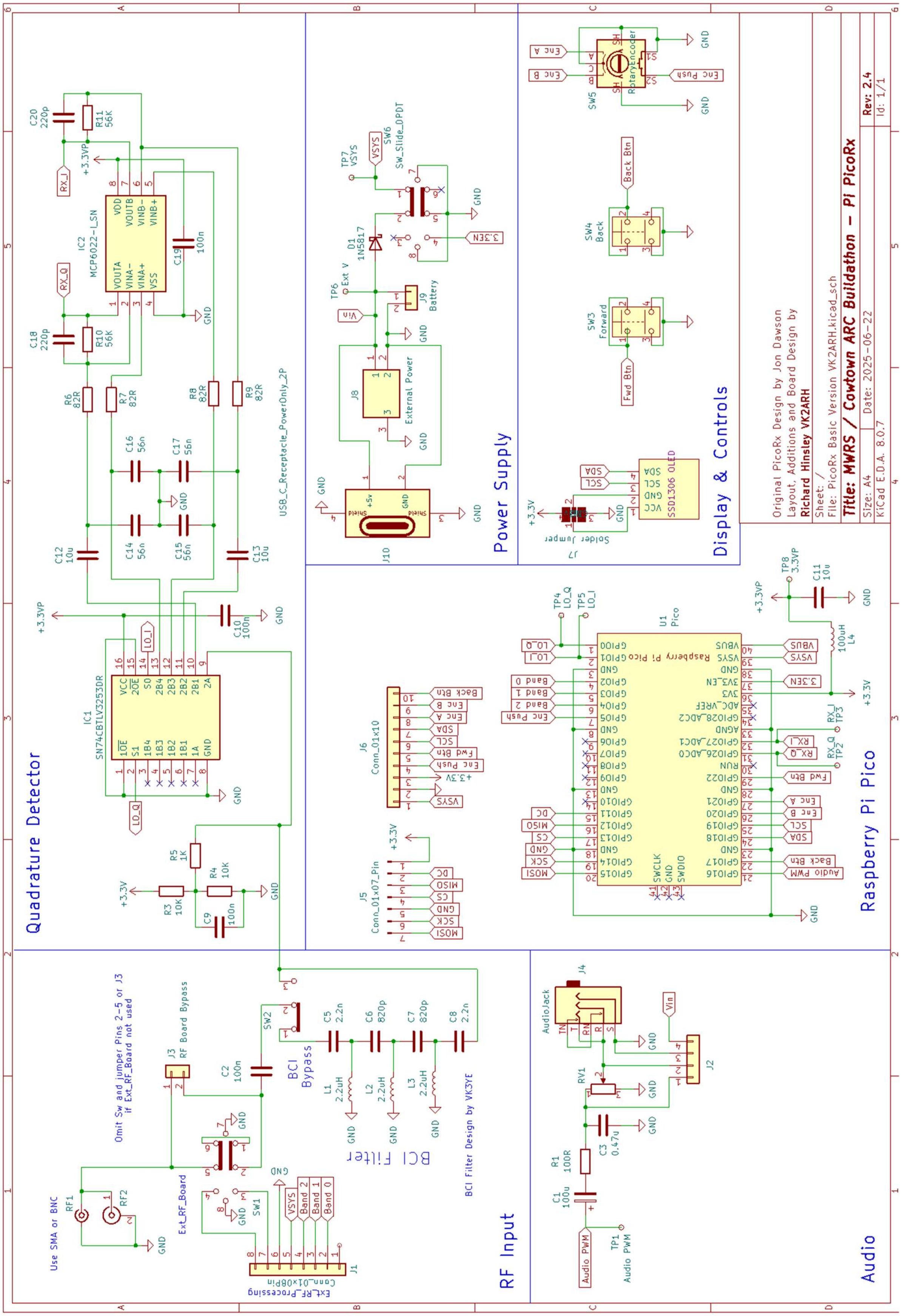
Here is the layout of the component and rear side of the main PicoRx PCB. Note this image show version 2.4 – photographs and diagrams in the construction manual show some version 2.3 board. The only difference was the inclusion of J10 (USB-C connector) and some associated power connections. Details of these changes are discussed in the power supply section of this manual.



Most components are placed on the component side of the board (above) except for the Pi Pico, power connectors, external LCD screen connector and the BNC antenna connector. If using an SMA antenna connector instead of the BNC connector this is mounted on the component side of the board. See P4 for a photograph of component placement.



The schematic diagram for the project design is shown below:



Building the PicoRx

Building the PicoRx is relatively straight forward and the easiest way to assemble the components is to start with the smallest components first and progressively work your way through to the larger components. This approach makes handling and soldering components to the board easier.

Step 1: SMD IC's

The most challenging components to solder are the two SMD IC's. Whilst also the smallest profile, soldering these components first provides maximum access to their footprint to enable the best possible outcome. Those with access to a hot plate or hot air gun familiar with soldering SMD components will have little difficulty performing this task. It is possible to solder the devices to the board by hand using a very fine tipped soldering iron as they are supplied in a SOIC-16 and SOIC-8 package.

The following soldering options discussion has been copied from the W7ZOI DC Receiver Project published by the CalQRP Group.

There are three basic options for soldering SMD parts:

- Hand Soldering
- Hot Air Soldering; and
- Reflow Soldering

Hand Soldering

"Parts are soldered directly to the PCB using a fine-tipped soldering iron and low temperature solder.

The basic technique is to apply solder to one pad of a footprint, use tweezers to slide one end/pin of the device into the molten solder, and then to solder the other end/pins. Care should be taken to align the component so that it is straight and flat. There is some flexibility when only one end/pin is soldered. You can always just unsolder the joint and remove the part with tweezers and let it cool. That's a good time to cool down your temper too if you find yourself frustrated.

On some devices, such as ICs with fine pitched pins, you will find that trying to solder pins individually results in solder bridges. Do not let it bother you. Embrace it! It's a pro-tip to lay down a bead of solder along the pins and use solder wick to remove excess solder. Just be careful to use the minimum amount of heat to remove the solder so the device is not damaged. Solder wick can also be used to remove excess solder from joints if it makes you feel better."

The PCB used in this project has slightly longer pads to accommodate hand soldering, and both SMD parts are SOIC footprint chips which are relatively easy to hand solder.

Some tools you will need for hand soldering SMD work:

- Fine tip tweezers to pick up, hold, and flip SMD parts
- A magnifier of some type such as a headset, low power microscope or a high magnification pair of glasses (I personally use a low cost 3x reading set of glasses for soldering which I purchased from a $1 store (although they cost about $3 😉)
- Low temperature, fine rosin core solder such as those available from Kester.
- Flux, you can/should apply flux to pads before you solder especially to ease the flow of solder.
- A silicone work surface is nice to have to keep those SMD parts from bouncing too far, but it's not too terribly important.
- Braided copper solder wick with resin for remove excess solder from SO8 components.

There are many good videos on YouTube showing different techniques:

https://www.youtube.com/watch?v=8Q6YNmBKjiU

https://www.youtube.com/watch?v=MiJOv3aRCYk

Hot Air Soldering

Hot air is used to melt solder paste under each end/pin of a device to make a joint.

The basic technique is to apply a small dab of solder paste to each pad of a device's footprint, place the device atop the dabs, and then to apply hot air to the pads until the past melts. The pads should be heated evenly until the paste melts. It's fun to watch this because surface tension will almost always position the part exactly where it is supposed to go with nice, clean joints. Hand soldered joints and alignment do not usually look as good.

This all sounds easy, and for the most part it is if you have the equipment. The tricky parts are applying the paste to the pads cleanly, and even and complete melting of the solder. Commercial processes use a stencil and a squeegee to apply paste to the pads consistently. As a hobbyist we usually squeeze the paste out of a tube or syringe and use toothpicks, craft knife blades, dental tools, etc. to shape the paste onto each pad.

Some tools you will need for hot air soldering SMD work:

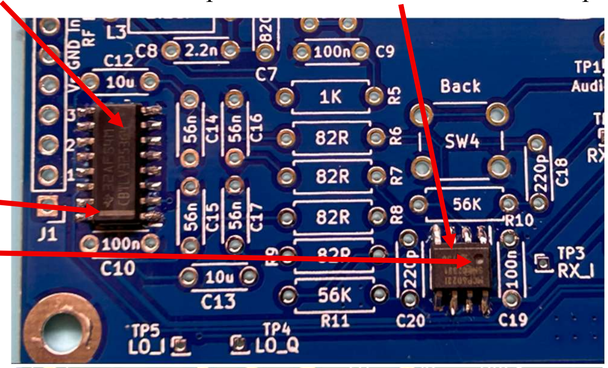
- Fine tip tweezers to pick up, hold, and flip SMD parts
- Magnifier of some type such as a headset or low power microscope or high magnification glasses
- Low temperature solder paste
- Hot air tool
- Creative mix of tools to apply/shape solder paste

Reflow Soldering

This is a commercial process where a stencil is used to squeegee solder paste onto a PCB, parts are picked/placed onto the PCB, and the boards are cycled through a thermal profile in a reflow oven. Many hobbyists have mastered this and used this for limited production. Stencils can be ordered along with PCBs for a small increase in cost and toaster ovens can be turned into reflow ovens with a bit of effort.

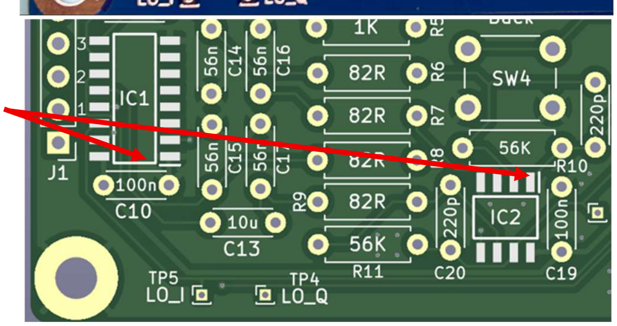
Hot plates designed for reflow soldering are also readily available and relatively inexpensive. These are a worthy investment if you plan to do a lot of SMD soldering, but not necessary to solder the two components required for this project. There are lots of YouTube videos that cover reflow soldering using a hotplate so check them out if interested.

The choice as to which technique to use will depend on the equipment you have available and your desire to expand your SMD soldering skills. This project can easily be completed by hand soldering the SMD's to the board. The following photograph shows the location of IC1 – SN74CBTLV3253DR chip and IC2 - MCP6022.   These chips were hand soldered to the PCB.



When placing the components on the board take note of the orientation of the chips on the footprint. Pin 1 is identified by a solid line on IC1 and is positioned at the bottom of the footprint.

Pin 1 on IC2 is identified by a sunken 'dot' on the IC and is positioned to the top right hand corner of the footprint.



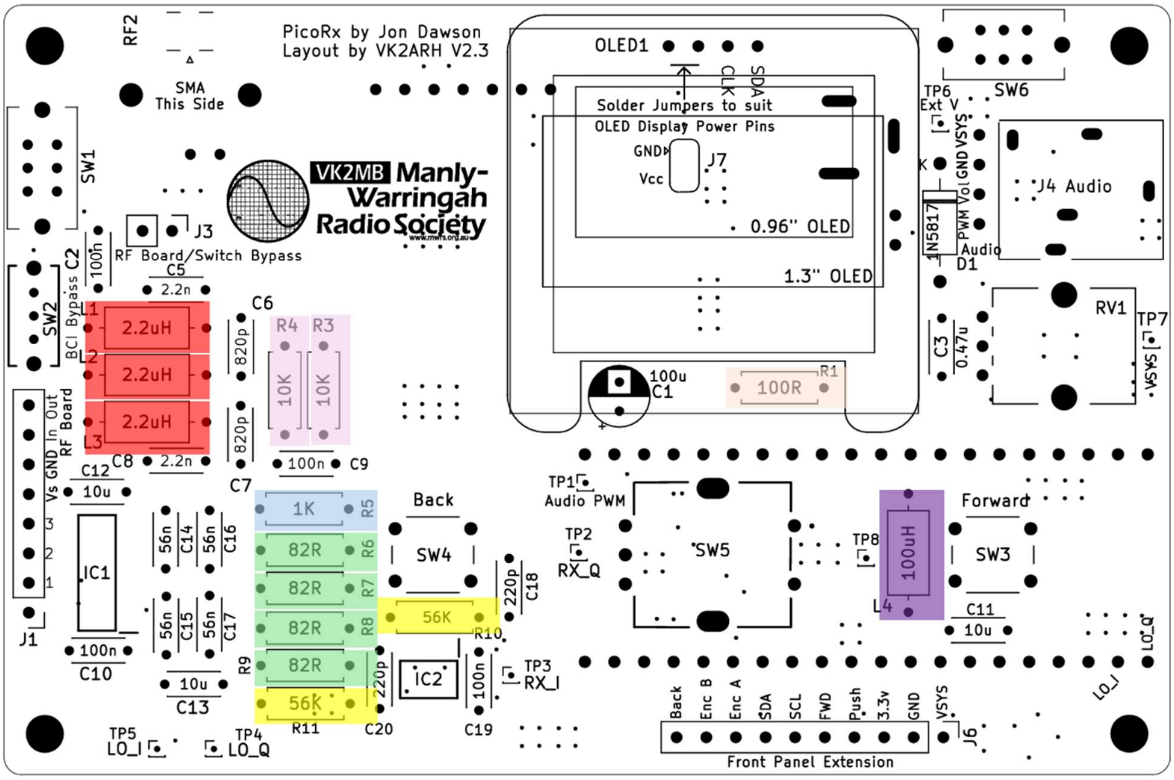
The pin 1 location is shown on the PCB silk screen for each IC by a solid line located next to the pin 1 as shown here.

