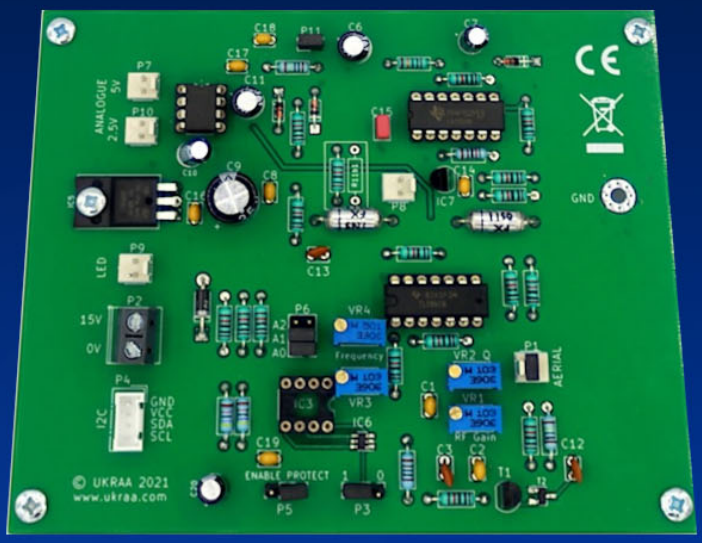
UK Radio Astronomy Association

Very Low Frequency Receiver

Kit Assembly Manual



www.ukraa.com

The UK Radio Astronomy Association
A charitable incorporated organisation
Registered Charity No. 1123866

Acknowledgements

Design Team

The UKRAA VLF Receiver design is a combination of public domain material and original improvements by BAA members John Cook and Peter King. The circuit board layout was undertaken by John Cook and Andrew Lutley.

Contributors

The following authors have contributed to the VLF Receiver Kit Assembly Manual: Andrew Lutley, Alan Melia, Dr Laurence Newell, Norman Pomfret and Andrew Thomas.

