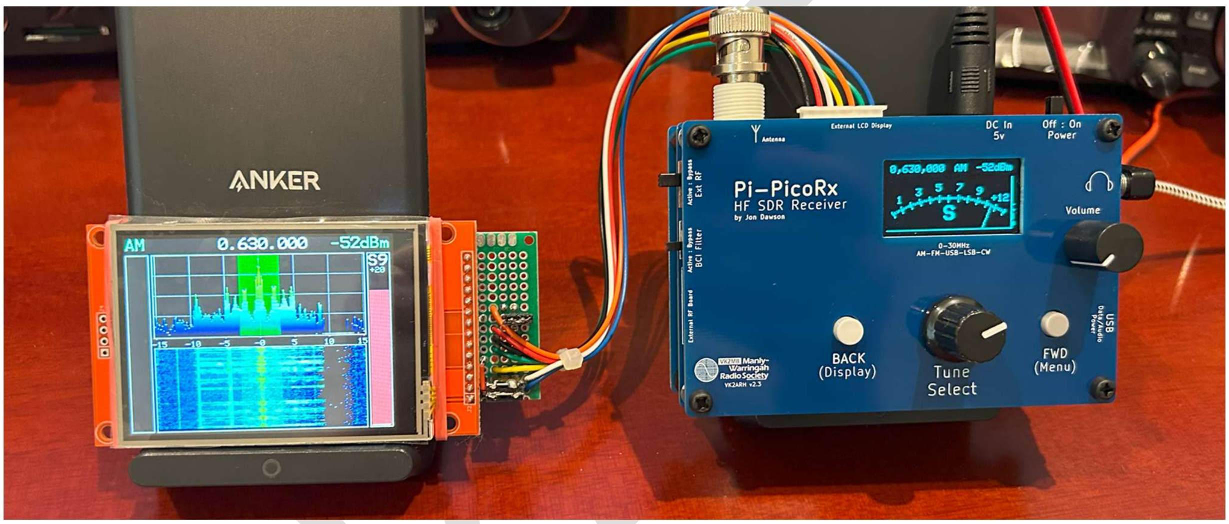
Pi-PicoRx (VK2ARH Version) Construction Manual



By Richard W5ARH / VK2ARH

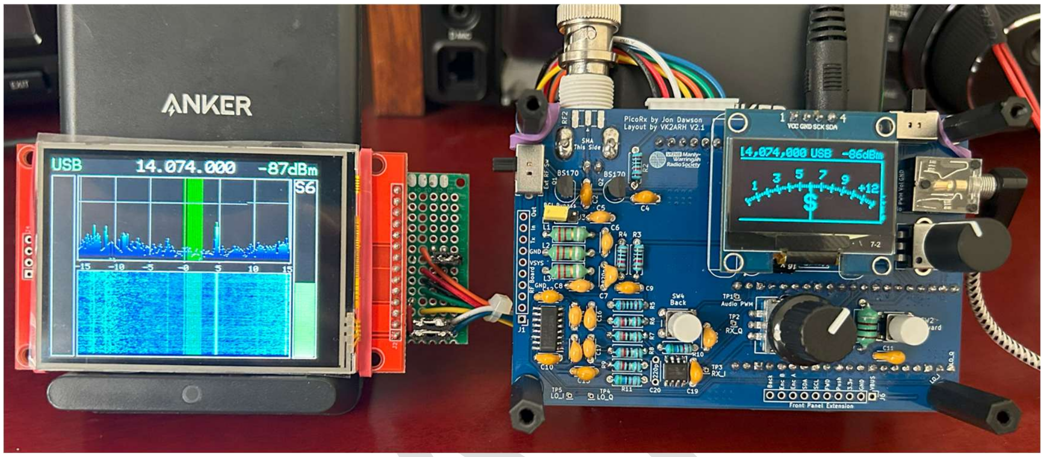
Version 1.0 28 May 2023 Draft Commenced

Table of Contents

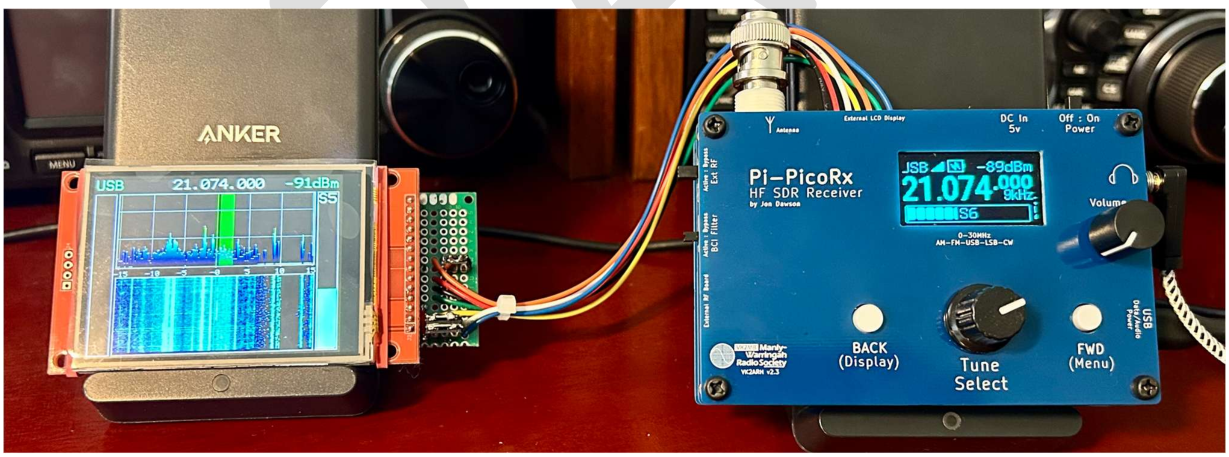
Introduction

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 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 LCD waterfall spectrum scope. The PicoRxcan 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.



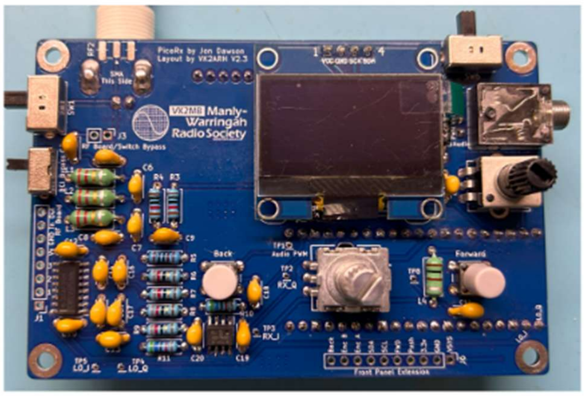
This project is based upon the Tayloe Quadrature Sampling Receiver and the Pi-PicoRx design and software developed by Jon Dawson. A full description of Jon's project, history and theory of operation can be found on his webpage. The three articles on the project shown on the left-hand side menu of his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

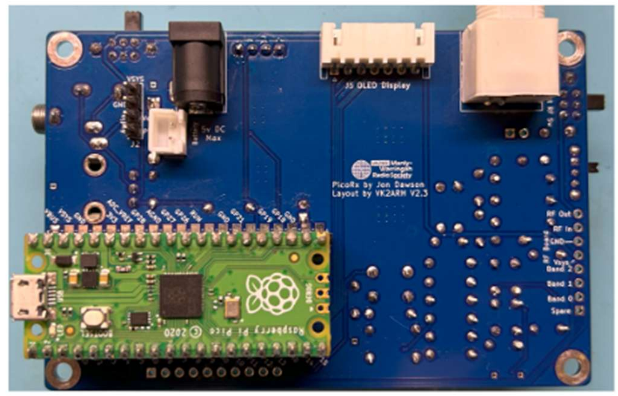
https://101-things.readthedocs.io/en/latest/breadboard_radio.html)

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 will 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:

Feedback

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 Hinsley
VK2ARH / W5ARH

Step by Step Kit Assembly

Equipment Needed:

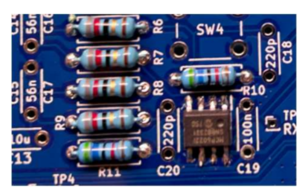
The following equipment is recommended to build the kit:

- 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.
- 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 distracted 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 😊)

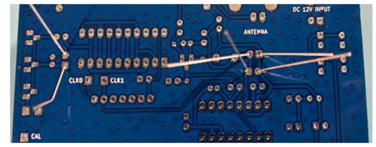
Construction Tips



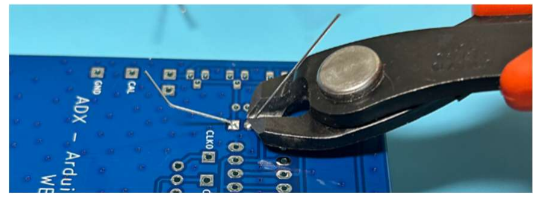
Whilst it makes no difference electronically which way the resistors (in this case) are mounted, good practice is to mount each device with the same orientation. By doing so it makes checking the value of components much easier.

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.



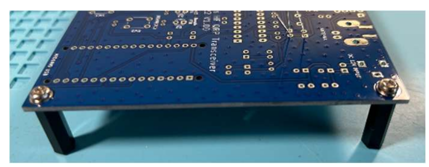
I place my finger on the end of the lead when snipping the lead to absorb the 'snap' shock of the cut – reducing stress on the recently soldered join (**but not on static sensitive devices**). It also stops the lead 'flying' away when you cut it. If the lead is soft and you cut slowly this technique 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 18mm 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.



In this manual each time a component is to be identified and soldered to the PCB, it is listed separately proceeded with a square check box. 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:

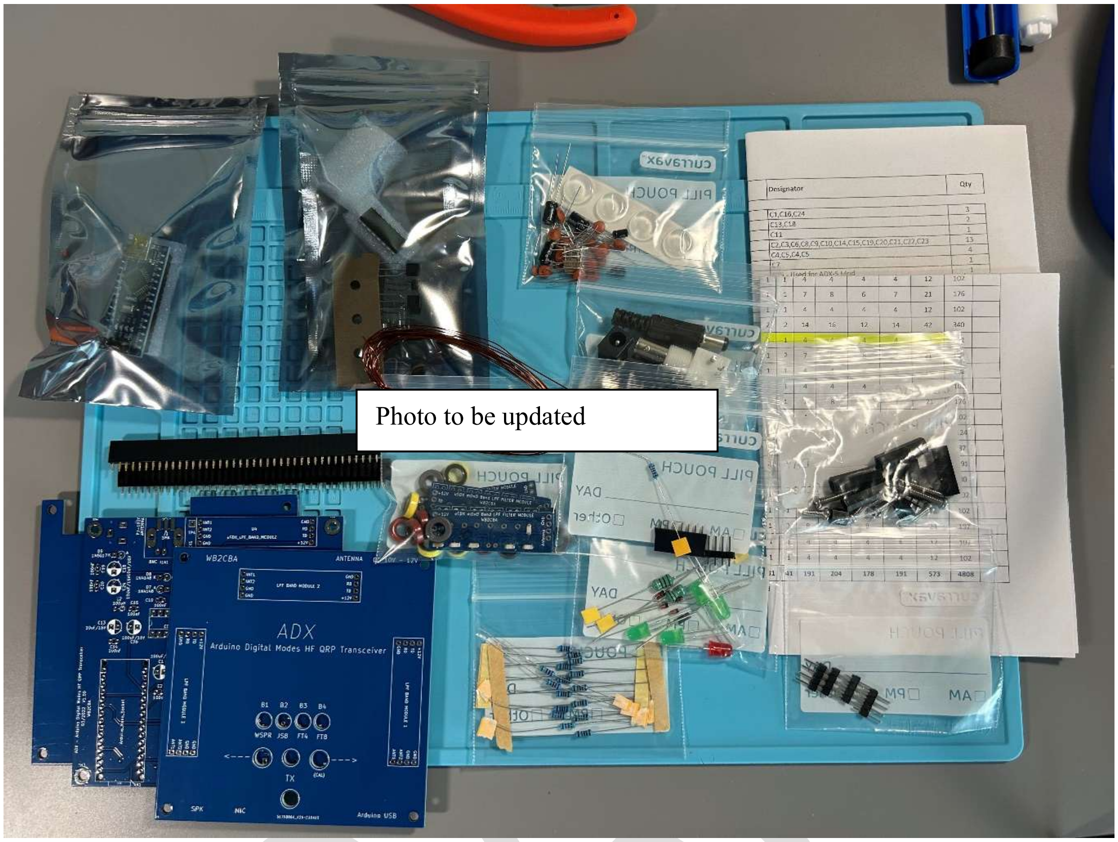
- ☑ Component 1
- ☑ Component 2 etc.