- ☐ Solder the SMD IC1 and IC2 onto the main board

Step 2: Resistors and Inductors

☐ Start with the resistors – these are the smallest components to install. All resistors are 1/4w and should be mounted horizontally, flush with the PCB.

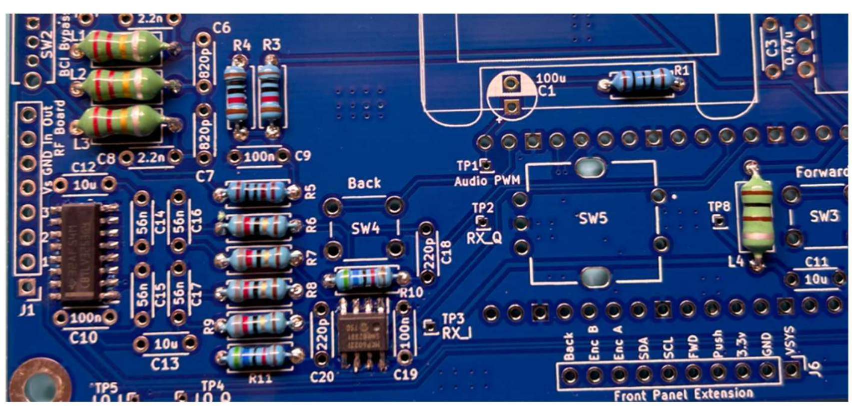
Colour	Reference	Value	Marking
	R1	100R	Brown-Black-Black-Black-Brown
	R3,R4	10K	Brown-Black-Black-Red-Brown
	R5	1K	Brown-Black-Black-Brown-Brown
	R6,R7,R8,R9	82R	Grey-Red-Black-Gold-Brown
	R10,R11	56K	Green-Blue-Black-Red-Brown



☐ Install the inductors. The inductors look like resistors but are larger in size and have a light green body.

Colour	Reference	Value	Marking
	L1,L2,L3	2.2uH	Red-Red-Gold-Gold
	L4	100uH	Brown-Black-Brown- Gold

Once installed the lower section of the PCB will look similar to this:



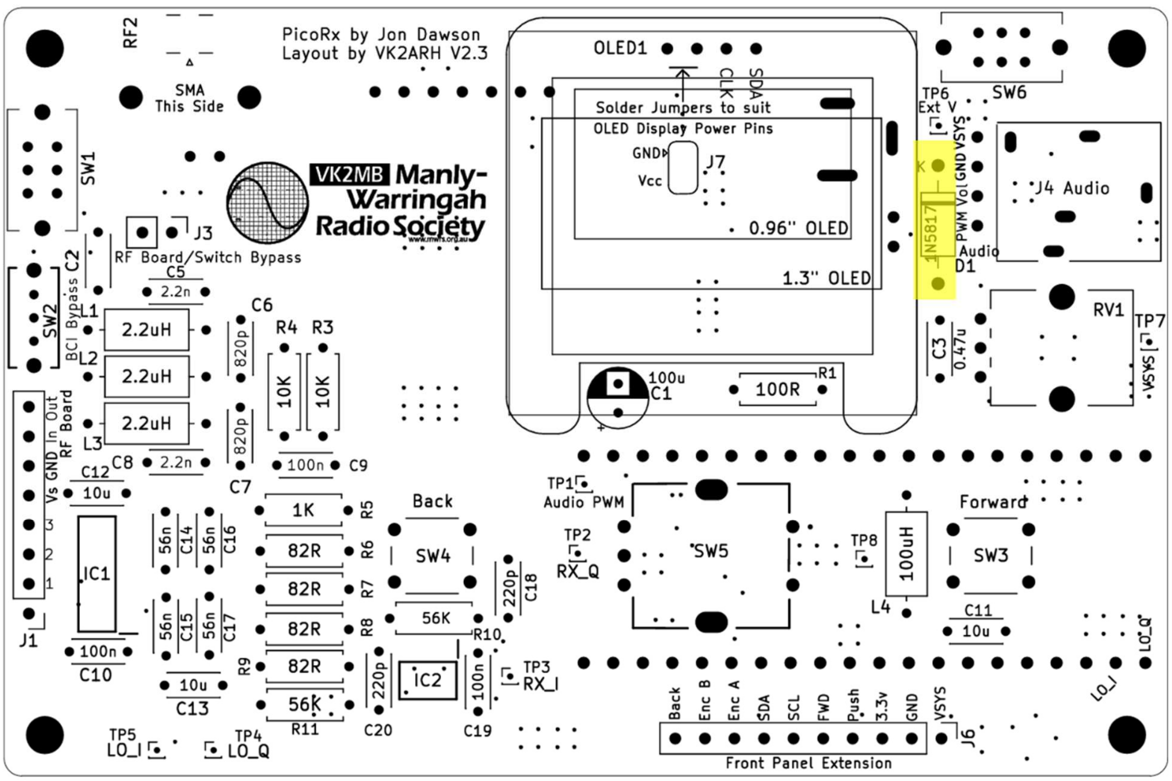
Step 3: Diode

☐ There is only a single diode used in this project:

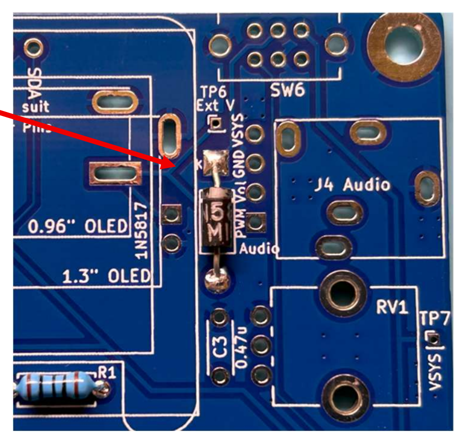
Colour	Reference	Value	Marking
	D1	1N5817	Black body with Silver Band at one End 1N5817 written on the diode

Install the diode with the correct polarization – the Cathode (the end with the silver band) is installed in the square footprint and this aligns with the screen print on the PCB and the letter K next to the footprint.

The location of the diode on the PCB is shown below:

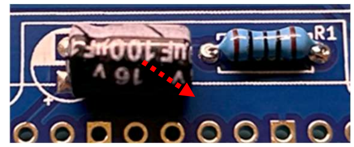


Once installed on the PCB the finished result should look like this:



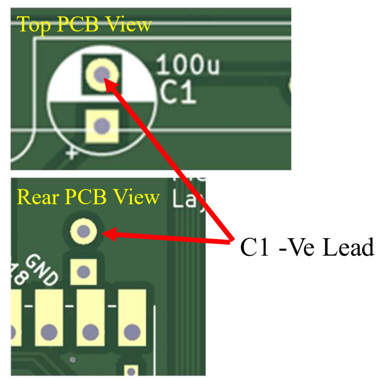
Step 4: Capacitors

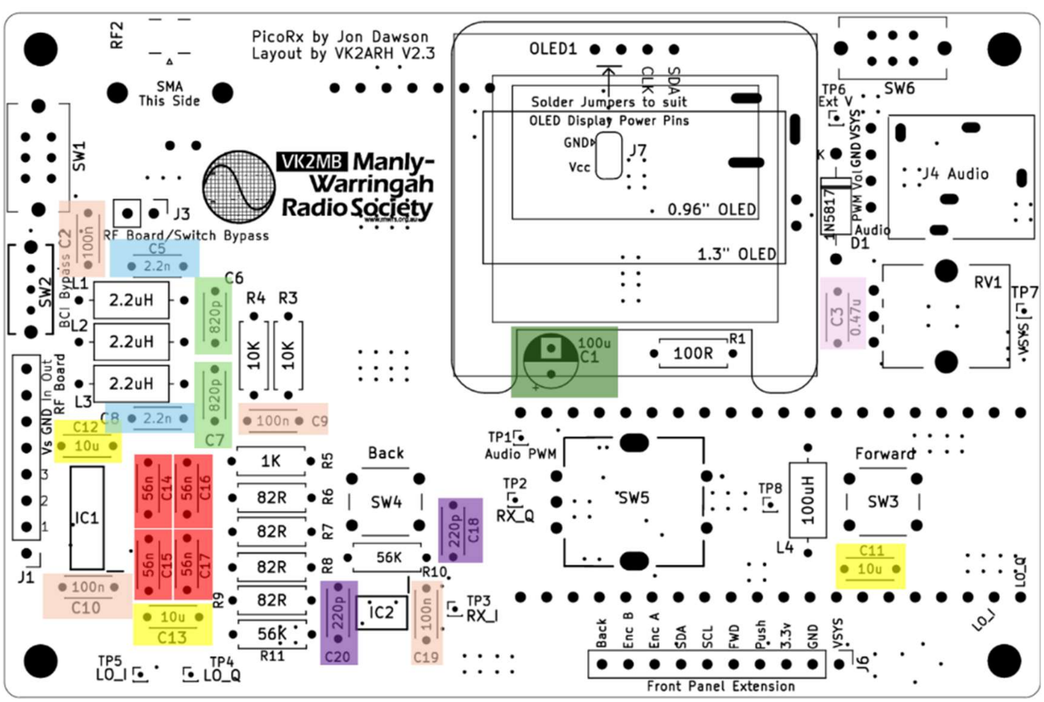
☐ Install the capacitors shown on the table below. The project uses an electrolytic capacitor and standard MLCC capacitors. The electrolytic capacitors are polarized and need to be installed in the correct orientation. The -Ve lead is the shorter of the two and the body of the capacitor indicates the -Ve lead. The MLCC's are not polarized and are identified with a numeric representation of their value. (See p34 for a capacitor identification chart).



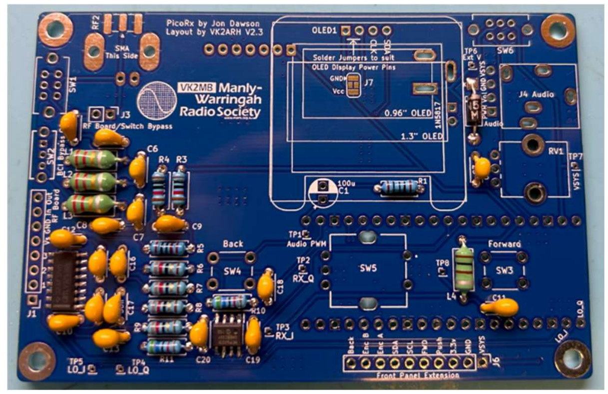
When installed on the board, -Ve (shorter lead) on C1 is installed facing the shaded side of the print screen symbol after bending the leads through 90° and **lying flat on the PCB** as shown in the top right-hand photo above. This orientation ensures the necessary clearance for the top panel, but you may need to leave longer legs on the capacitor to enable it to be maneuvered clear of any interference with the OLED display as indicated by the arrow in the above photo. **The electrolytic capacitors -Ve lead is soldered to the round solder pad.** You may choose to install the capacitor on the bottom side of the PCB to avoid any possibility of interference with the OLED display, but observe the correct polarization using the shape of the solder pads as your guide as shown on the lower right hand image. If you fit the capacitor on the rear of the board, a photograph indicating its location is shown on the following page.