Table of Contents

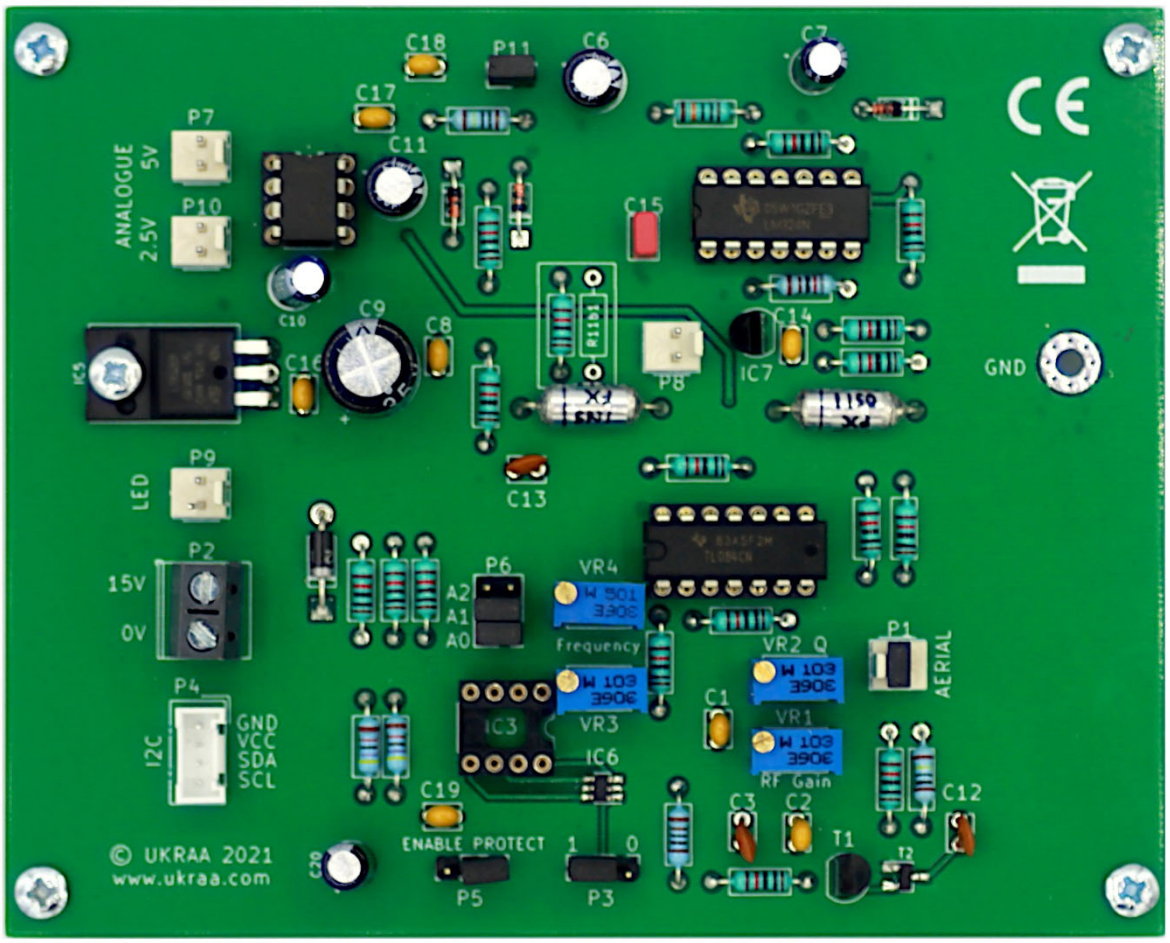
Introduction

UKRAA

The UK Radio Astronomy Association (UKRAA) is a non-profit-making charitable incorporated organisation. It was established by the Radio Astronomy Group of the British Astronomical Association (BAA) to facilitate the production and sale of radio astronomy products.

Any suggestions or recommendations for improvement of this Manual would be appreciated. See the Contacts page for further details.

The UKRAA VLF Receiver



The assembled UKRAA VLF Receiver

The UKRAA VLF Receiver is designed to record Sudden Ionospheric Disturbances (SIDs) induced by solar flares. It does this by monitoring transmissions from Earth-based beacons, which are affected by changes in the ionosphere, giving an indirect indication of events on the Sun. The main motivation for this work is to correlate these radio observations of solar activity with those from optical observers, and to follow the cycles of sunspots as they appear on the Sun.

The VLF Receiver User Manual explains how the unit may be connected to a data logger. Please visit www.ukraa.com for more information on how to share your observations with others.

Assembly Instructions

The following assembly instructions should be followed carefully. The VLF Receiver is not a simple electronic assembly, and is not regarded as a project suitable for a beginner. If you follow the instructions carefully you should be successful in building a VLF Receiver that works. If you are not careful, you run the risk of having a problem that may be very difficult to locate and fix later. If you are in doubt about your abilities, you are advised to seek out a local electronic hobbyist or Radio Amateur with constructional experience.

It is recommended that you read these instructions right through before commencing on the construction of the kit. You may need to prepare for some of the ideas suggested later in the text before you get too far into the construction. It is appreciated that experienced constructors will have developed their own techniques, but we suggest you think carefully before diverting from the sequence suggested.

Tools Required

- Wire cutters
- Needle-nose pliers
- Soldering iron (low-voltage (24V) iron preferred at least 15-25 watt, small tip, 25 to 40 watt is better)
- Light duty resin core solder 22SWG or 0.8mm
- Sponge

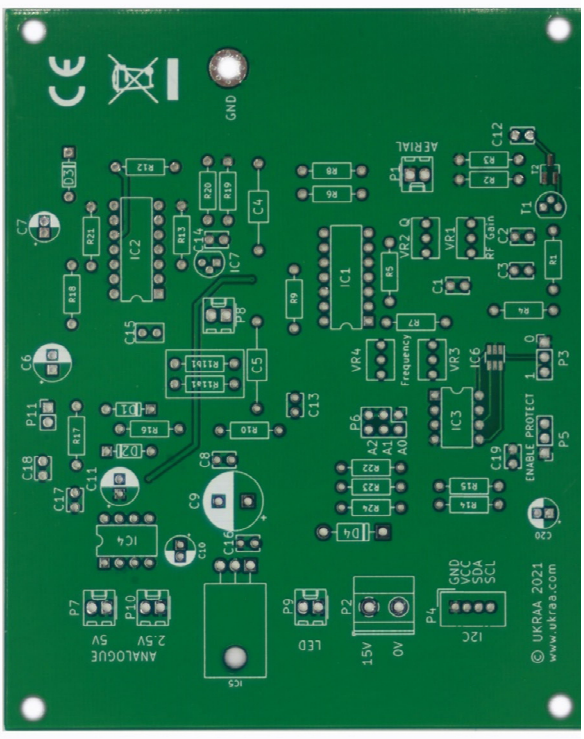
- Magnifying glass
- Solder sucker or solder wick