Page | 15 ☑

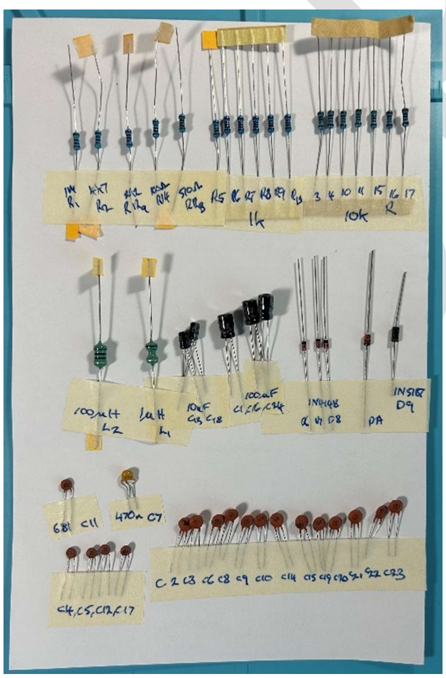
Bill of Materials (BOM)

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	1
	Pin Headers Female x 20	Pico and Mounting External Display	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	Pico	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
	J4	3.5mm Audio Socket	1
	J6	10 Pin Header	1
	J7	Solder Jumper on PCB	
Connectors	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 LCD Screen	1
	J5	7 pin plug + cable	1
	Small Vero Board	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	R_Potentiometer	1
Switches, Knobs and RF Connectors	SW1,SW6	SW_Slide_DPDT	2
	SW2	Micro SPST BCI Bypass Sw	1
	SW3,SW4	6mm x 9.5mm Push Switch	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.

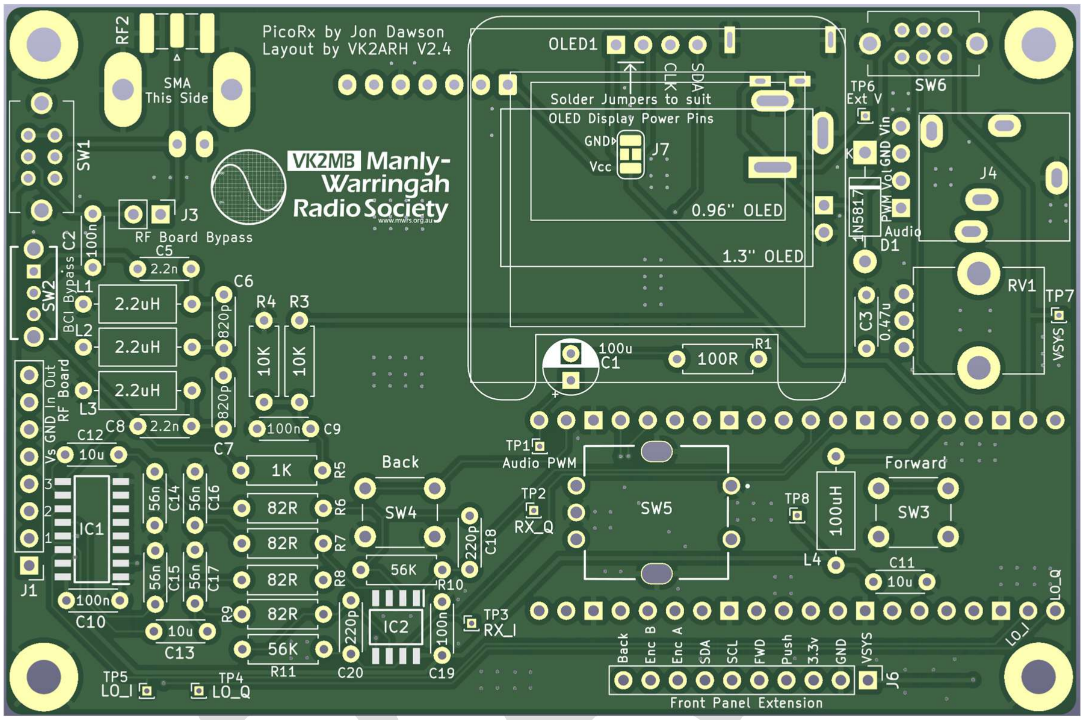


In this example (from a previous Buildathon project) the smaller through hole components have been checked and stuck onto a piece of paper with masking tape before being labelled to speed identification and used during the build. Testing of each component not only assists in identification but ensures 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.

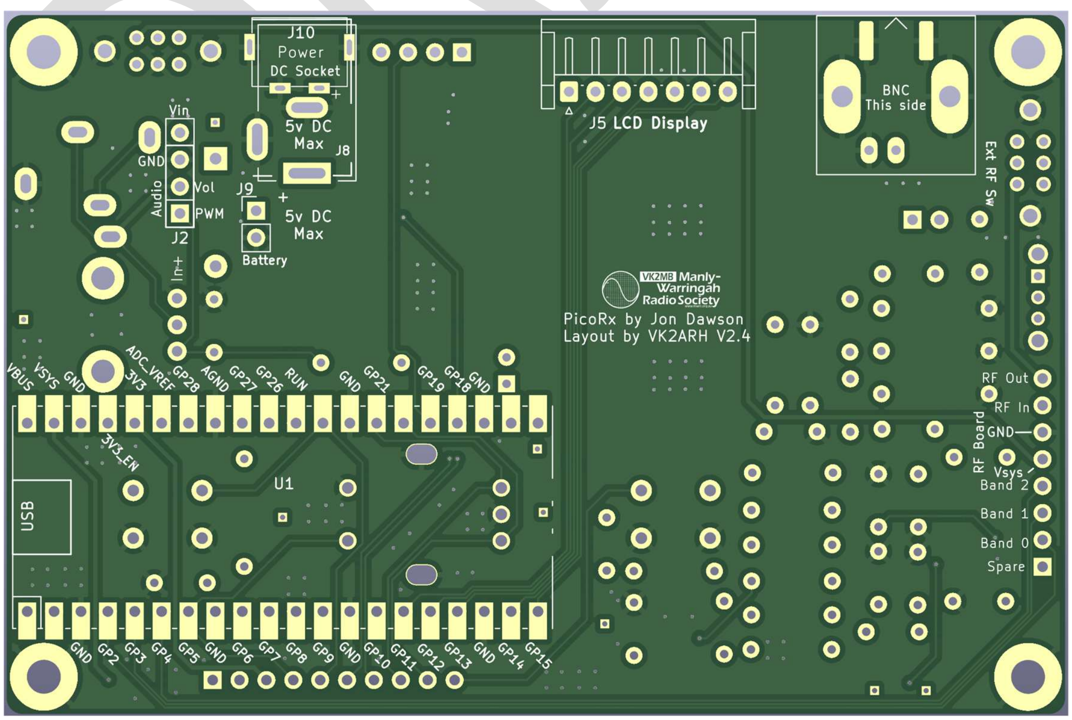
Resistors can easily be tested with a Multimeter; however, capacitors and inductors may need an LCR meter which is not essential but handy if you want to check the tolerances of components. It is not necessary to obtain a high precision meter but be aware and allow for the meter tolerance when measuring 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

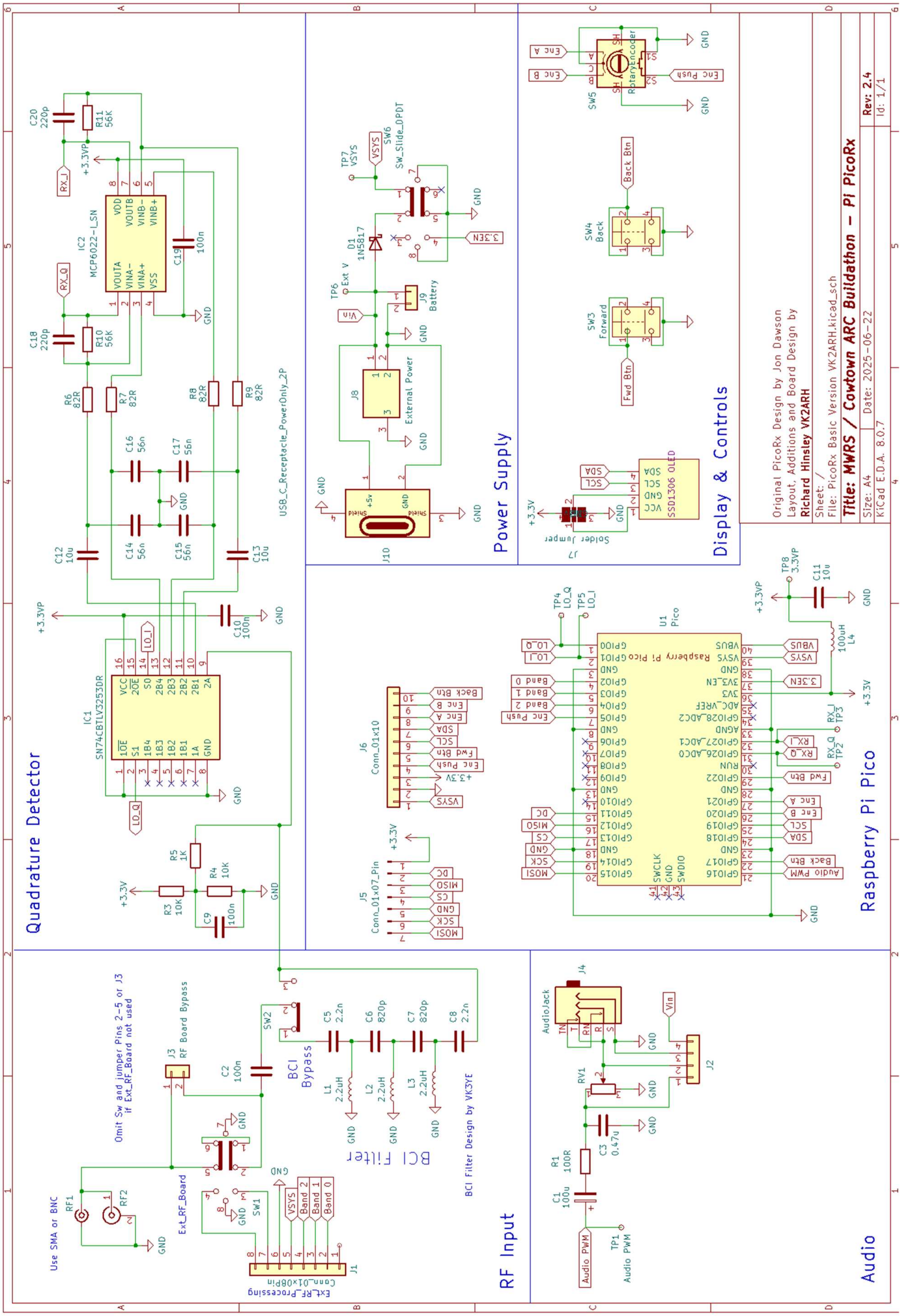
Here is a layout of the component side and rear side of the main PicoRx PCB. Note this images show version 2.4 – photographs and diagrams in most of the construction manual show the version 2.3 board. The only difference was the inclusion of J10 (USB-C connector) and some associated power connections.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

There is nothing too complicated about the PicoRx build 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. In this way it makes handling and soldering components to the board easier.