Colour	Reference	Value (F)	Marking
(dark green)	C1	100u 16v	100uF 16v Electrolytic
(peach)	C2, C9, C10, C19	100n	104
(pink)	C3	0.47u	474
(light blue)	C5, C8	2.2n	222
(light green)	C6, C7	820p	821
(yellow)	C11, C12, C13	10u	106
(red)	C14, C15, C16, C17	56n	563
(purple)	C18, C20	220p	221

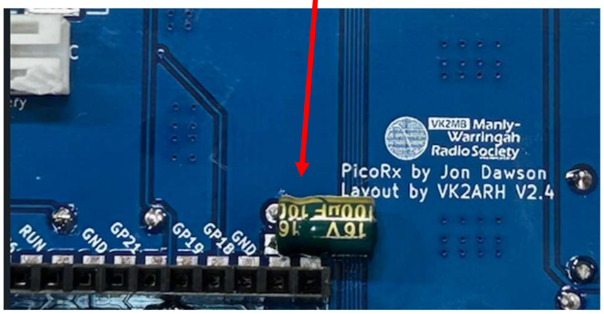




Once the capacitors are installed the board should look like this before installation of C1:

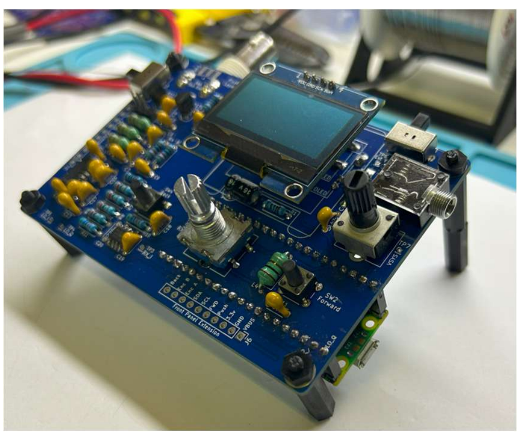


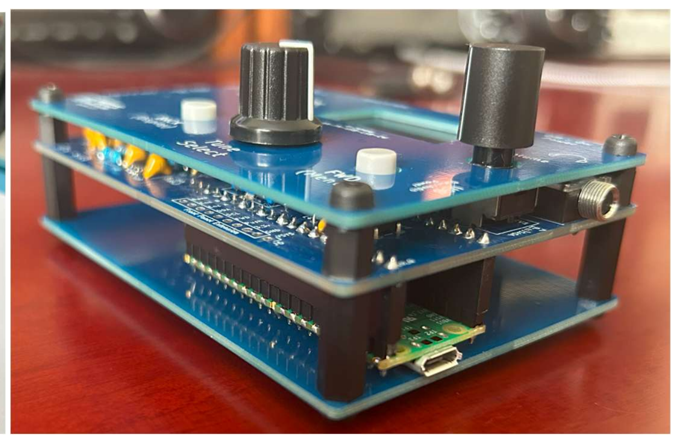
If you chose to install C1 on the rear side of the PCB to avoid interference with the OLED display (which may occur with some larger capacitors), the fitting on the rear will look similar to that shown below:



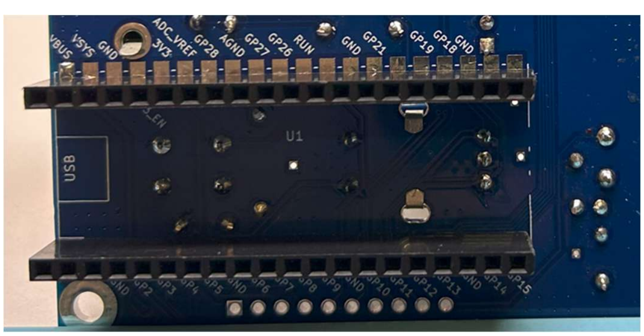
Step 5: Installing Pin Headers and Sockets for the Pi Pico

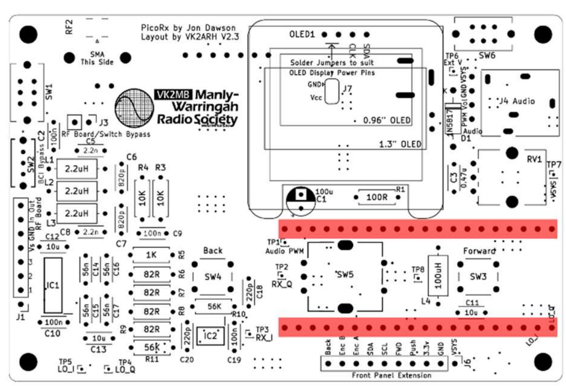
The Pi Pico is supplied without any header pins attached so we need to attach male header pins to the Pico and female header pins to the main PCB. The Pico is mounted on the underside of the board as shown in the photos below:





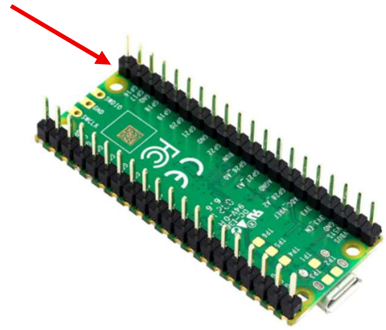
This will give you an idea as to the mounting location and orientation of the Pico. The female headers are positioned on the rear of the mainn PCB as shown in the photograph below.



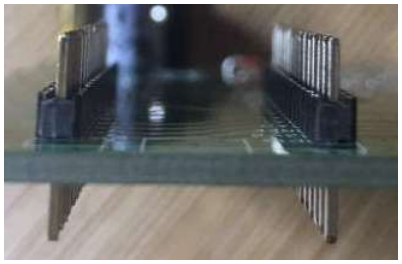


Installing the Male Pin Headers onto the Pico

☐ Carefully cut a single 40 pin male header strip in half with your wire cutter to form two 20 pin headers and then solder each 20-pin header onto the Pico with the orientation shown here:



Ensure that the pins are soldered perpendicular to the Pico board.



The best way to do this is to place the pin headers into the main PCB (long lead into the board) to ensure correct alignment as shown on the left. Then sit the Pico onto the short pin headers component side up and carefully solder the **Pico to the pin headers**.

Solder quickly and spread the heat load, soldering every second pin and then returning to pick up the unsoldered pins. The pico can now be lifted from the PCB and once turned over will look like the photo above, with the long header pins facing away from the underside of the Pico board.

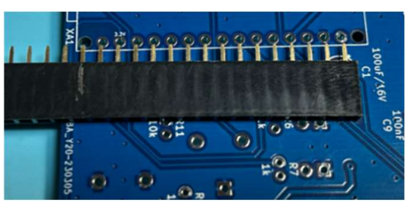
I strongly recommend that you **DO NOT solder the Pico pin headers directly to the main PCB** but install female pin headers on the main PCB to house the Pico, this gives you the flexibility of easily replacing the Pico should it be required.

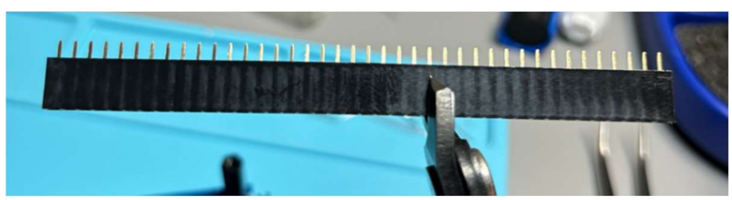
Preparing the Female Pin Header Strips (The illustrations shown below relate to preparing the header strips for the Arduino Nano used in the ADX project, but the technique and principles are the same when preparing them for the Pico's **20 Pin** header not 16 pin as shown in the illustrations). If your kit is supplied with 20 pin header strips you can skip this next step.

The MWRS Pi-PicoRx project was supplied with 20 Pin female headers so this section can be skipped and go straight to 'Installing the 20 Pin Headers at the top of p18.

If you don't have a 20 pin female header, a longer header strip can be cut down to form a 20 pin header. This technique is also used to prepare the 14 pin header used for connecting to the external TFT display on p33.

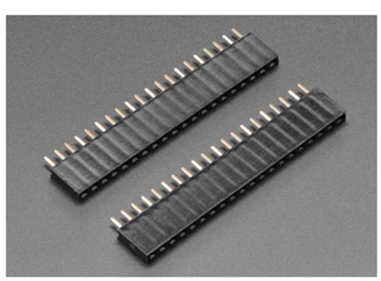
Align the strip up against the 20 holes for the Pico on the main PCB and then score the pin header where it is to be cut. Double count the number of pins to be left on the header before you make the cut. Take your wire cutters and cut the strip at the score mark. Hint: if you cut the 20 pin strip from one end of the original 40 pin header, you will have one nice clean end on the pin header which you can solder toward the edge of the PicoRx and this will look a lot neater.

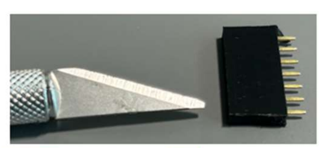


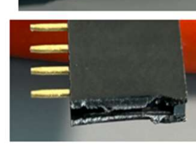


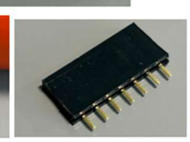
When finished your strip will end up looking like those shown below. The cutting process will destroy one of the headers and leave a ragged edge, but this is cleaned up easily with a modeling knife and/or sandpaper.

If using a standard 40 pin header strip, you will end up with a 20 pin and a 19 pin female header. As the Pico requires a second 20 pin header, you will need to use the second 40 pin strip and cut the 20-pin section from it. After sacrificing the 21st pin on each strip, you will be left with two 19 pin strips which can be repurposed on another project.



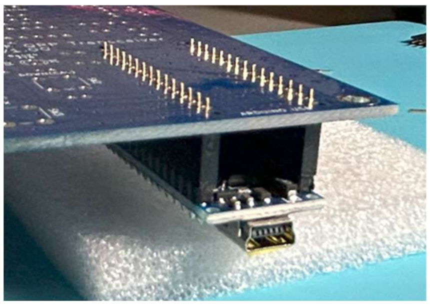


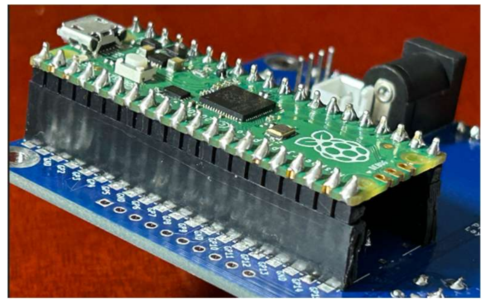




Installing the Pin Header

☐ Install your pin headers onto the rear side of the PicoRx PCB. Fit the pin headers and the Pico onto the board, turn the board upside down and place a piece of foam or similar underneath the Pico so that the weight of the board allows the pins to protrude through the solder side of the board. This ensures vertical alignment of the headers. Now solder the pin headers to the PCB. Act quickly as the heat from the soldering iron can easily melt the header. Consider spacing out the soldering by missing every second pin and then returning to solder the remaining pins and in doing so spread the thermal impact on the header pins. The finished results should look similar this even though the example photo on the left shows is an Arduino Nano – the same principle applies to the Pico:





I recommend removing the Pico at this stage and leaving it off the PCB until you have undertaken power on test and configured the software before inserting the Pico onto the main PCB.

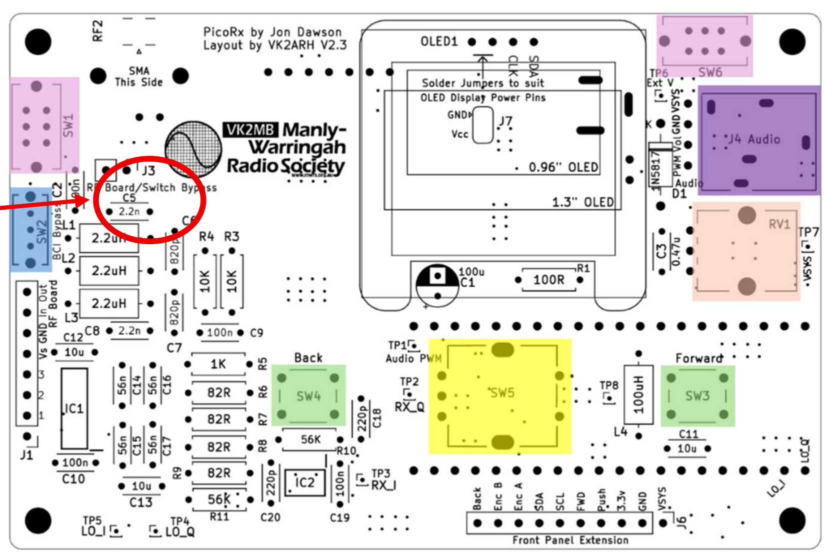
Step 6: Installing Switches, Sockets, Encoder and Variometer

☐ The final components can now be soldered to the main PCB. Start with the smallest and work up to the largest component.