Kit Contents

- Printed Circuit Board (PCB)
- Bag A – Resistors
- Bag B – Capacitors
- Bag C – Semiconductors
- Bag D – Connectors, sockets and miscellaneous hardware

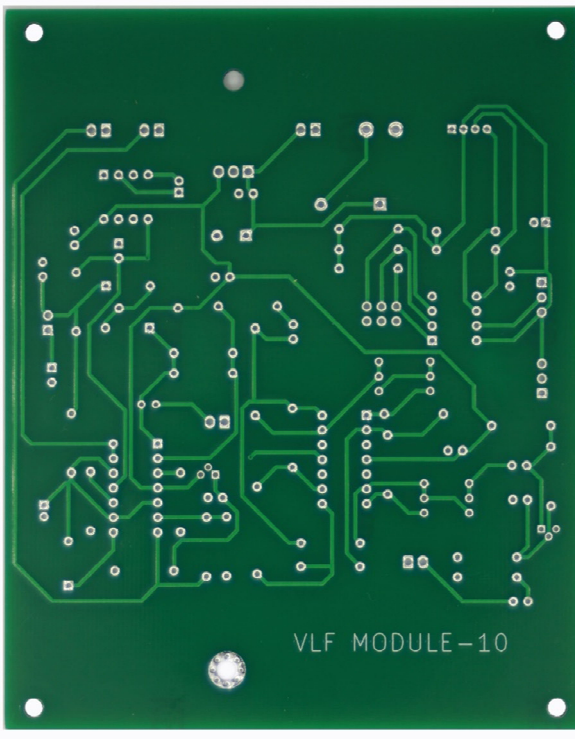
Printed Circuit Board (PCB)

The photographs below show revision 10 of the VLF Receiver circuit board. Please note that the colour of the circuit board supplied with the kit may vary from that illustrated.



Component Side

All components are inserted from this side and soldered on the other side.



Solder Side

Components are soldered on this side.

Assembly Sequence

The assembly instructions will follow the sequence below; full details are given in the later sections:

- Before starting work, inspect the components supplied and check against the list in Appendix 1.
- Install the Resistors and Capacitors *(Bags A and B)*.
- Install the dual in-line (DIL) sockets *(Bag D).*
- Install the Semiconductors *(Bag C).*
- Test the VLF Receiver.

The intention of the recommended assembly sequence is to install components in order of height above the board surface:

- Resistors, diodes and *non* electrolytic capacitors
- DIL IC sockets
- Polarised header plugs and pin clusters for jumpers
- Electrolytic capacitors
  (remember the polarity is important and the longer wire is the positive lead)
- The variable resistors (trim pots)
  (mount with the adjuster screws on the left hand side of the board, as shown in the illustration on page 4)
- Install the regulators and transistors last and then test before installing the ICs.

A detailed assembly sequence is listed in Appendix 2. We recommend that you print this out and tick off each stage as it is completed.

Pre-installed Components

The surface-mounted FET transistor (T2) and temperature sensor chip (IC6) have been pre-installed on the component side of the PCB.

Install the Resistors

Remove the fixed resistors from Bag A.

Fixed resistor

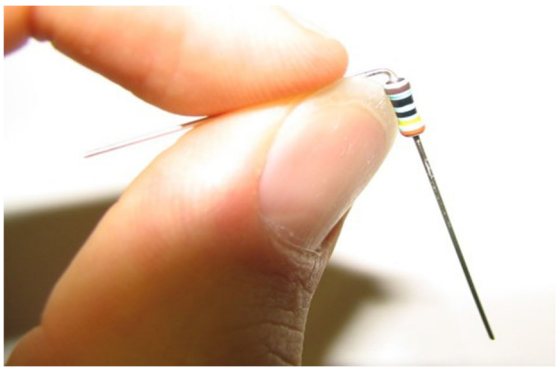
The resistors have the values written on each plastic bag. If in doubt the value of resistors is given by the colour-coded bands, see the section at the end if you are unfamiliar with resistor colour codes. The resistors supplied are all five-band, 1% tolerance.