Step 1: SMD IC's

The most difficult 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.

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 SO-8 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

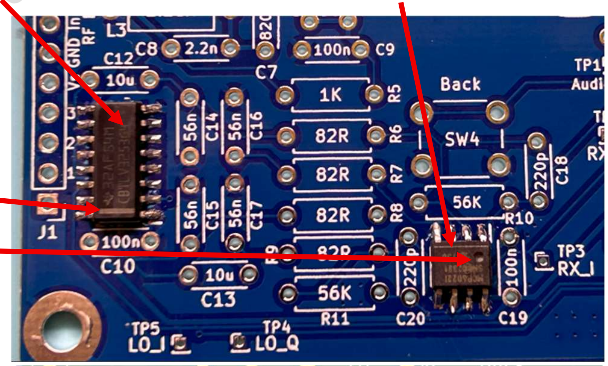
This is a commercial process where a stencil is used to squeegee solder paste onto a PCB, parts are picked/placed onto the PCB, and the boards are cycled through a thermal profile in a reflow oven. Many hobbyists have mastered this and used this for limited production. Stencils can be ordered along with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

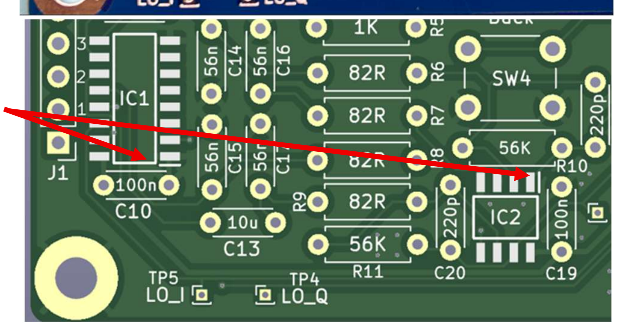
The choice as to which technique to use will depend on the equipment you have available and your desire to expand your SMD soldering skills. This project can easily be completed by hand soldering the SMD's to the board. The following photograph shows the location of IC1 – SN74CBTLV3253DR chip and IC2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



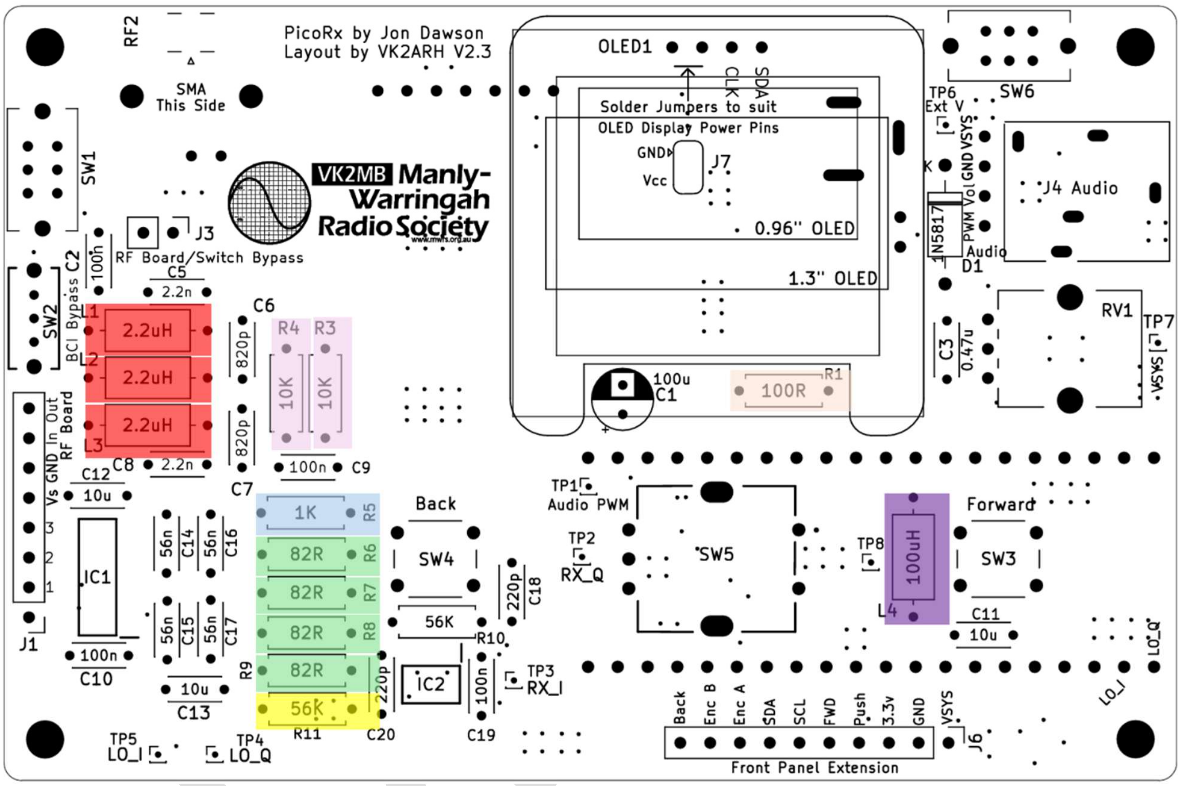
The pin 1 location is shown on the PCB silk screen for each IC by a solid line located next to the pin 1 as shown here.



Step 2: Resistors and Inductors

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

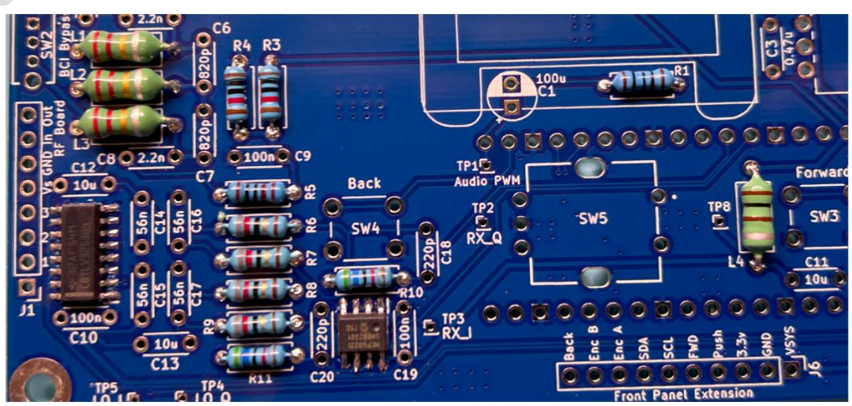
Colour	Reference	Value	Marking
	R1	100R	Brown-Black-Black-Black-Brown
	R3,R4	10K	Brown-Black-Black-Red-Brown
	R5	1K	Brown-Black-Black-Brown-Brown
	R6,R7,R8,R9	82R	Grey-Red-Black-Gold-Brown
	R10,R11	56K	Green-Blue-Black-Red-Brown



Next 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:



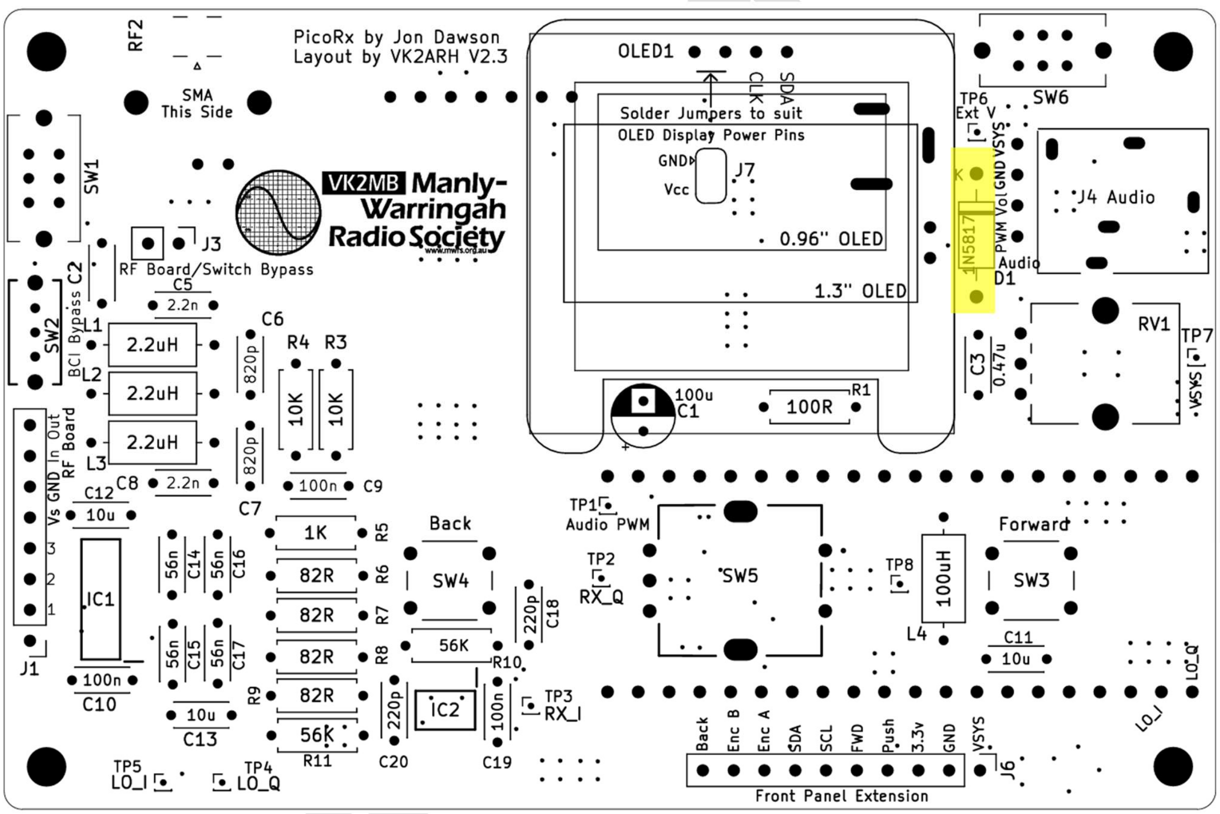
Step 3: Diode

There is only a single diode used in this project:

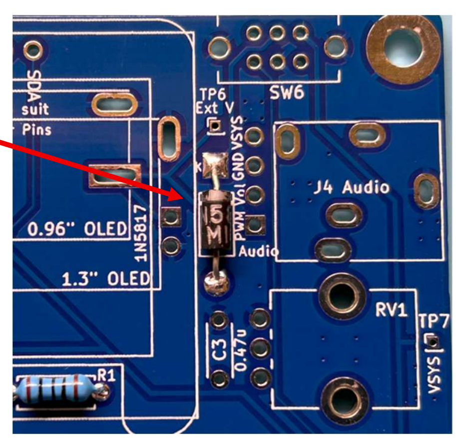
Colour	Reference	Value	Marking
	D1	1N5817	Black body with Silver Band at one End 1N5817 written on the diode

Ensure that you 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:

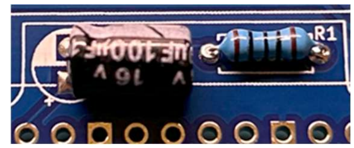


Once installed on the PCB the finished result should look like this:



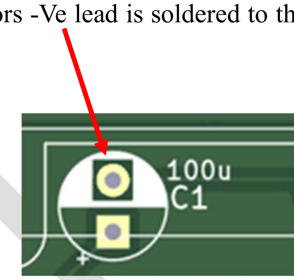
Step 4: Capacitors

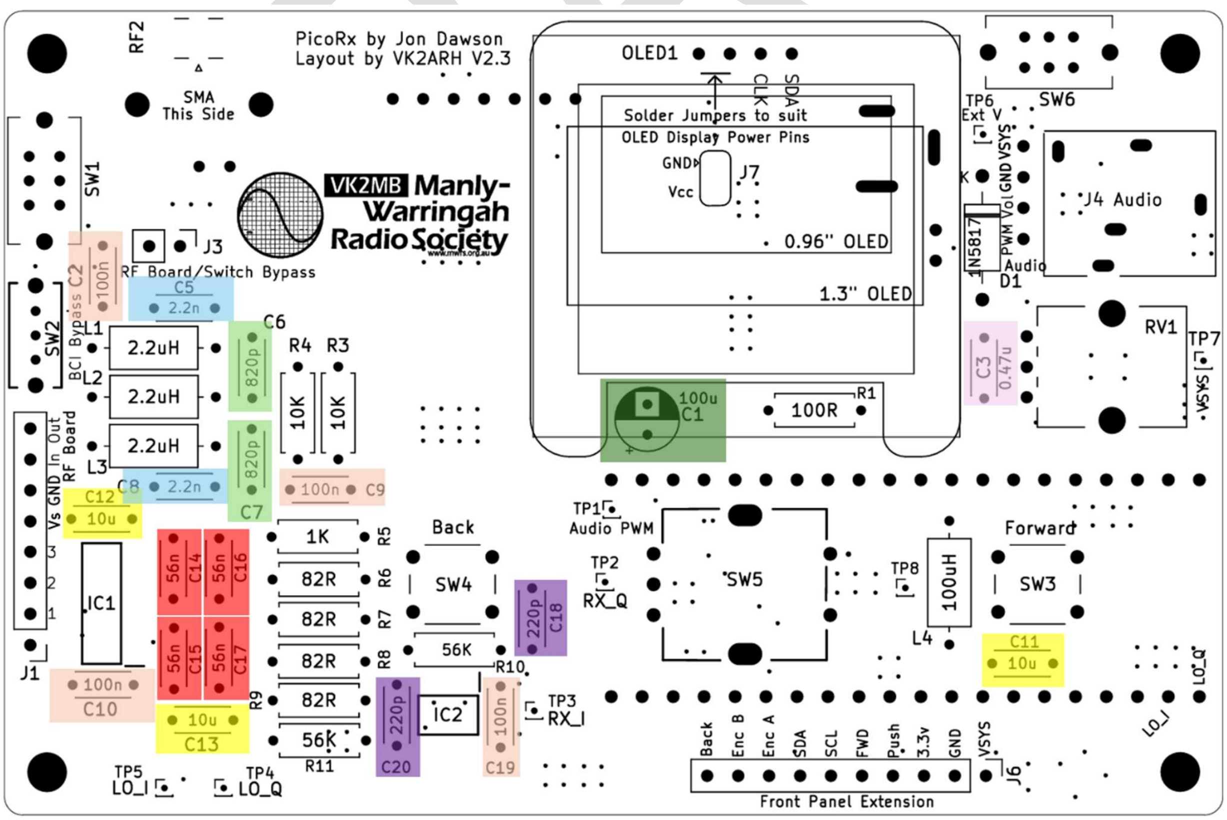
This project uses an electrolytic capacitor and standard MLCC capacitors. The electrolytic capacitors are polarized and need to be installed in the correct orientation. The -Ve lead is the shorter of the two and the body of the capacitor clearly indicates the -Ve lead. The MLCC's are not polarized and are identified with a numeric representation of their value. (See p40 for a capacitor identification chart).



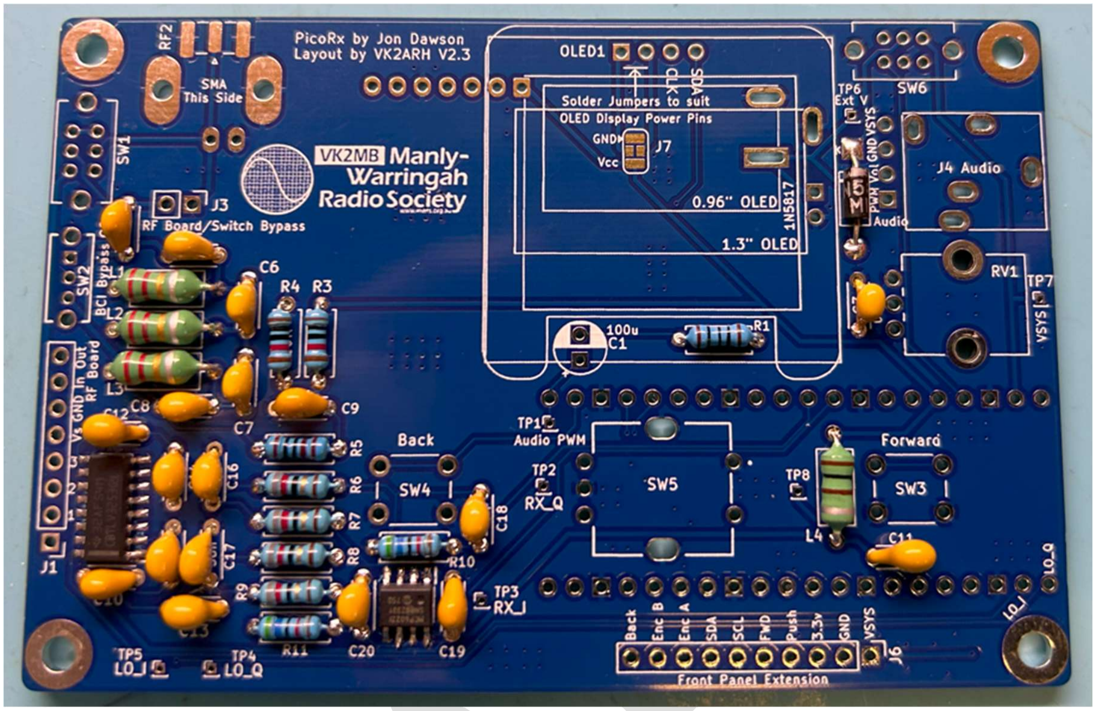
When installed on the board, the -Ve (shorter lead) on C1 is installed facing the shaded side of the print screen symbol after bending the leads through 90º and lying flat on the PCB as shown in the top right hand photo above. This orientation ensures the necessary clearance for the top panel. The electrolytic capacitors -Ve lead is soldered to the round solder pad.

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



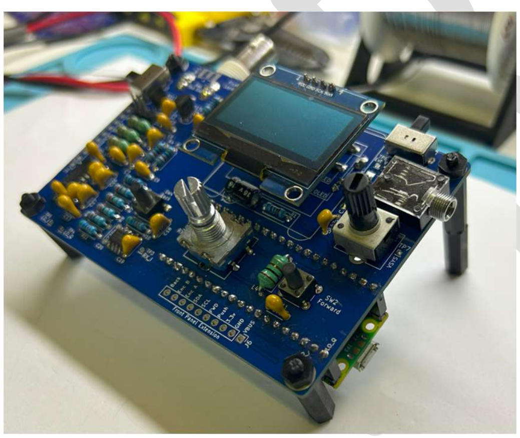


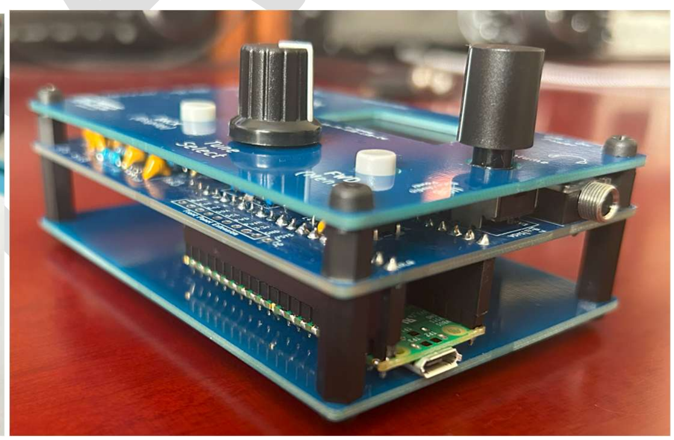
Once the capacitors are installed the board should now look like this before installation of C1:



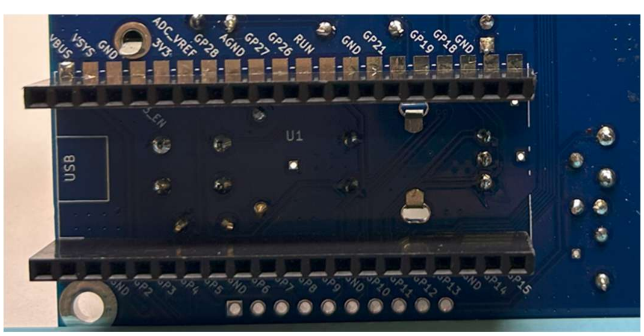
Step 5: Installing Pin Headers and Sockets for the Pi Pico

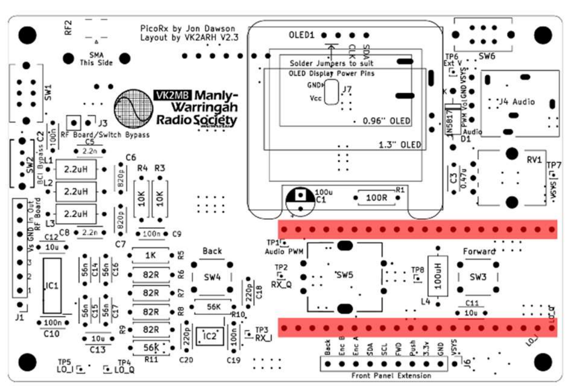
The Pi Pico is supplied without any header pins attached so we need to attach male header pins to the Pico and female header pins to the main PCB. The Pico is mounted on the underside of the board as shown in the photos below:





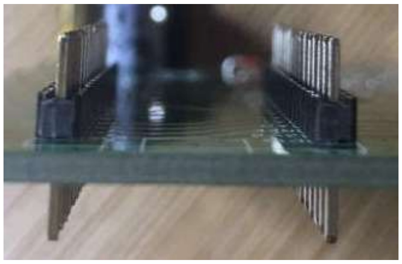
This will give you an idea as to the mounting location and orientation of the Pico. The female headers are positioned on the back of the board as shown in this photograph.