Colour	Reference	Value	Comments
	RV1	10K Variable Resistor	10KB
	SW1, SW6	Micro SW_Slide_DPDT	Power and RF Board
	SW2	Micro SW_Slide_SPDT	BCI Filter
	SW3, SW4	6mm Micor Push Switch	Back and FWD buttons
	SW5	RotaryEncoder	PEC11R-4215K-S0024
	J4	AudioJack	

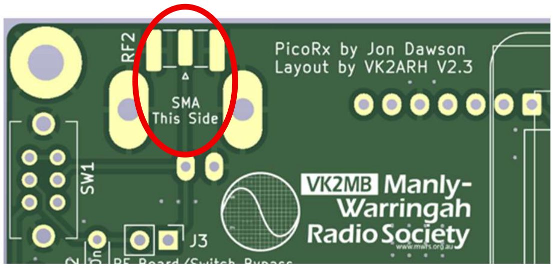
NOTE: If you plan to operate the PicoRx as a stand-alone unit with no external RF board it is not necessary to install SW1, instead simply solder a jumper over J3 to connect the RF input permanently to the main PCB.

You can always install SW1 and remove the jumper at a later stage if you wish to start experimenting with external RF components.

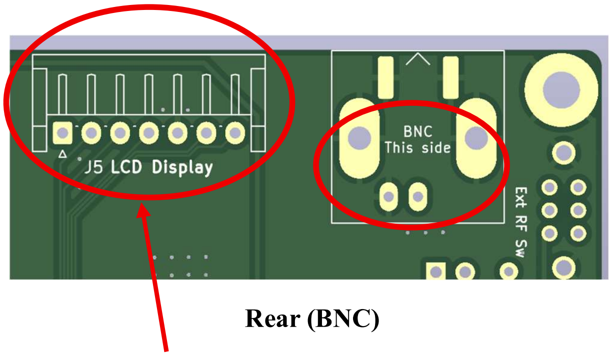


Step 7: Antenna Connector and TFT Display Sockets

☐ Install the Antenna connector. The PCB supports either a BNC **or** SMA antenna connector. If you use a BNC connector this is installed on the rear of the PCB and if you choose an SMA connector this is installed on the front of the main PCB. Both connectors **can't** be installed at the same time. The screen print on the PCB identifies which side to install the antenna connectors.



Front (SMA)



Rear (BNC)

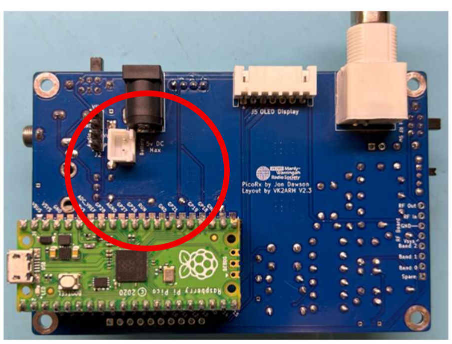
☐ Install the connector for the external TFT display. The optional colour TFT display is attached to the PicoRx using a 7 pin molex connector which is located next to the antenna connector **on the rear of the PCB**.

Step 8: Power Sockets

☐ Install the power sockets. You can either install the 5.5mm x 2.1mm DC socket **or** the USB-C power only socket. The 2 pin JST socket (J9) is an option to support an external battery or power supply connection.

☐ It is easier to install the 4 pin header (J2) at this time. J2's V_{in} pin can be used to supply power from an external power supply mounted on a daughter board if you wish to use that capability. Audio input and output are also available via J2. Power to external boards can be supplied through the VSYS, and GND connections on J1 and J6. J6 also has a 3.3v output pin. Most builders will initially install either the DC socket or the USB-C power socket.

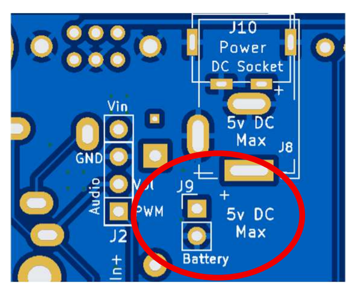
All power options are mounted on the rear of the main PCB. The finished board with the Pico installed will now look similar to that shown below. **NOTE: There is no onboard overvoltage protection, so caution should be exercised if using the 5.5mm power socket to ensure that you DO NOT supply more than 5v to the PicoRx. The use of the USB-C socket will avoid this issue and may be a safer option.**

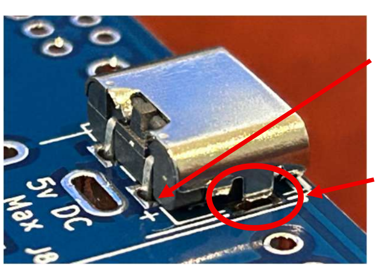


The 2 Pin JST connector is polarity sensitive. The location of the square +ve solder pad is marked on the PCB screen print and it's the pin closest to the DC power socket.

GND is the round solder pad, next to the 'Battery' screen printed text.

The 5.5mm x 2.1mm DC plug is center +ve



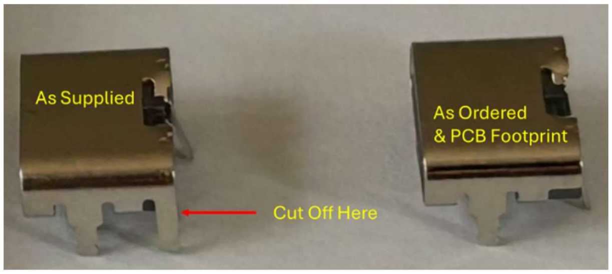


If you choose to use the USB-C power only socket, this is installed on the rear side of the PCB as shown on the left. This ensures correct polarity of the incoming power supply.

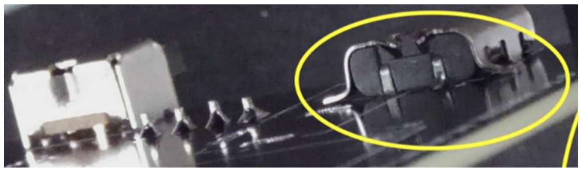
The casing leads are relatively short – you should wick molten solder down the mounting hole on each side of the casing to securely attach the USB-C socket to the PCB thereby maximize the structural support for the connector.

NOTE: Installing the USB-C Power Socket

The USB-C power sockets supplied for the kit were delivered with a slightly different footprint to the one ordered and designed onto the PCB – The supplied USB-C power socket has two power leads (+5v and GND) **plus four legs** attached to the metal shield of the socket, instead of two as per the PCB footprint. These legs are soldered to the PCB to secure the USB-C socket to the PCB. This is not a 'show stopper' … So read on😊.



Both variants are shown on the right. The easiest way to progress is to cut the 2 additional (rear) legs off the USB-C socket flush with the bottom of the socket. Or you can bend to rear legs out at 90^{o} so that they sit flush with the top of the PCB. Either way they will match the original footprint. Alternatively, you could drill additional small holes into the PCB at the required location, scrape away at the PCB to reveal enough of the PCB ground shield to enable you to solder the additional two legs to the PCB. If you do a good job soldering the two-legged connector to the board, this is not necessary.

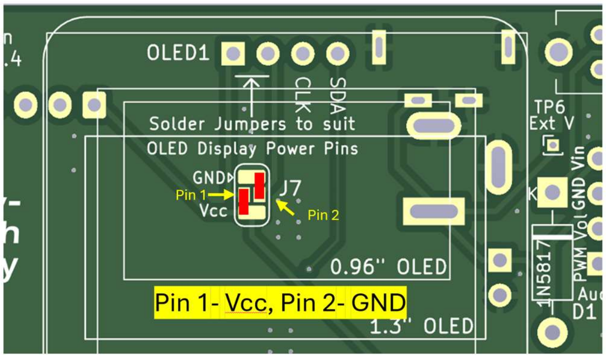


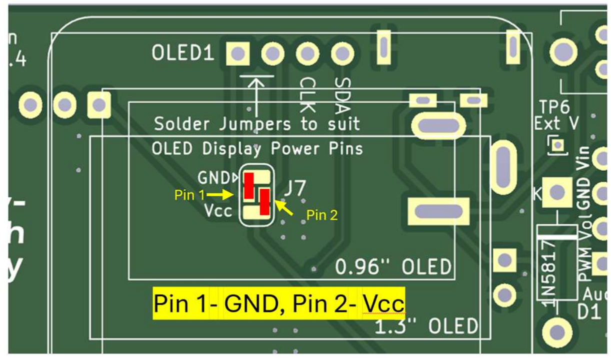
My recommendation - use pair of wire cutters and remove the additional two legs at the rear of the shield as shown in the photograph above right, or bend them out at right angles to sit flush with the PCB as per the photograph on the right (from another project but it illustrates the point).

Step 9: Installing the 1.3” OLED Display

☐ Before installing the OLED display, you should identify the VCC and GND pin location on your display and short out the appropriate pads on J7 with a solder bridge to match your display’s power pins.

Different manufacturers vary the location of these pins. J7 facilitates connection of OLED pins 1 and 2 to either Vcc or GND.

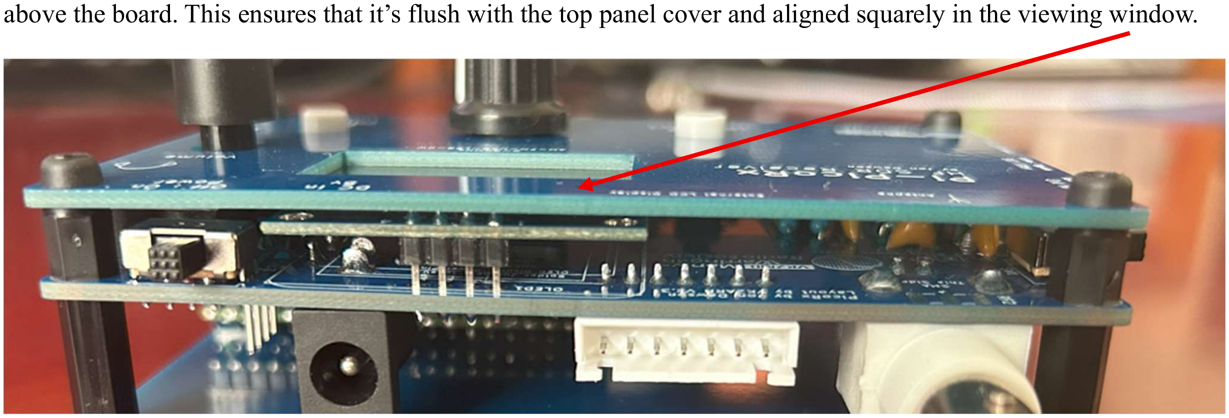




In the example below, the OLED display has pin 1 as Vcc and Pin 2 as GND – so the jumpers are soldered as shown below on the right.

☐ **Check your soldering with a multimeter to ensure that you haven’t shorted out the power supply and the required polarity is correctly aligned with the pin requirements on your OLED screen. Do these power checks before you install the OLED display.**

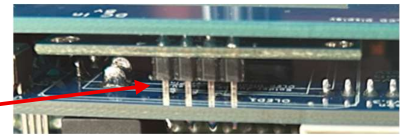
Care should be taken when soldering the OLED to the board to ensure that it is positioned squarely to and elevated above the board. This ensures that it’s flush with the top panel cover and aligned squarely in the viewing window.



☐ Install the OLED display top main PCB. I have found the easiest way to do this is:

- Solder the four-pin header to the OLED display with the longer pins on the underside and perpendicular to the OLED board.
- Temporarily attach the front panel using the 7mm spacer between the PCB and the front panel, having placed the OLED (with its four header pins installed) in but not yet soldered to the mounting holes on the PCB.