It is suggested that the variable resistors ('trimpots') *(VR1...4)* should be installed later after the small ceramic capacitors have been soldered in. Resistor locations are marked on the component side of the PCB with the symbol R followed by a number: R1, R2 etc.

The following table indicates the values of the resistors:

Identification	Quantity	Resistance
R1, R2, R5, R6, R7, R8, R9, R10, R11a, R12, R16, R19, R20, R21, R22, R23, R24	17	10k
R3, R17	2	1M
R4	1	1k
R18	1	100k
R13	1	8.2k
R14, R15	2	47k

To prepare a resistor for insertion into the PCB, bend the two leads so that they form a right angle to the resistor body, the wire spacing allowed for insertion is 0.4in.



Bending resistor wires prior to insertion.

Do not bend the leads close to the body of any component because it may damage the lead anchorage.

To install, place the leads of the resistor through the appropriate holes and press the resistor down against the component side of the PCB. There is no required orientation for resistors - either lead may be inserted in either hole.

Turn the board over while holding the resistor in position and bend the leads slightly outward to hold the resistor in place and solder the leads to the circuit side. You may find it convenient to insert all the resistors with the same value first, and then move onto the next value. This avoids errors due to a wrong resistor being installed and also avoids having to continually refer to the colour code charts (in Appendix 2), which can be hard on the eyes, once the resistors are removed from the bandolier tape.

Several resistors may be loaded at the same time and retained by splaying the leads slightly where they come though the PCB. They can then all be soldered at the same time. See the section below describing how to make a simple assembly frame. It is suggested that you use the component list as a guide and "cross-off" the resistor as you locate its correct position and mount it. It may be useful to do the same on the layout diagram. It then soon

becomes clear which components you have yet to install, and makes locating their position much easier.

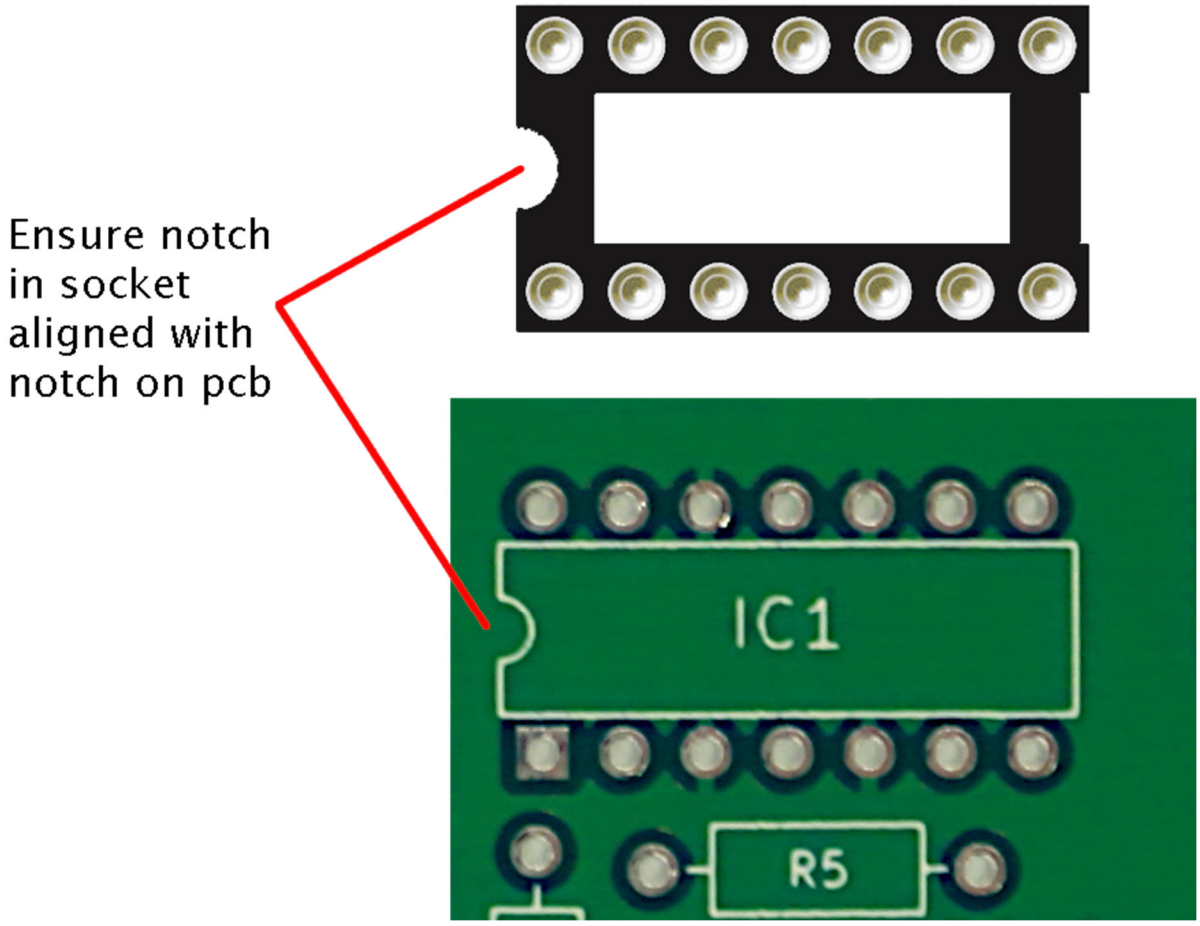
Resistor R11a (10k) is provided in the kit. This sets the receiver to unity gain (1X). You will note that there is provision on the circuit board for R11b which is not provided or installed. Removing R11a (snip the leads) and installing a 100k resistor in R11b will provide a gain of 10X.