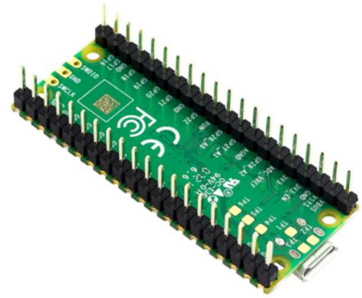




Installing the Male Pin Headers onto the Pico

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



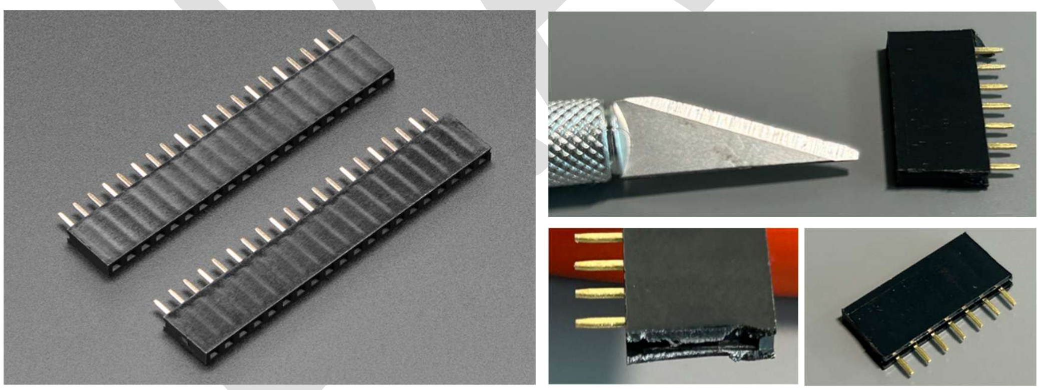


The best way to do this is to place the pin headers into the main PCB (long lead into the board) to ensure correct alignment as shown on the left. Then sit the Pico onto the short pin headers component side up and carefully solder the **Pico to the pin headers**. Solder quickly and spread the heat load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

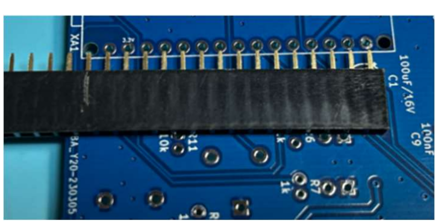
I strongly recommend that you **DO NOT solder the Pico pin headers directly to the main PCB** but install female pin headers on the main PCB to hold the Pico, this gives you the flexibility of replacing the Pico.

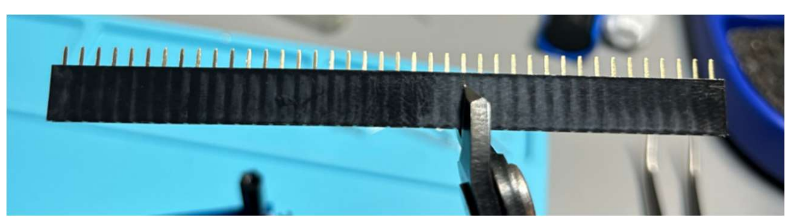
Preparing the Female Pin Header Strips (The illustrations shown relate to preparing the header strips for the Arduino Nano used in the ADX project, but the technique and principles are the same when preparing them for the Pico's **20 Pin** header not 16 pin as shown in the illustrations).

If you do not have a 20 pin female header a longer header can be cut down to 20 pins. 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. 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 21^{st} pin on each strip, you will be left with two 19 pin strips which can be repurposed on another project.

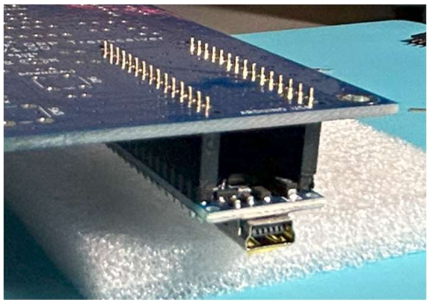


Double count the number of pins to be left on the header before you make the cut. Take your wire cutters and cut the strip at the score mark. Hint: if you cut the 20 pin strip from each end of the original 40 pin header, you will have one nice clean end on each 15 pin header which you can solder toward the edge of the ADX and will look a lot neater.





Now to 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:



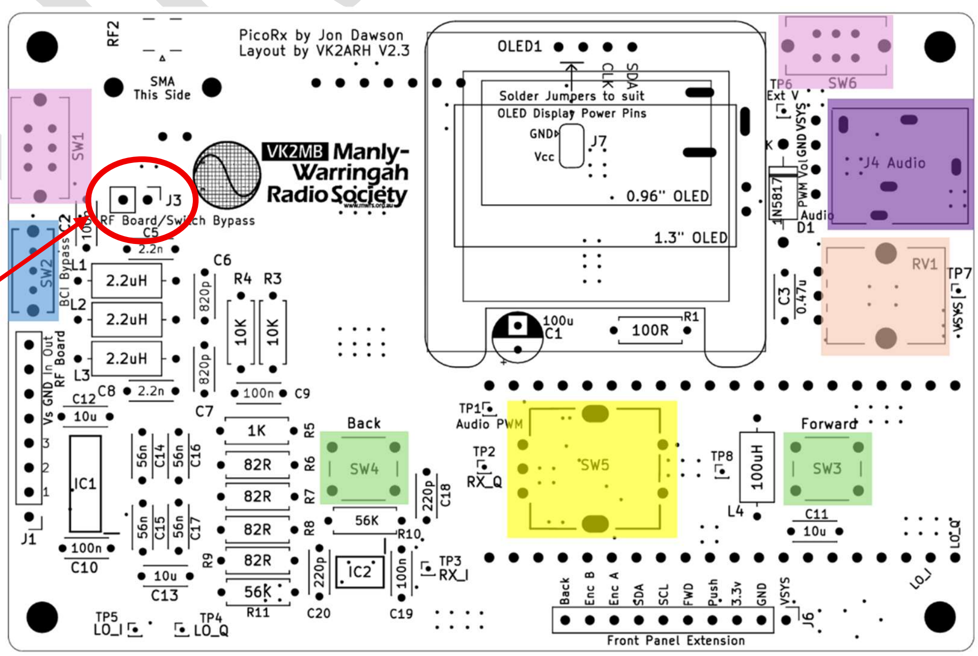
Step 6: Installing Switches, Sockets, Encoder and Variometer

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	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	

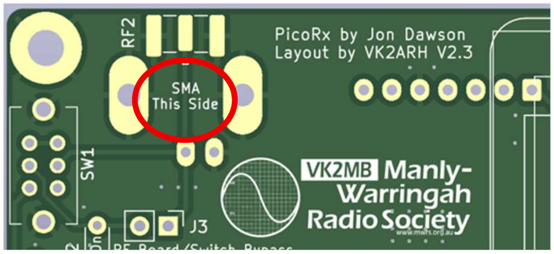
NOTE: If you plan to operate the PicoRx as a stand-alone unit with no external RF board for experimenting with external components it is not necessary to install SW1, instead you need to solder a jumper over J3 to connect the RF input permanently to the main PCB.

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

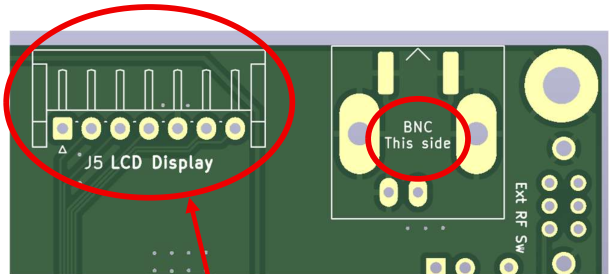


Step 7: Antenna Connector and LCD Display Sockets

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



Front (SMA)

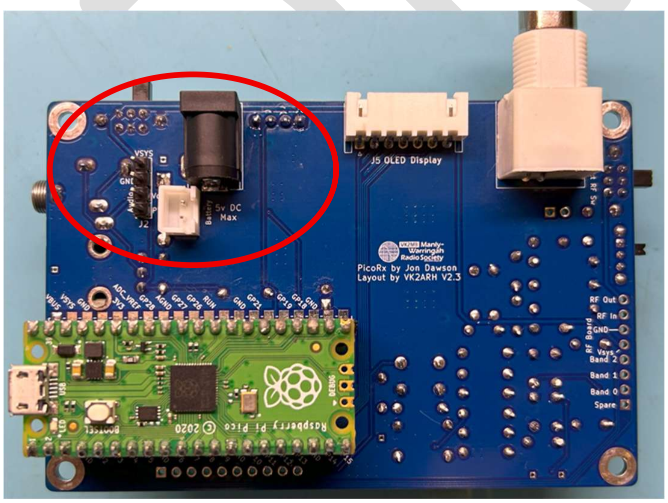


Rear (BNC)

The optional colour LCD 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.

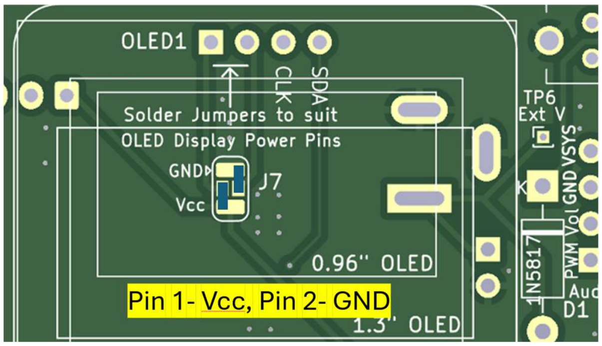
Step 8: Power Sockets

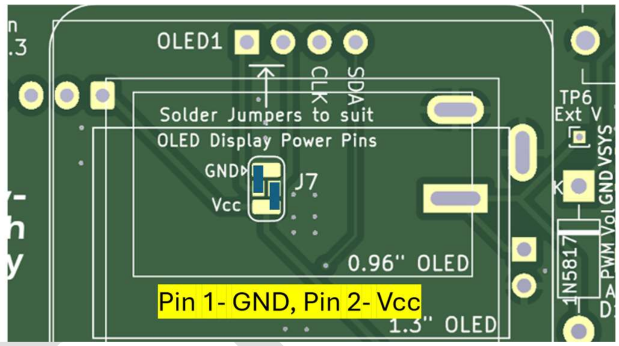
When installing the power socket – you have a few choices. You can either install the 5.5mm x 2.1mm DC socket <u>**or**</u> the USB-C power only socket. The 2 pin JST socket is an option to support an external battery connection or power supply connection. You should install a 4 pin header at this time for external connections for sound and power should you wish to use that capability. All power options are mounted on the rear of the main PCB. The finished board with the Pico installed will now look similar to this. **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 (don't ask me how I know ☹). The Vin pin on the 4 pin header can be used to supply power from an external power supply mounted on a daughter board if you wish to use that capability.