- Turn the PicoRX upside down, align and position the OLED so that it rests parallel against the underside of the top cover and aligned squarely in the display window. Solder only a single pin to the main PCB.
- Check and adjust the OLED position by melting the solder and reposition the OLED holding it in place until the solder sets.
- When you are happy that the OLED is in position, solder the remaining three header pins to the main PCB. The finished result will have the header pins soldered proud of the main PCB supporting the OLED as show:



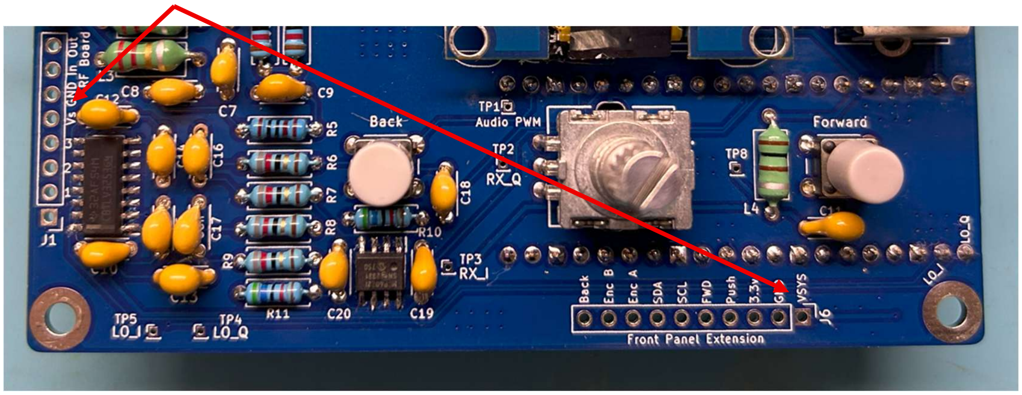
You may also like to trim the top of the pins if they are preventing the OLED fitting up against the PCB top cover window. If you experience any interference from C1 consider repositioning it, or relocating it to the rear of the board as discussed on p15-16 of this manual.

Congratulations 😉 you have now completed building the PicoRx's main PCB.

Step 10: Power Supply Testing

☐ **Before powering up the PCB remove the Pico processor from the board** (if you haven't already done so) and perform a continuity check on the DC inputs to ensure that you have no shorts.

This can be achieved by measuring the resistance between Vsys and Ground connection points on the external connectors as shown below:



As an extra precaution during the first power up, if you have access to a current limiting power supply, connect the PCB to a 5v source after setting the current limit to 50mA. The PicoRx will draw approx. 36mA with the standard 1.3" OLED display.

> *When you add the optional external 2.8" Colour TFT display (p31) the PicoRx will draw approx. 95mA. You will need to adjust the power supply current limit to approx. 120mA when you are ready to test the installation of the TFT display*

☐ Connect power to your PicoRx, switch the power switch to ON and check that you have approx.. 5v across the Vsys and GND measurement points you used earlier to check for shorts.

☐ Turn the power switch OFF and proceed to install the software onto your Pico controller (Step 11 below) **before** installing the Pico into the header pins on the PCB.

Step 11: Installing the Software

Loading the firmware into the Pico is achieved as follows:

☐ Download Jon's firmware from: https://github.com/dawsonjon/PicoRX/releases Choose the appropriate binary for either the original Pico or the Pico2 depending upon which Pico you are using. Both work fine but need to be installed on the correct Pico variant. The Pico2 with the more powerful processor and uses approx. half the computing power to operate the Pi-PicoRx, but the initial Pico version still works well.

NOTE: Firmware specific to the MWRS build which has Australian radio stations frequencies loaded into the firmware can be downloaded from here:

https://github.com/VK2ARH/Pi-PicoRx/tree/main/PicoRx%20VK2ARH%20Version

☐ Unzip the package which should contain two files as shown below (the example shows Jon Dawson's Pico2 version)

Name	Date modified	Type	Size
Today			
battery_check_pico2.uf2	5/27/2025 7:55 PM	UF2 File	58 KB
pico2rx.uf2	5/27/2025 7:55 PM	UF2 File	452 KB

☐ Connect a micro-USB cable from your computer to the Pico while holding in the pushbutton by the USB connector on the Pico and then release the button. The Pico should show up as a new USB drive on your computer .

☐ Copy the FW files you downloaded earlier onto the USB drive.

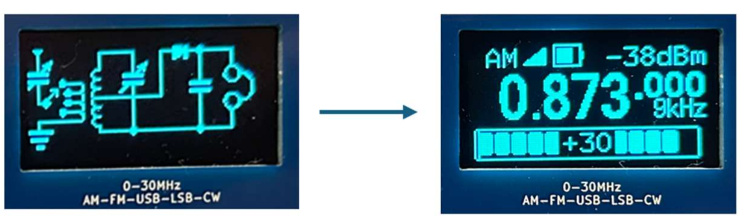
☐ Jon's original source is available from the same link shown above for those who wish to modify and develop the software.

Check Operation of the Pi- PicoRx

☐ Once the software is loaded, power supply testing completed and power turned off, install the Pico onto the main board, upside down on the back of the main PCB as shown here:

You may find handling and testing the PicoRx easier if you temporarily install the 4 x M3 x 15mm F-F spacers on the back of the PCB using the screws provided and then turn the PCB face up. This will give you a stable platform to configure and test your project before installing it into the housing.

☐ Power on the PicoRx using a 5v power supply (see p29 for discussion of power supply options). You should see the OLED come to life and display the 'Splash Screen' before displaying the standard display similar to that shown below:



☐ Press the BACK (Display) button several times and watch the display scroll through the various display options.

☐ Press the FWD (Menu) button and rotate through the various menu options to confirm that the rotary encoder is working.

☐ Press the rotary encoder to select a menu option and then press the BACK button to return to the main display.

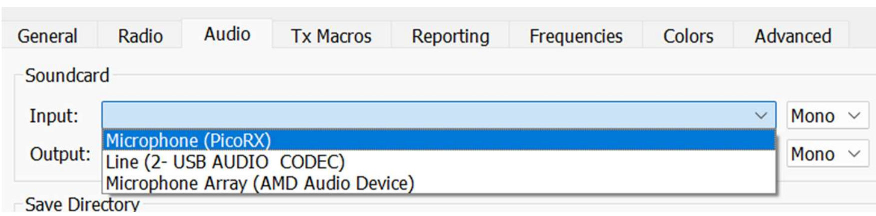
If successful you will have confirmed that the Pi-Pico has the software successfully loaded and is working correctly with the encoder, push button switches and the OLED screen.

If you do not see the correct operation of the Pi-PicoRx, check that you have successfully downloaded the software and go back to basics and start 'fault finding' your build. We are only checking the operation of the UI and the Pico at this stage.

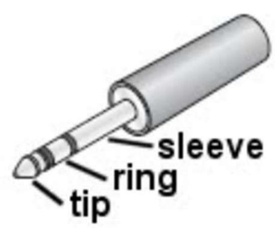
Check the correct installation of components and that they are soldered without solder bridges, dry joints or shorts. The board has been built and tested several times, so any problems with the operation of the board at this stage are likely unique to the individual build (or the components you used) and beyond the scope of this manual to diagnose. Good luck with your diagnostic journey.

Connecting the Audio Output

Audio is available via the Pico's USB port and this can be accessed by PC based software (eg: Flrig, WSJTx etc.) or you can use redirection software on your PC to route the incoming USB audio to your PC speakers. Incoming USB audio shows up on your computer as Microphone (PicoRx) as shown below:



The easiest way to hear the audio output is to connect earphones directly to your PicoRx using the 3.5m audio socket located just above the volume control, or you can use this same socket to plug into an external audio amplifier and speaker.



Whilst the socket is a stereo socket, the audio from the PicoRx is only a single channel and both left and right audio channels are connected to the one audio channel. The 3.5mm audio plug should be wired with Sleeve to ground. Ring and Tip carrying the audio output. *(This is the standard way of wiring and audio plug on earphones etc.).*

☐ Connect the audio output from the PicoRx to either a set of earphones or to an external power amplifier and speaker to hear the radio audio output.

Antenna Connection

The antenna connector (either BNC or SMA) supports a standard 50 Ohm antenna connection. The performance and operating experience of the PicoRx as with all radios will be antenna dependent. The use of external LNA's between the antenna and the PicoRx can aid in pulling out weak signals, but if you have a good antenna system (eg: your regular Amateur Radio HF antenna) an external LNA is not essential.

Be wary of overloading local AM radio reception with too strong a signal from your antenna system. This will result in a strong but distorted reception. In my experience I have found AM reception with a 40-60dB signal strength provides a good quality audio output, particularly if using an external speaker/amplifier.

Incoming RF signals can also be supplied via the External RF board connectors, but the use of external boards is not covered in this construction manual.

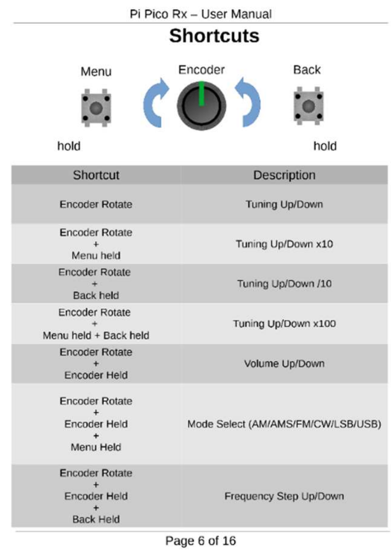
Initial Setup, Configuration, and Operating Tips

Accessing the Menu to Configure and Operate the PicoRx

A full description of the Pi-PicoRx's menu and setting structure can be found in Jon Dawson's PicoRx operation manual. This can be found in the user manual folder found here:

https://github.com/dawsonjon/PicoRX/tree/master

Jon's manual provides a lot of information about each of the options and configuration of the PicoRx, including diagrams to assist with navigating the PicoRx's menu structure. On Jon's original design and in the manual, he shows the Menu button being on the left hand side of the encoder and the Back button being on the right hand side of the encoder. As shown on the right:

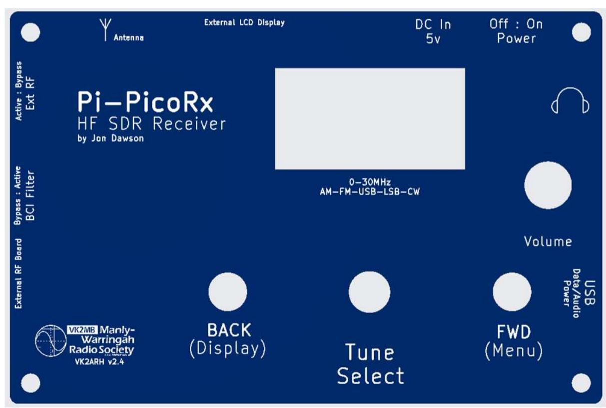

Pi Pico Rx – User Manual

Shortcuts

Shortcut	Description
Encoder Rotate	Tuning Up/Down
Encoder Rotate + Menu held	Tuning Up/Down x10
Encoder Rotate + Back held	Tuning Up/Down /10
Encoder Rotate + Menu held + Back held	Tuning Up/Down x100
Encoder Rotate + Encoder Held	Volume Up/Down
Encoder Rotate + Encoder Held + Menu Held	Mode Select (AM/AMS/FM/CW/LSB/USB)
Encoder Rotate + Encoder Held + Back Held	Frequency Step Up/Down

Page 6 of 16

However, in this project the position of the Back and Menu button are reversed – the **Menu button is on the Right and the Back button is on the left**. Being right-handed I found this orientation more intuitive. (Sorry to the 'lefties' out there).



The basic operation is described below:

- The FWD (Menu Button) activates the Menu Structure – you rotate the encoder to scroll through the various menu options, press the encoder button to select the option you wish to configure/change. Once an option is chosen, press the encoder button to select and store the option in firmware and return you to the operating display.
- The BACK (Display button) will cycle through the various display options available on the OLED display during normal operation. Pressing the BACK button at any time whilst in Menu mode will return you to the operating display without making any changes.
- The Tune/Select rotary encoder allows you to change frequencies (up and down) by rotating the encoder during normal operation. The frequency will change by the frequency step you have selected.
- Pressing the encoder during normal operation will activate the Memory Recall function. Rotating the encoder will then scroll though the available memories that have been configured in ascending or descending order. Pressing the encoder button when displaying a memory will pull that memory setting currently displayed into operation. Return to normal operation by hitting the BACK button.

I recommend that you download the Pi PicoRx User Manual and familiarize yourself with the capabilities and operation of the PicoRx. Jon and the team of developers have done a great job delivering an amazing range of functionality for you to use and enjoy. Learn to take advantage of it and master the use of your PicoRx.

External RF Board Bypass

If you are not using an external RF board to deliver additional capability or experimentation, the Ext. RF switch should be switched to the **Bypass** position for normal operation. The option to permanently bypass this feature by installing a jumper (which can be removed at a later stage if you wish to use an external RF board) is discussed on P18 of this manual. Switching the Ext. RF switch to **Active**, routes the incoming RF from the antenna connector to the external RF board and connects the incomming RF from the external board to the PicoRx.

BCI Filter Bypass Switch

This implementation of the Pi-PicoRx has a very effective Broadcast Interference (BCI) Filter on the main PCB. This is particularly useful if you wish to suppress interference from strong local AM radio broadcast stations when listening to other areas of the HF spectrum.

For normal operation you can choose to either activate or bypass the BCI filter depending on any level of interference, HOWEVER when listening to local AM radio stations, the BCI filter is set to Bypass otherwise the BCI filter will suppress the AM radio signal.

Setting the Volume

The volume control on the Pi-PicoRx adjusts the volume from 0 to the maximum volume which is set in the menu options:

Menu > Volume > {0 through 9} : 0 = Mute, 9 = Max

The dedicated volume control makes for easier operation of the Pi-PicoRx without reverting to menus or multiple function button/encoder input. In addition to using the menu, the maximum volume can be adjusted by rotating the encoder with the encoder button pressed in. See the Pi-PicoRx user manual p6 for operational shortcuts.

If you are not hearing any sound from your PicoRx – check that the menu volume setting is not set to 0. Most users will set the volume to 9 and adjust the volume using the dedicated volume control on the front panel.

OLED Driver and Orientation of the External TFT Displays

OLED – 1.3" Driver

The original PicoRx used a 0.96" screen, however this project uses the larger 1.3" screen which provides an improved user experience. You need to change the display driver from the standard OLED SSD1306 to the SH1106 driver. Changing this driver ensures that the display is moved two pixels to the right and correctly aligned the required display in the center of the screen and does not 'chop off' the left hand side of the display as shown below:

→

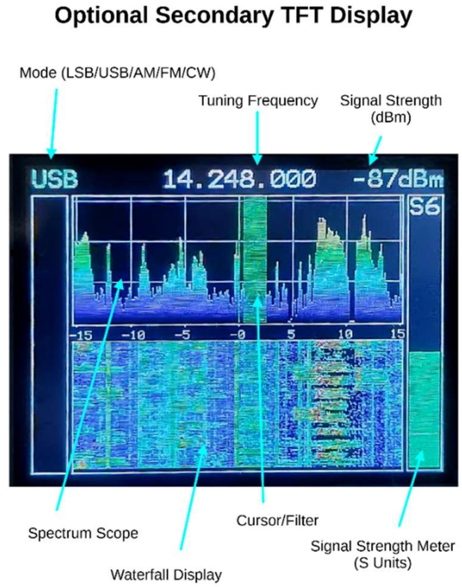
SSD1306 Driver SH1106 Driver

The settings for the OLED display can be found in:

Menu > HW Config > OLED Type > {SH1106 or SSD1306} SH1106 for the 1.3" OLED

External TFT Display (Optional)

The addition of an external TFT display enables the spectrum scope functionality to be displayed on a larger colour display and adds another dimension to the PicoRX. Connections for the display are discussed at p32 of this manual.



There are several display orientations available in firmware to match the orientation of your external TFT display, and you will need to activate the LCD display in firmware. You should connect the TFT display before you power up the PicoRx.

By default the TFT display is set to OFF – to activate the TFT display go into the Menu with the TFT display connected and rotate through the TFT settings until you see the desired orientation appropriate for your TFT display.

Menu > HW Config > TFT Settings > {OFF; Rotation 1 – 8}

You also have two colour options for your TFT display either RGB or BGR (you'll see what impact it has on the waterfall when its operating) – choose RGB or BGR to suite your own preference. (BGR is the traditional colour scheme used in most waterfall displays).

Menu > HW Config > TFT Colour > {RGB or BGR}

Squelch Settings

If you are not hearing any sound, it may be because you have an incorrect squelch setting. Adjust the squelch setting to ensure that the squelch is not suppressing the audio output, to do this go to:

Menu > Squelch > {S0 – S9+30dB} : Start with S0 and adjust from there if squelch is needed

AGC Speed

There is a range of ACG speeds which can impact the reception experience, and these should be adjusted to suit your requirements:

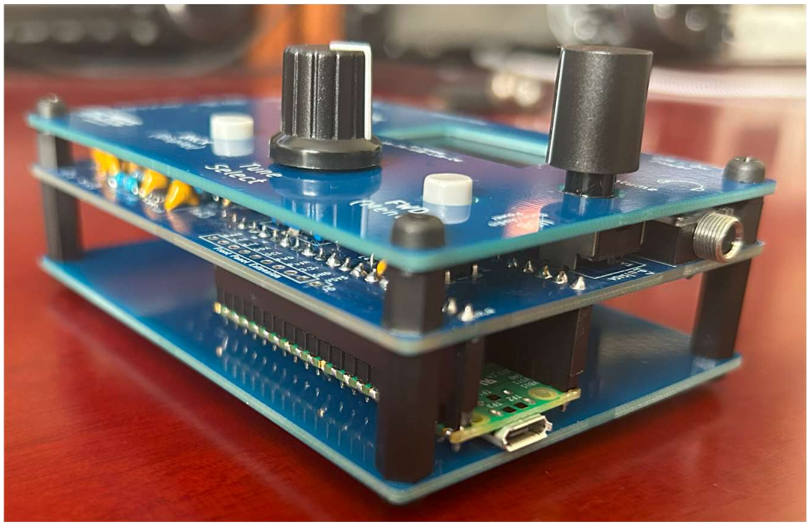
Menu > ACG Speed > {Very slow, Slow, Normal, Fast, 60dB – 0db(in 6dB increments)}

Attenuation of Strong Local AM Stations

The quality of local AM station reception can be impacted significantly if the signal is too weak or by a strong signal which is overloading the receiver. I have found that my EFHW antenna overloads the AM receiver when listening to local broadcast stations and I need to use either external attenuation or switch to a less efficient antenna. Signal strength is the range of 40-60dB seems to provide a good quality audio output from the PicoRx for local AM stations.

Housing the Pi-PicoRx

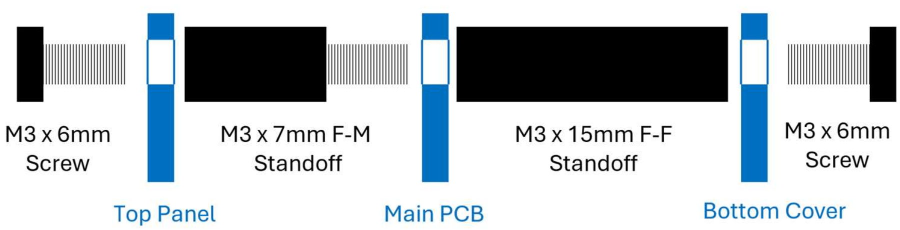
Assembling the Pi-PicoRx into the 'PC Sandwich' housing result in practical and attractive housing for the project. The finished project looks something like that shown below.



The housing is put together using the following:

- Top Cover
- Main PCB
- Bottom Cover
- 8 x M3 6mm Nylon (or Stainless Steel) Machine Screws
- 4 x M3 7mm M-F Nylon Insulated Standoff's
- 4 x M3 15mm F-F Nylon Insulated Standoff's
- 2 x Minature Push Button Caps
- 2 x Knobs (for the Rotary Encoder and the Volume Control).
- 4 x Silicon Rubber Anti Slip Bumper Feet (Optional)

The housing is assembled on each corner using the component side view as shown below. Start by installing the Main PCB between the two standoffs, then add the Top Panel and Bottom Cover securing them with the screws.



Push button caps, encoder and volume control knobs are then added. The orientation of the encoder knob is not important, but you should rotate the volume control fully anticlockwise and push the knob down on to the volume control shaft with the position marker in approx. 7:00 position as shown on the right:

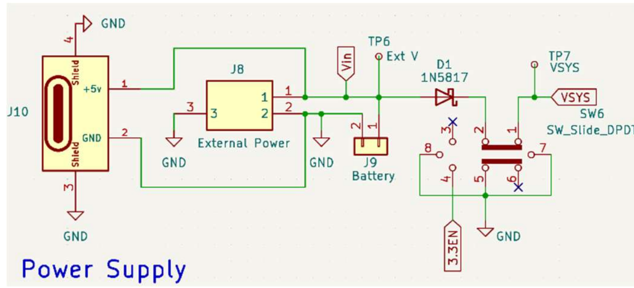
If you wish to install Silicon Rubber Anti Slip feet, these can be stuck on the bottom panel after assembly. These prevent the Pi-PicoRx sliding around on a smooth surface during operation.

Additional Technical and Optimization Information

Power Supply Options

There are several choices to power the PicoRx.

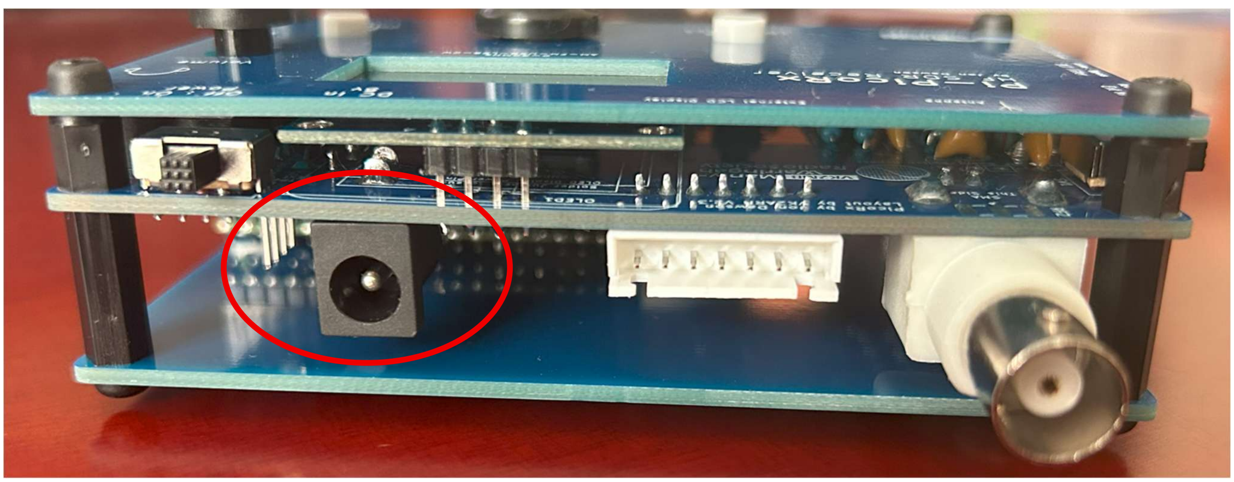
- External 5v DC Supply
- Pico USB Port
- Battery (3.7 - 4.5v)
- Battery power management system feeding into the battery connection or V_{in} pin on J2.



External 5v Supply via DC Plug

The 5.5m x 2.1mm DC power plug is **center +ve** and is located near the power switch as shown in the photo below. This power feeds V_{SYS} on the Pico via the On/Off switch and is reverse polarity protected using a 1N5817 diode.

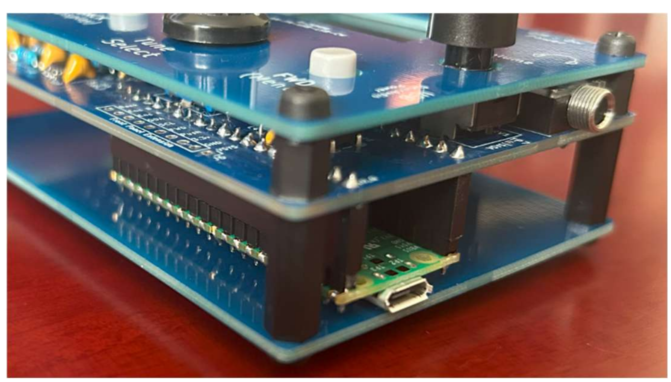
Do NOT connect more than 5v DC to this socket otherwise you will damage the Pi-Pico and possibly the PCB tracks if too much current draw results. To avoid the potential overvoltage supply, you have the option of replacing the 5.5mm DC plug with a USB-C Power only connector. The USB-C connector has been deliberately located in the same position as the 5.5mm DC socket, to ensure that you use one or the other but not both sockets.



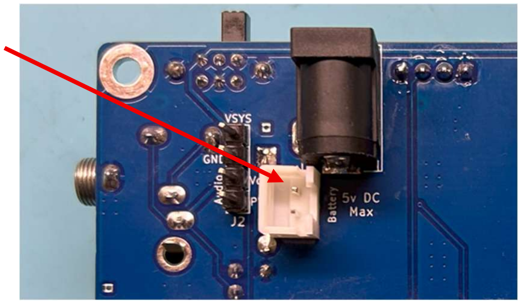
Power using the Pico Micro USB Port

The PicoRx can be powered directly from the USB port on the Pi Pico, with no need for any additional power supply connection. This is a convenient way of powering the device if you are using a USB connection for CAT control or USB audio.