Step 9: Installing the 1.3" OLED Display

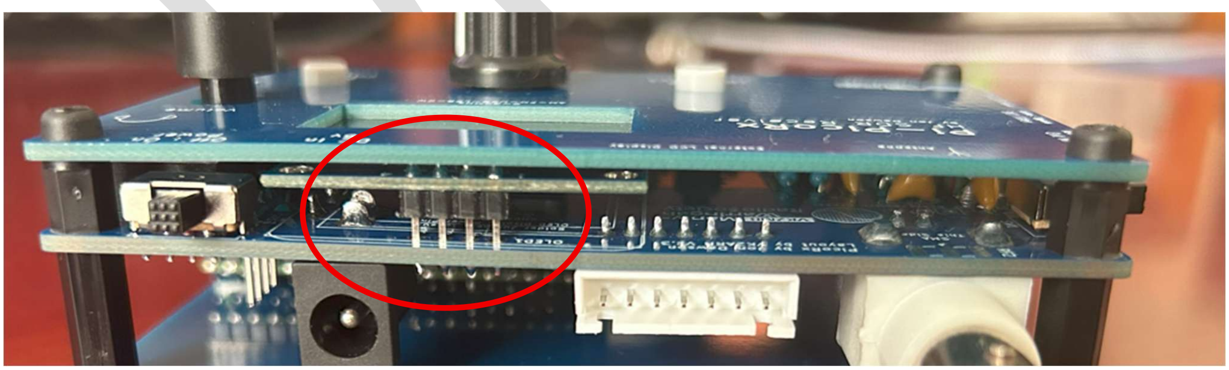
Before installing the OLED display, you should check the VCC and GND pin location on your display. Different manufacturers vary the location of these pins. To accommodate both options, J7 facilitates connection of the OLED pins 1 and 2 to either Vcc or GND. Solder and short out the pads on J7 to achieve the desired connections as shown on the diagram below:





In my own PicoRx the OLED display had pin 1 as Vcc and Pin 2 as GND – so the jumpers were soldered as shown here to the right. Check your soldering with a multimeter to ensure that you have not shorted out the power supply and the required polarity is correct.

Care should be taken when soldering the OLED to the board to accomplish two aims. The first is to ensure that it is positioned squarely to the board and secondly that it is elevated above the board. In this way it sits flush up against the top panel cover and the OLED display is aligned squarely in the viewing window.

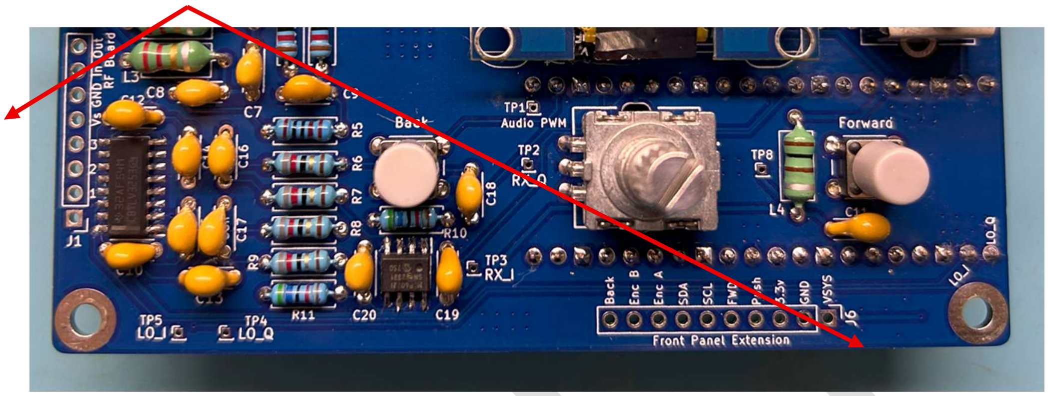


I have found the easiest way to do this is to first solder the four-pin header supplied with the OLED ensuring that the longer pins are on the rear and perpendicular to the OLED board. Assemble the top cover using the 7mm spacers and screws having placed the OLED (with its four header pins installed) in but not yet soldered to its mounting holes. Turn the picoRX upside down align and position the OLED and solder only one of the pins. 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 it 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 shown above.

Congratulations 😊 you have now completed the build of the PicoRx radio – all that is now left it to install the software, power up and test that it is working, mount it in the PCB enclosure and start using and enjoying your PicoRx.

Before powering up the Pico on your newly built board for the first time I recommend that you remove the Pico from the board and then 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:



Then connect power to your PicoRx, switch the power switch to ON and you should see just under 5v across these same measurement points.

Turn the power switch OFF, and proceed to install the software onto your Pico controller (Step 10 below) before installing the Pico into the header pins on the board.

As an extra precaution during the first power up on the board, if you can connect the 5v input sourced from a current limited power supply set the current limit to 50mA, and this will current limit the power in the event of any shorts etc. The PicoRx will draw approx. 36mA with the standard 1.3" OLED and 95mA if you have added the optional external 2.8" Colour LCD display.

Step 10: Installing the Software

Loading the firmware into the Pico is a relatively easy process.

- 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, however the Pico2 with the more powerful processor uses approx. half the computing power to operate the PicoRx.
- Unzip the package which should contain two files as shown below (for the Pico2 version)

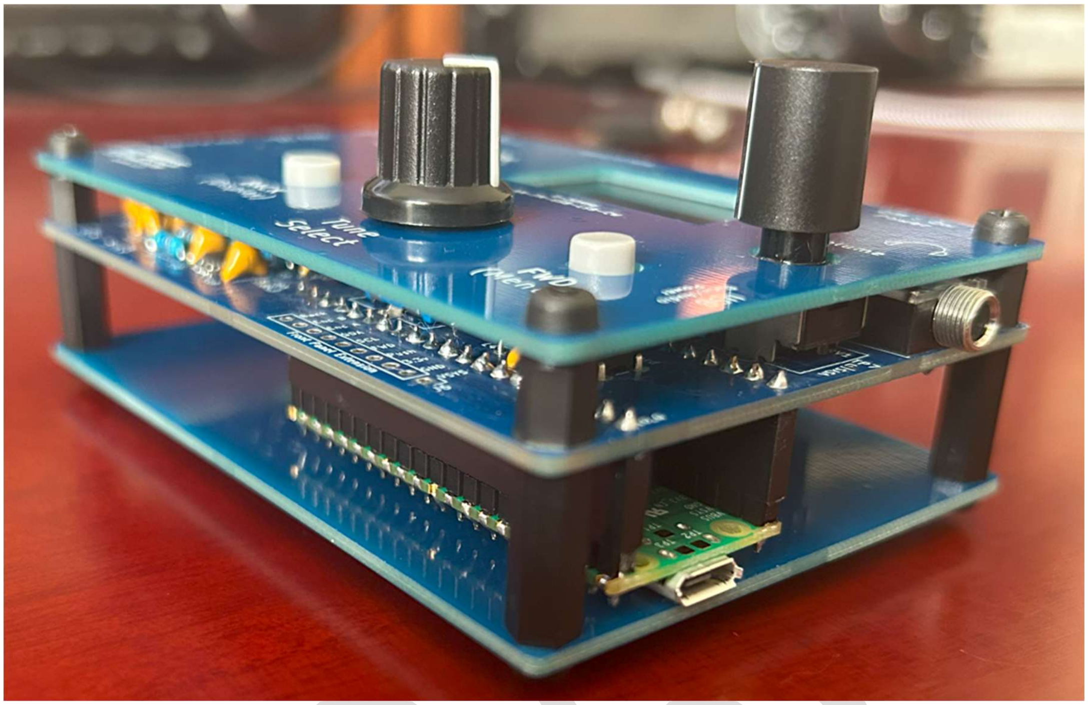
Name	Date modified	Type	Size
Today			
battery_check_pico2.uf2	5/27/2025 7:55 PM	UF2 File	58 KB
pico2rx.uf2	5/27/2025 7:55 PM	UF2 File	452 KB

To load the firmware:

- 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 into the USB drive. When the copy is complete, the Pico will restart and your OLED screen should come to life with the PicoRx user interface.
- Jon's original source is available from the same link shown above for those who wish to modify and develop the software.

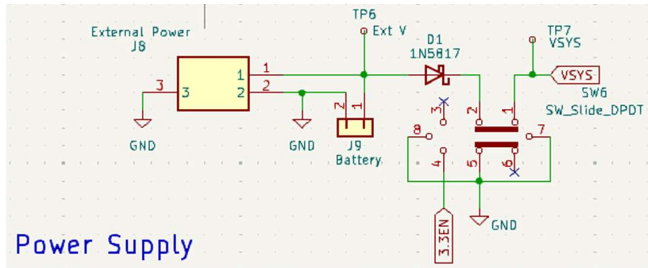
Housing the Pi-PicoRx

Assembling the



Additional Technical and Optimization Information

Power Supply Options



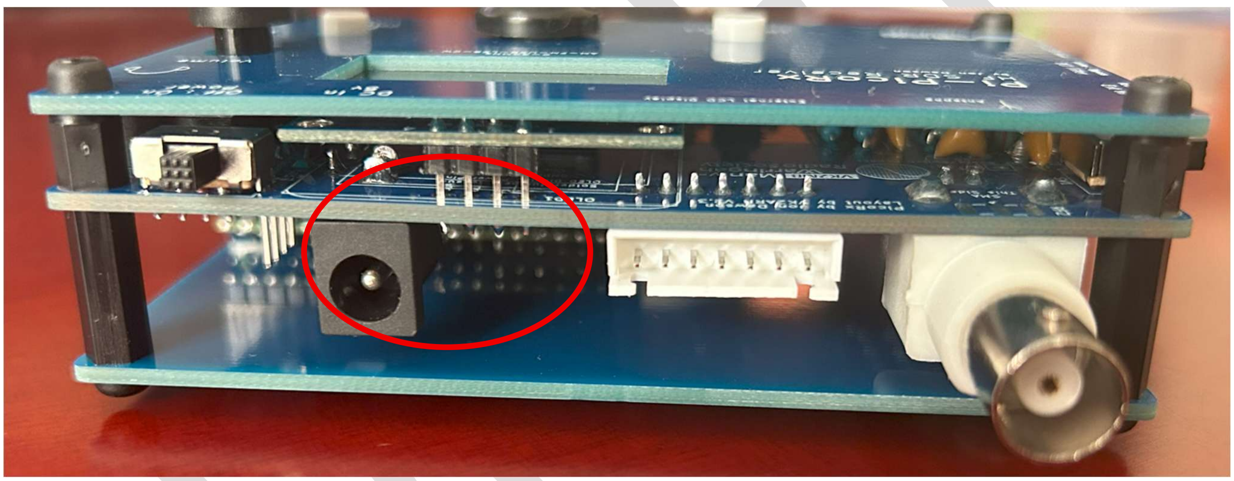
There are several choices to power the PicoRx.

- External 5v DC Supply
- Pico USB Port
- Battery (3.7 - 4.5v)
- Battery power management system feeding into the battery connection.