Take care to ensure that any external power passed to external boards or components does not exceed the maximum current capacity of 300mA total for the Pico's onboard 3.3v regulator.



Battery Power Connector



The battery power connector on the bottom of the main PCB is designed to accept a 2 pin JST connection to an external battery and the +_ve pin is indicated by the arrow.

The PicoRx can operate from a 3.7v Lithium Iron battery or from 3 x 1.5v AA or AAA batteries.

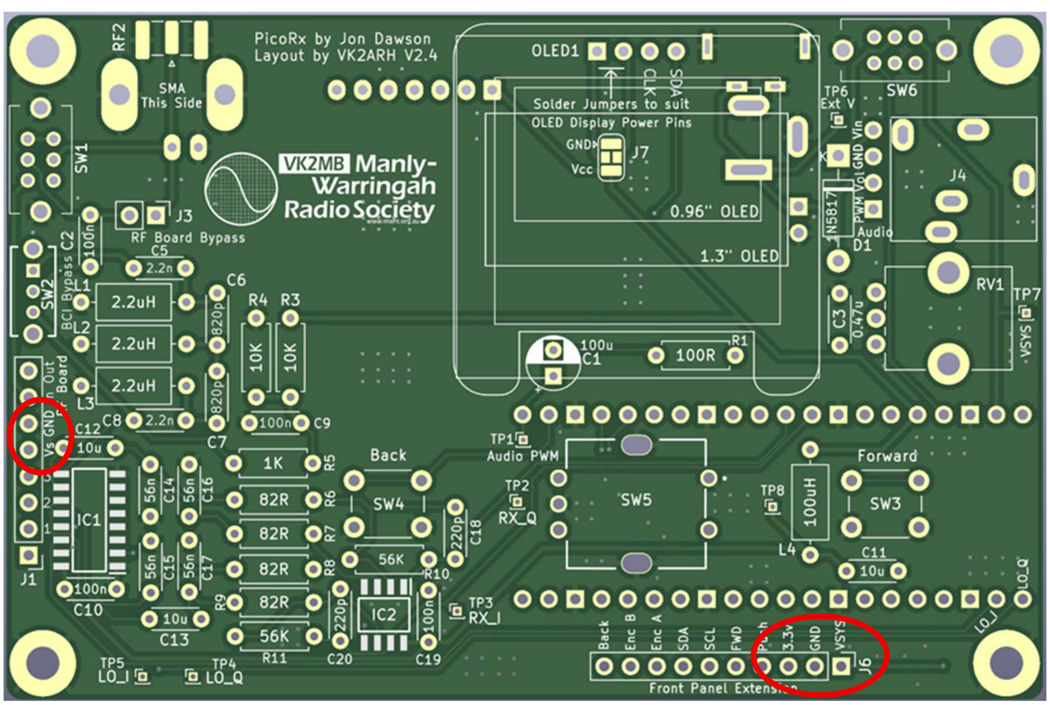
Power should not be connected to the DC plug if you are using the battery connector unless you have installed a protection diode (eg: 1N5817) between the power source (battery) and the battery connector:

Battery Power Management System

There are a number of Lithium-Ion battery power management modules available which can be used externally to the PicoRX main board to supply power and if desired charge a Lithium Ion battery connected to the J9 battery connector. These external power sources can be connected to the PicoRx using the JST battery connector or the V_{in} pin on J2. As the design and operation of these modules vary, the individual builder should assume responsibility for the safe operation and connection of these modules and batteries to the PicoRx after reviewing the circuit schematic diagram on p10.

Power Supply to External Boards and Equipment

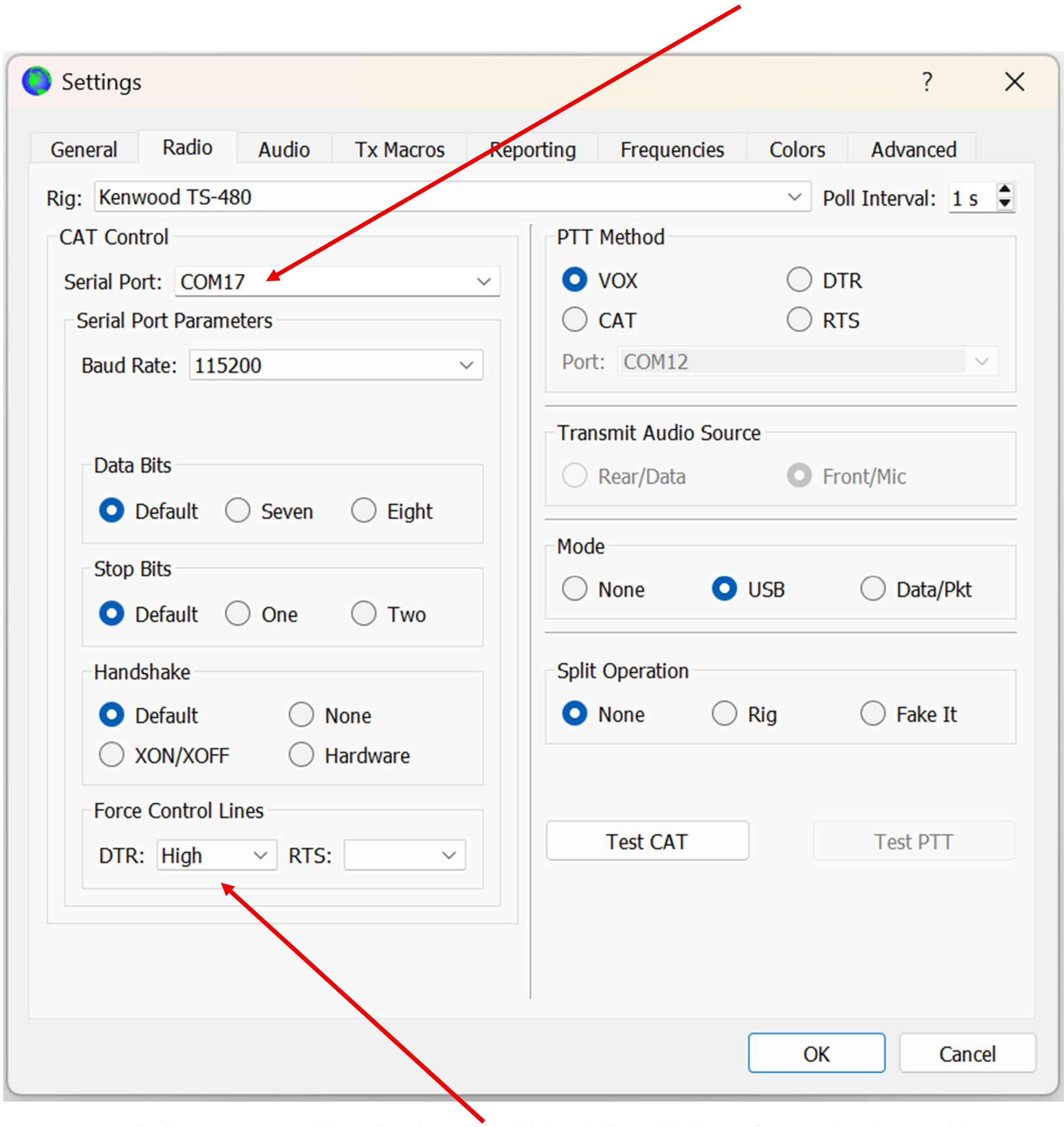
The board has been designed to provide experimenters with choices to power external boards or components. V_{SYS} and GND pins are available at the connectors for the RF board (J1), and the Front Panel Extension Connectors (J6) which also contains a 3.3v output. Do not exceed 300mA total current draw on the 3.3v supply.



CAT Control

Cat control is provided through a USB serial port interface. The PicoRx emulates a subset of the Kenwood TS-480 protocol. The CAT interface allows the receiver to be controlled via a host device by software such as grig, wsjtx and fldigi.

The settings for WSJTx are shown below (you will need to select your own serial port connection)

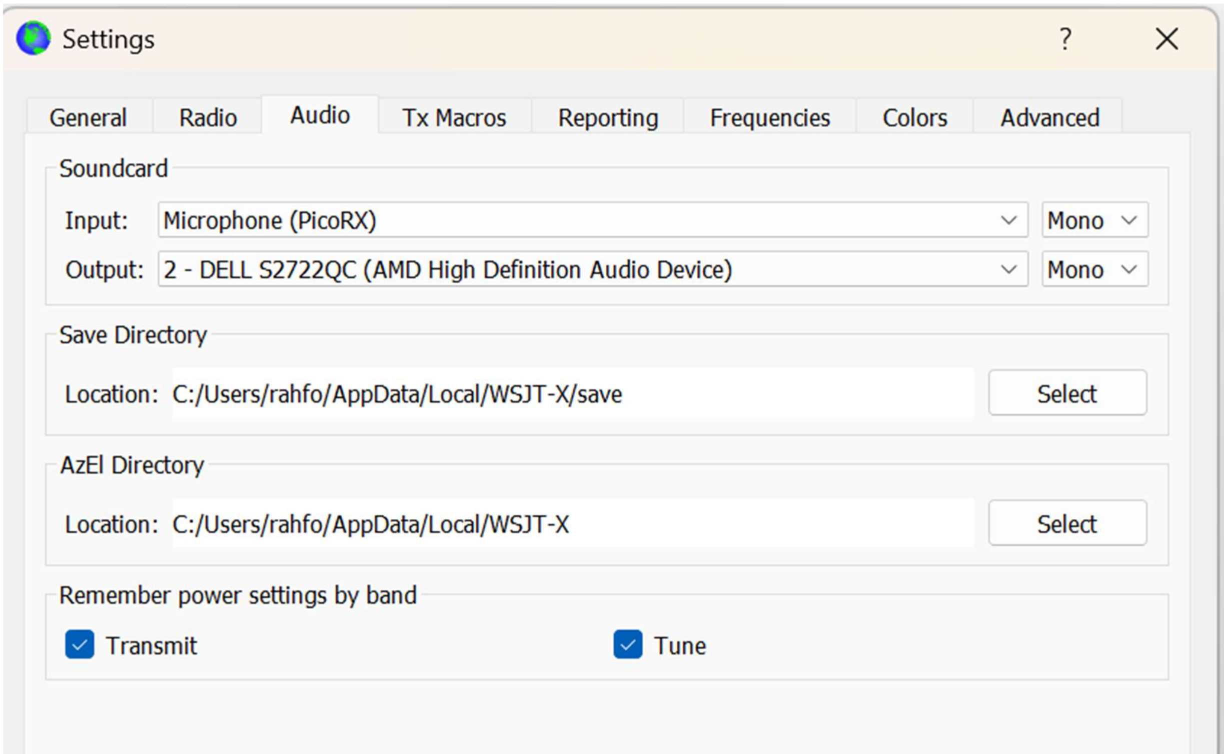


NOTE: For Windows PC's you will need to force DTR high, I believe this is not the case for Linux and iOS.

WSJT-x Audio Settings

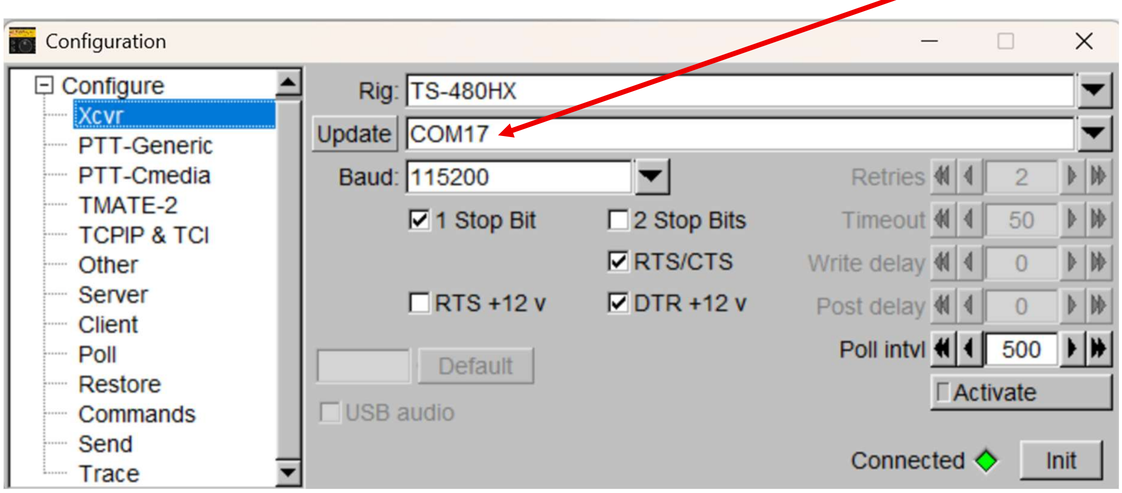
The Pi Pico Rx supports USB audio, and when connected should appear as a USB microphone. This allows a host device to easily make audio recordings (e.g. using audacity), and is compatible with software such as wsjtx, fldigi and QSSTV. When combined with USB cat control allows a fully functional PC connection using only a single USB cable. The direct digital audio connection provides superior audio quality compared to an analogue connection using a sound card.

The following are typical settings for using the PicoRx as a receiver with WSJT-x



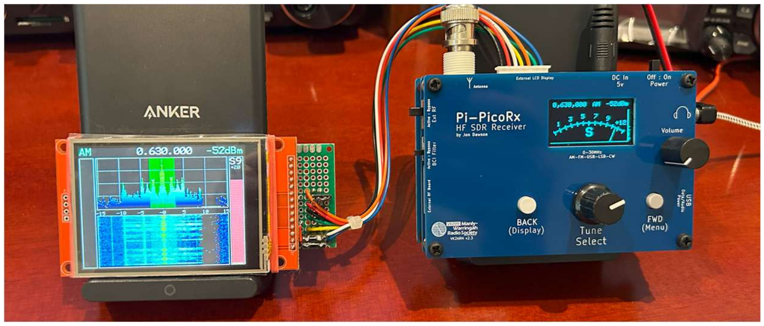
FLRig Settings

The following settings are required for connection to FLRig. (Choose your own assigned COM Port).

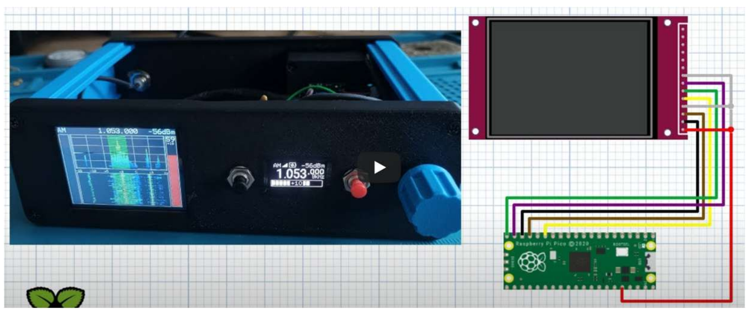


Connecting an External Colour TFT Display

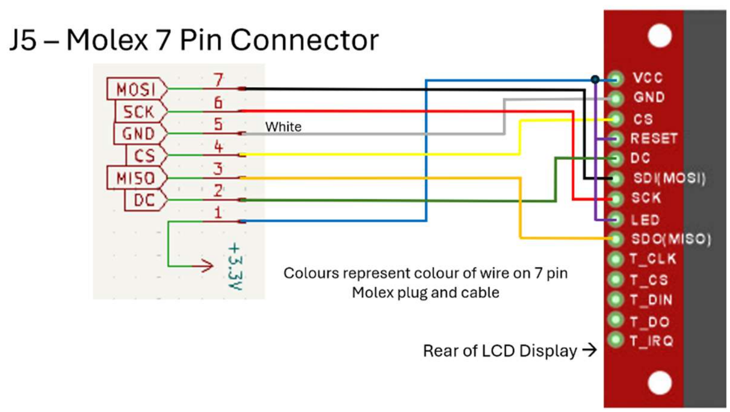
Whilst not essential for the operation of the PicoRx, the addition of an external colour waterfall display makes an interesting addition to the project.



A range of mounting options are available to suit your own requirements and an example is shown below. The provision of the external front panel extension connector (J6) makes mounting in external enclosures a little easier. Here is an example of the TFT display and other UI controls mounted in an enclosure from Jon Dawson's website.



The PicoRx mainboard contains a 7 pin Molex connector. The connections to the TFT screen using the connections shown on the right.

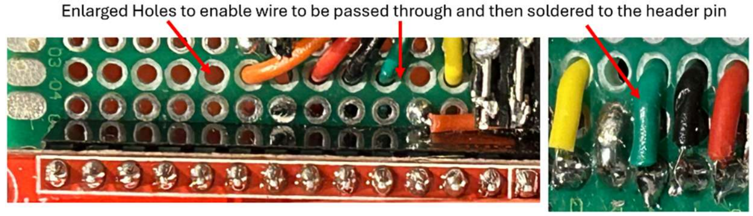


The TFT and PicoRx will draw approximately 95mA during normal operation.

The wires on the Molex 7 pin connector are quite fragile if soldered directly to the TFT screen and with movement are likely to break easily at the solder joint.

The kit is supplied with a section of prototype PCB on which to solder a 14 pin female header strip which you cut from the 40 pin strip. The TFT display is plugged into this header strip.

Enlarged Holes to enable wire to be passed through and then soldered to the header pin



Enlarge the line of holes (using a drill bit) two rows away from the header strip and insert the wires through these holes, bend them and solder to the bottom of the header strip. This provides a degree of mechanical protection for the solder joint and reduces the possibility of the solder connection breaking.

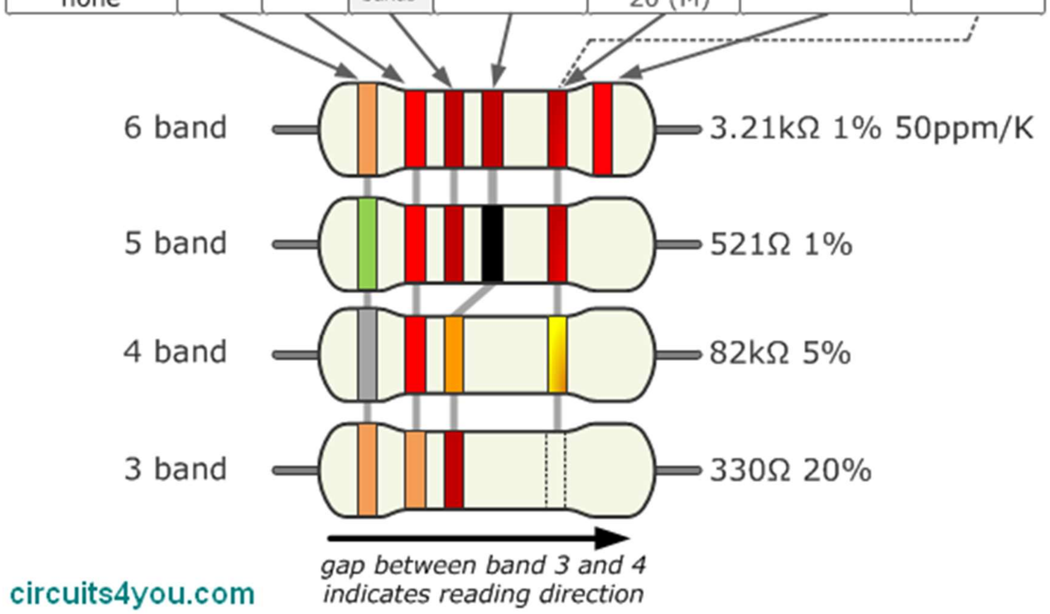
These photos should give you the idea, although on my own board I have soldered some jumpers and headers to the prototype PCB to enable switching of some of the lines. This is not required for the PicoRx Project.

(I was using them to experiment with whist developing the project, and I was too lazy to build another to photograph for the manual 😊)

Resistor, Capacitor and Inductor Identification Charts

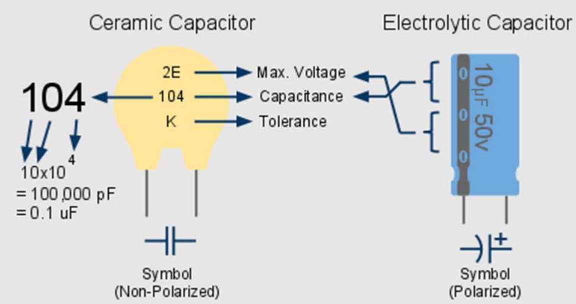
Resistor Color Code Chart

Color	Signficant figures			Multiply	Tolerance (%)	Temp. Coeff. (ppm/K)	Fail Rate (%)
black	0	0	0	x 1		250 (U)	
brown	1	1	1	x 10	1 (F)	100 (S)	1
red	2	2	2	x 100	2 (G)	50 (R)	0.1
orange	3	3	3	x 1K		15 (P)	0.01
yellow	4	4	4	x 10K		25 (Q)	0.001
green	5	5	5	x 100K	0.5 (D)	20 (Z)	
blue	6	6	6	x 1M	0.25 (C)	10 (Z)	
violet	7	7	7	x 10M	0.1 (B)	5 (M)	
grey	8	8	8	x 100M	0.05 (A)	1(K)	
white	9	9	9	x 1G			
gold			3th digit only for 5 and 6 bands	x 0.1	5 (J)		
silver				x 0.01	10 (K)		
none					20 (M)		



Capacitor Identification Charts

Capacitors



Max. Operating Voltage

Code	Max. Voltage
1H	50V
2A	100V
2T	150V
2D	200V
2E	250V
2G	400V
2J	630V

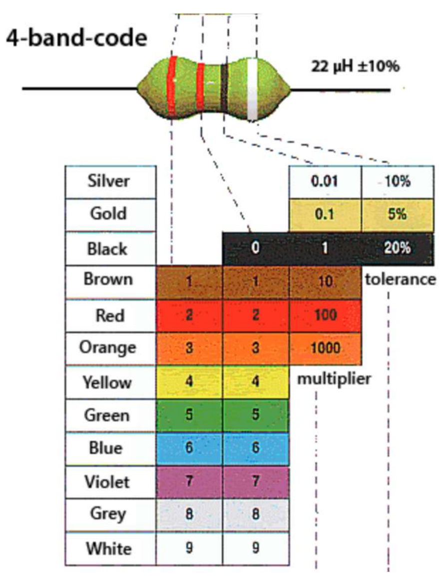
Capacitance Conversion Values

Microfarads (µF)		Nanofarads (nF)		Picofarads (pF)
0.000001 µF	↔	0.001 nF	↔	1 pF
0.00001 µF	↔	0.01 nF	↔	10 pF
0.0001 µF	↔	0.1 nF	↔	100 pF
0.001 µF	↔	1 nF	↔	1,000 pF
0.01 µF	↔	10 nf	↔	10,000 pF
0.1 µF	↔	100 nF	↔	100,000 pF
1 µF	↔	1,000 nF	↔	1,000,000 pF
10 µF	↔	10,000 nF	↔	10,000,000 pF
100 µF	↔	100,000 nF	↔	100,000,000 pF

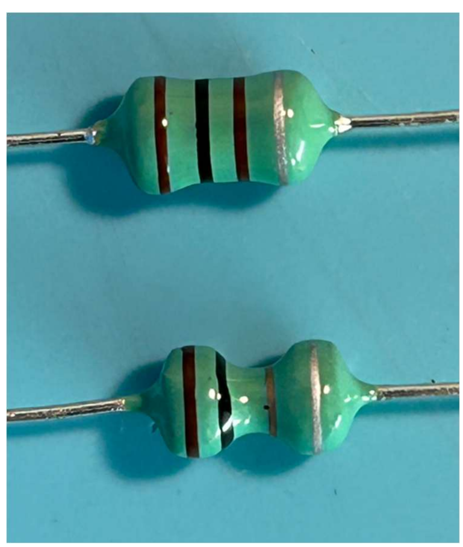
Tolerance

Code	Percentage
B	± 0.1 pF
C	±0.25 pF
D	±0.5 pF
F	±1%
G	±2%
H	±3%
J	±5%
K	±10%
M	±20%
Z	+80%, -20%

Inductor Identification Chart



Silver			0.01	10%
Gold			0.1	5%
Black		0	1	20%
Brown	1	1	10	
Red	2	2	100	
Orange	3	3	1000	
Yellow	4	4		
Green	5	5		
Blue	6	6		
Violet	7	7		
Grey	8	8		
White	9	9		



Brown: Black: Brown: Silver
1: 0: x 10 (multiplier)
= **100uH** 10% tolerance

Brown: Black: Gold: Silver

1: 0: x 0.1 (multiplier)
= **1 uH** 10% tolerance