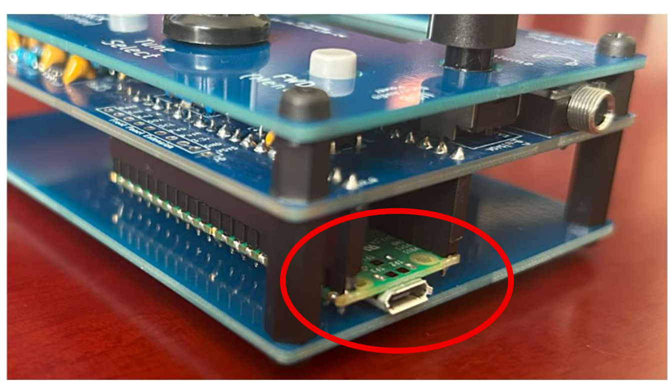
External 5v Supply via DC Plug

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 VSYS on the Pico directly and is reverse polarity protected using a 1N5817 diode.

Do not connect more than 5v DC to this socket



Power using the Pico Micro USB Port

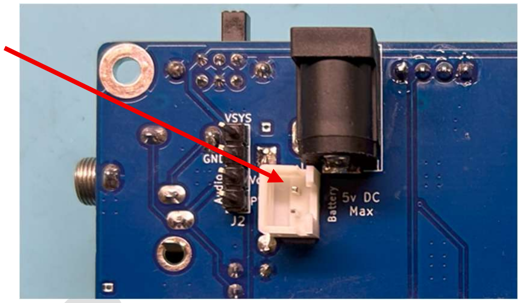


The PicoRx can be powered directly from the USB port on the Pi Pico. This is a convenient way of powering the device if you are using a USB connection for CAT control or USB audio.

The VSYS output port on the Pico supplies 5v and 3.3v power to the external power connections.

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 molex connection to an external battery and the +_ve pin is indicated by the arrow.

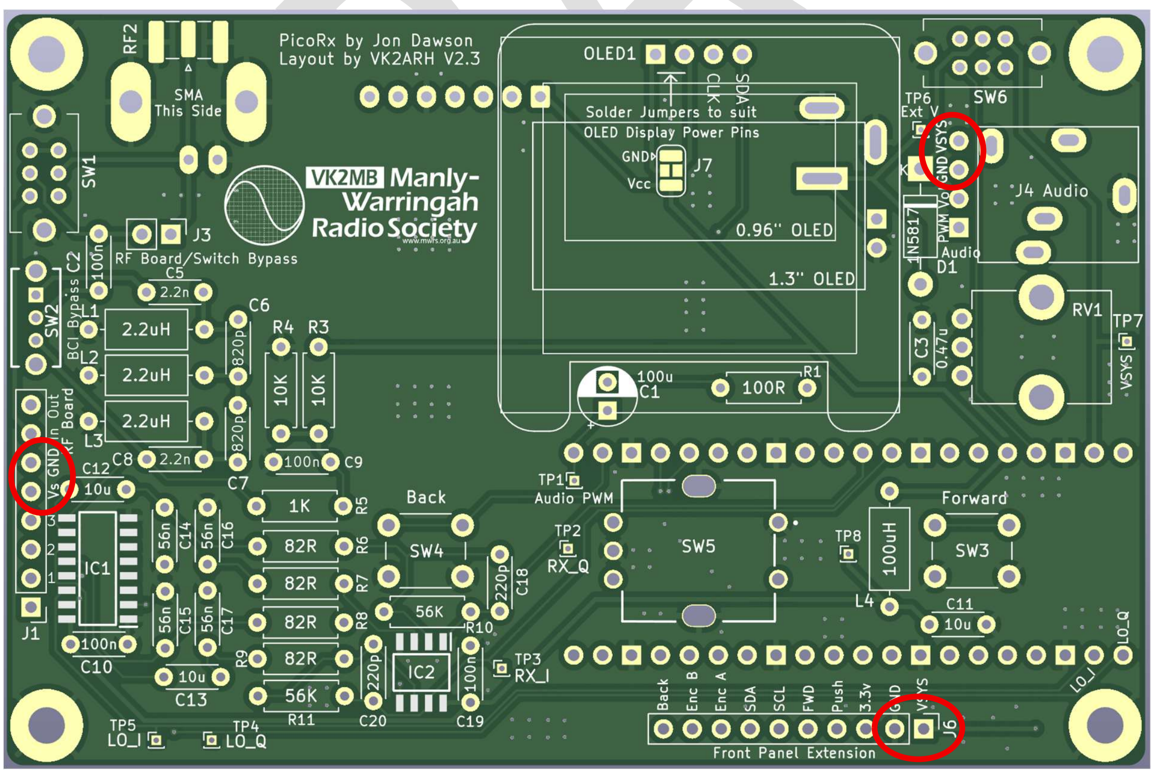
The PicoRx can operate from a 3.7v Lithium Iron battery or from 3 x 1.5v AA or AAA batteries.

Power should not be connected to the DC plug if you 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 Iron battery power management modules available which can be used externally to the PicoRX main board to supply power and charge the Lithium Iron battery. These external power sources can be connected to the PicoRx using the molex battery connector. As the design and operation of these modules vary, the individual builder should assume responsibility for the operation and connection of these modules and batteries to the PicoRx.

Power Supply to External Boards and Equipment

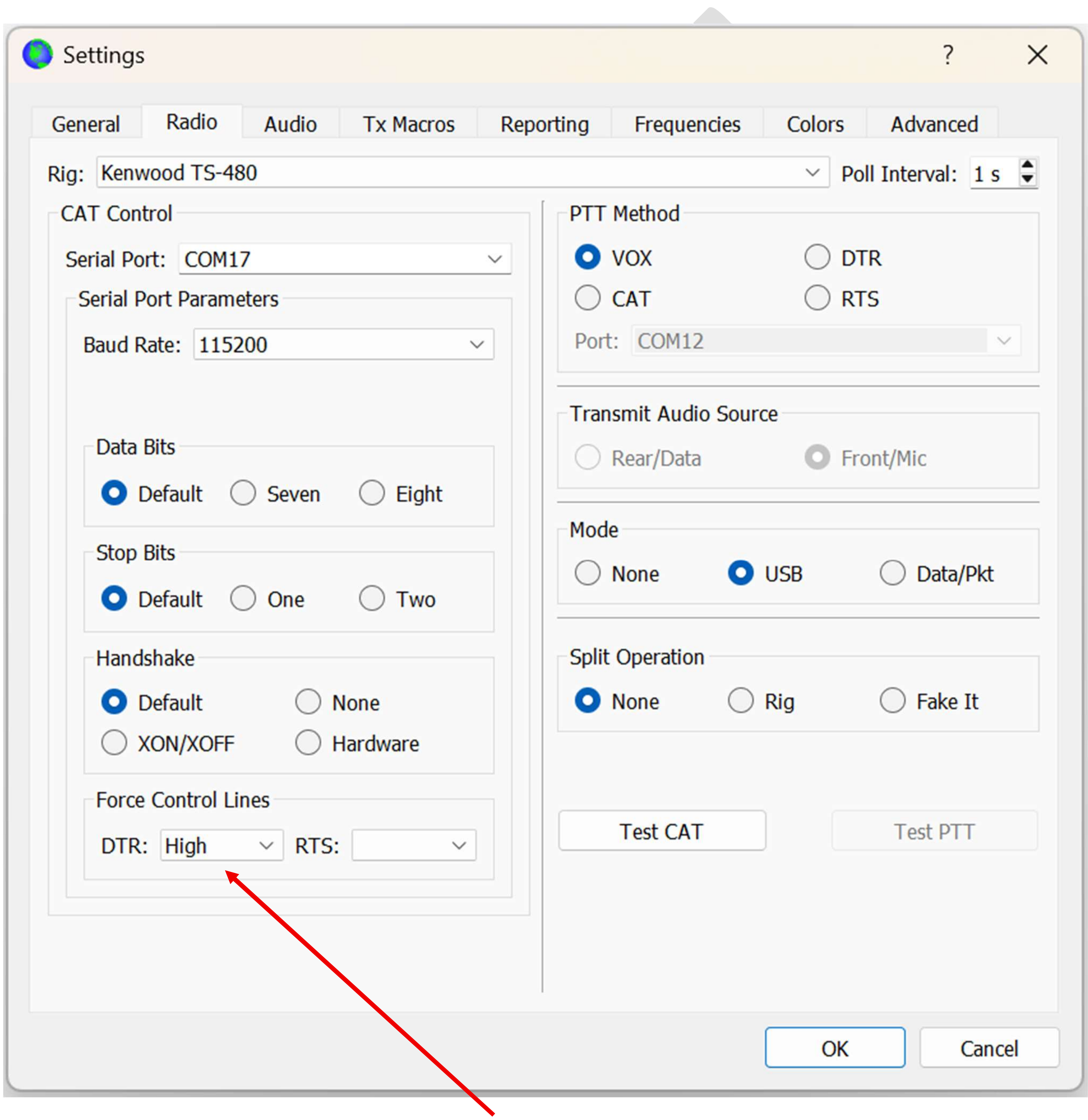


The board has been designed to provide experimenters with choices to power external boards or components. VSYS and GND pins are available at the connectors for the RF board, audio extension pins, and the Front Panel Extension Connectors which also contains a 3.3v output. Do not exceed 300mA total current 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)

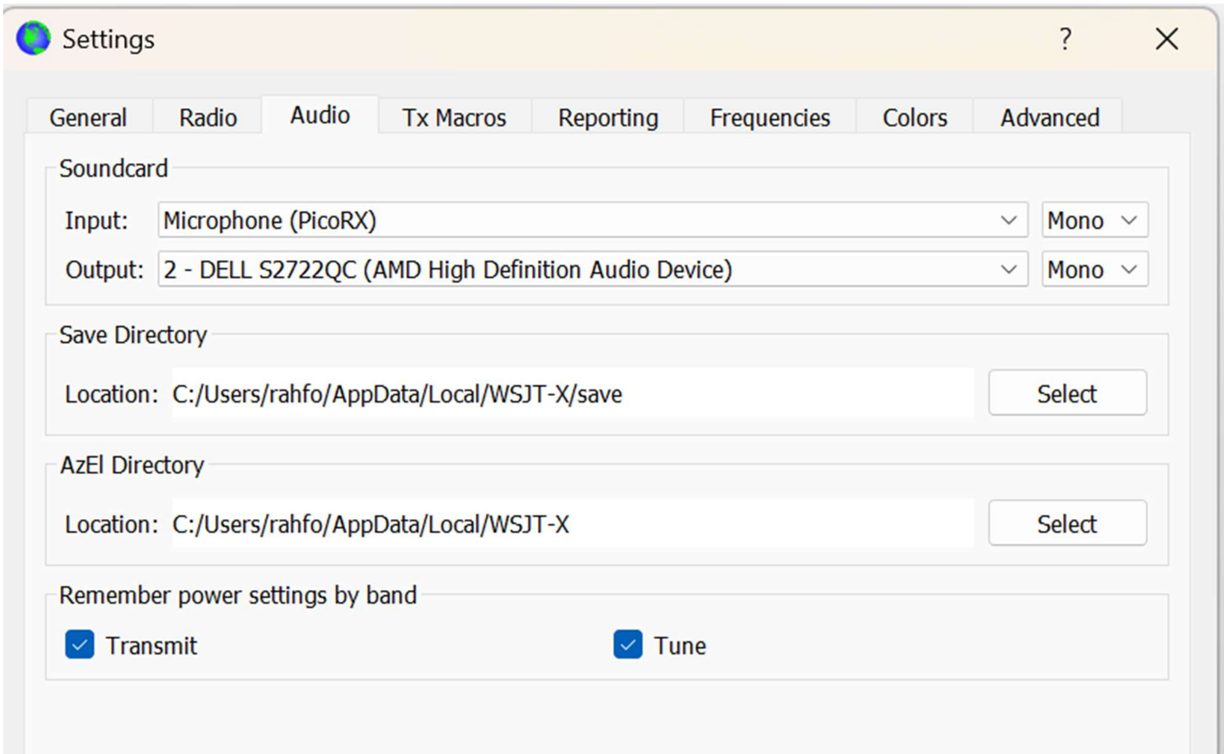


NOTE: For Windows PC's you will need to force DTR high, I believe this is not the case for Linux and iOS.

WSJT-x Audio Settings

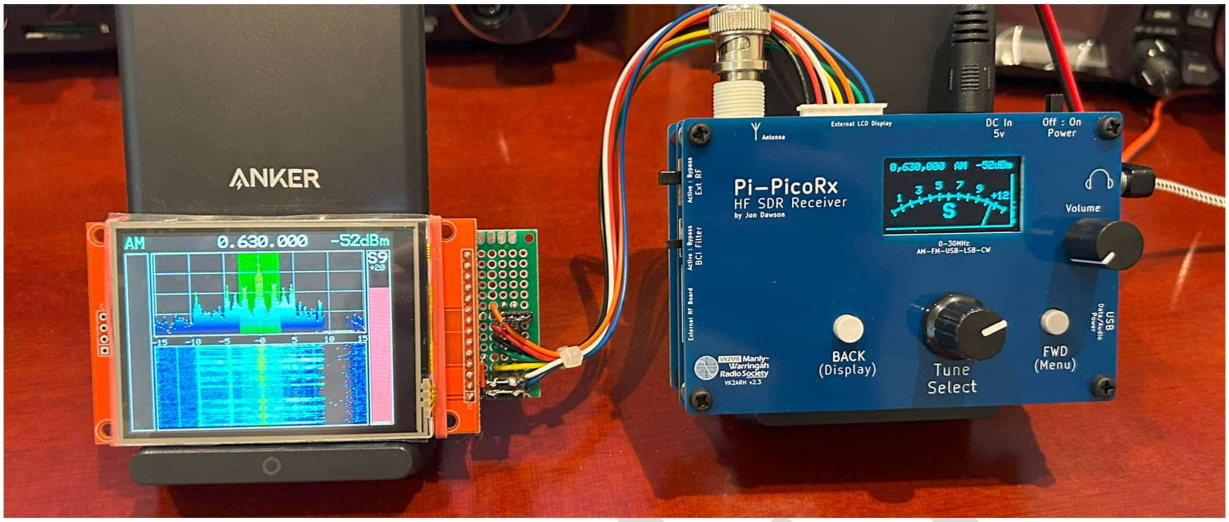
The Pi Pico Rx supports USB audio, and when connected should appear as a USB microphone. This allows a host device to easily make audio recordings (e.g. using audacity), and is compatible with software such as wsjtx, fldigi and QSSTV. When combined with USB cat control allows a fully functional PC connection 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

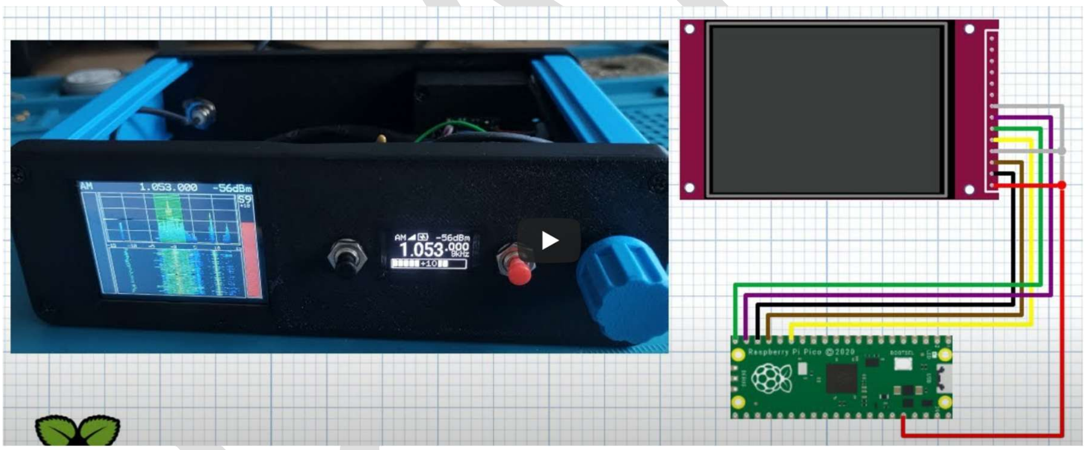


Connecting an External Colour LCD Display

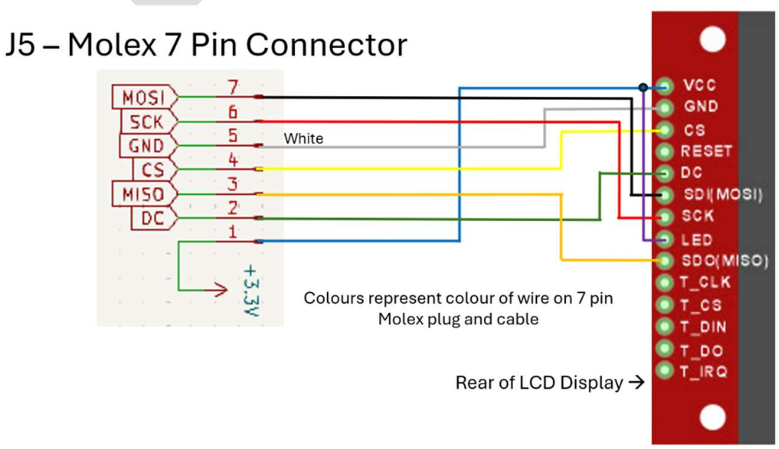
Whilst not essential for the operation of the PicoRx, the addition of an external colour waterfall display makes an impressive addition to your project.



A range of mounting options are available to builders to suit their own requirements and example is shown below. The provision of the external front panel extension connector makes mounting in external enclosures a little easier. Here is an example from Jon Dawson's website.



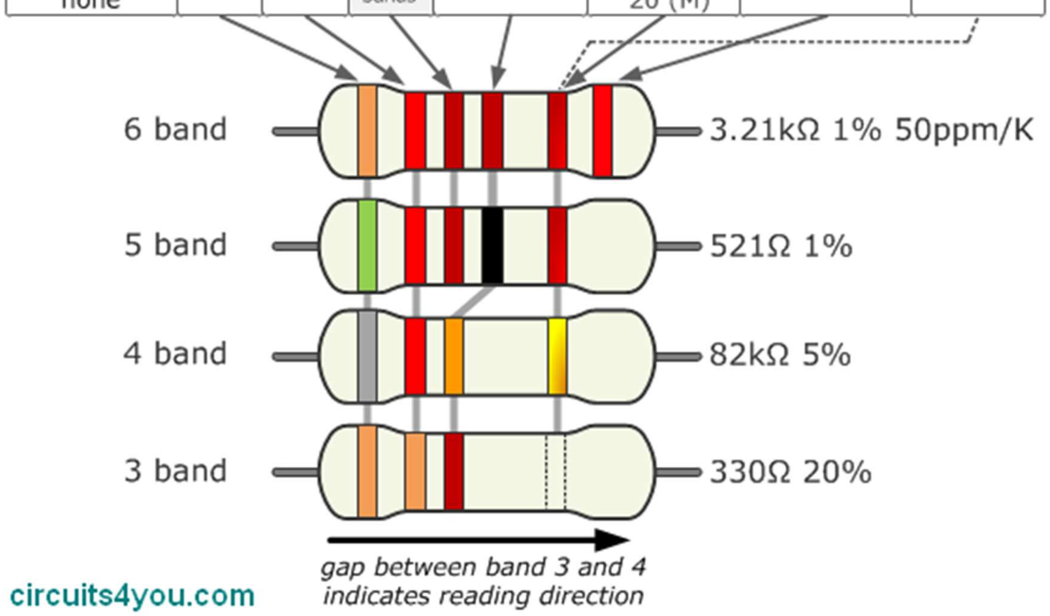
The PicoRx mainboard contains 7 pin molex connector which can be wired directly to the LCD screen using the connections shown below.



Useful Information

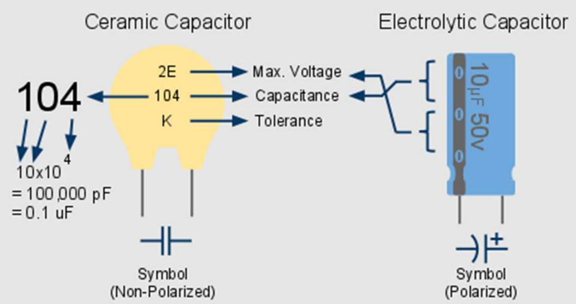
Resistor Color Code Chart

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 Chart

Capacitors

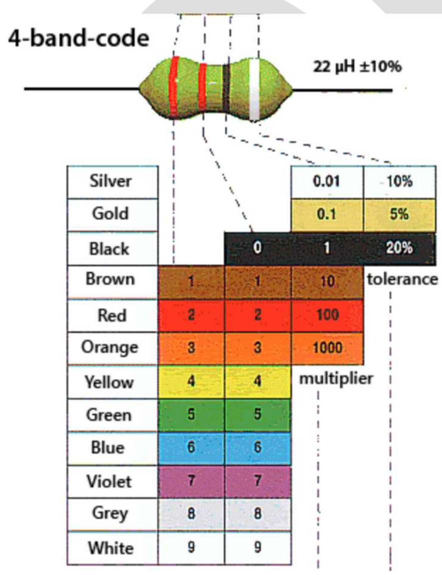


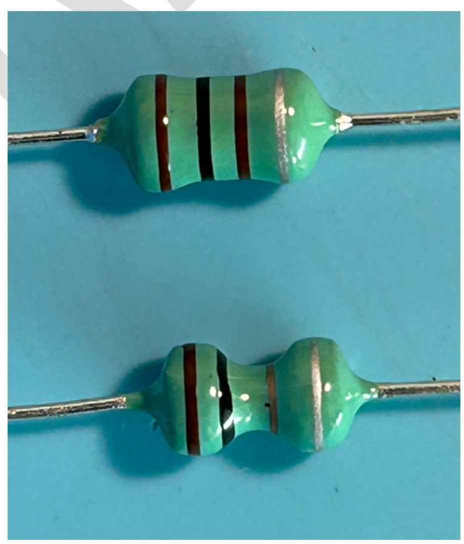
Max. Operating Voltage	
Code	**Max. Voltage**
1H	50V
2A	100V
2T	150V
2D	200V
2E	250V
2G	400V
2J	630V

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





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