Install the Sockets

It is suggested you next install the sockets (Bag D). Take care to install the IC sockets the right way round, since the orientation mark on the socket will serve as a reference to the correct orientation for the insertion of the IC. **Do not insert the ICs at this time.**

Socket locations are marked on the component side of the PCB with the symbol IC followed by a number: IC1, IC2 etc.

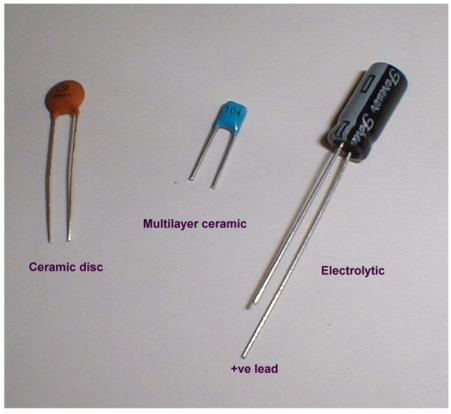
Identification	Quantity	Type
IC1, IC2	2	14 pin
IC3, IC4	2	8 Pin



Install the Capacitors

Remove the Capacitors from Bag B.

Identification	Quantity	Capacitance Value
C12	1	1nF Ceramic disc
C4, C5	2	1n5 1% Polystyrene
C3, C13	2	10nF Ceramic disc
C1, C2, C8, C14, C16, C17, C18, 19	8	100nF Ceramic multilayer
C15	1	10nF Polyester
C7*, C10*, C20*	3	10uF 63V Electrolytic * Shown as 16V on circuit diagram, rated voltage of component supplied may vary.
C6*, C11*	2	100uF 16V Electrolytic * As above.
C9	1	220uF 35V Electrolytic



A selection of Capacitors

Capacitor locations are marked on the component side of the PCB with the symbol C followed by a number: C1, C2 etc.

The 1nF disc capacitors are labelled '102', the 10nF labelled '103', and the 100nF labelled '104'. You may need the magnifying glass to read these numbers!

Install the ceramic, polystyrene and polyester capacitors using the same technique as for the resistors. It is easiest to install the shorter components first, leaving the taller components until later in the sequence. Thus fit the electrolytic capacitors last. Once again it is suggested that one type and value is selected and all these are installed, before moving on to a different value.

The electrolytic capacitors have a polarity that must be observed. The PCB is marked with a "+" to indicate the correct orientation. Where there is no fixed orientation it is usual to choose an orientation for the capacitors that allows their value to be easily read when installed.

Now install the taller electrolytic capacitors, observing the polarity marking on the board. The longer of the two leads is the positive lead, but there is also a grey band up the sleeve with a "−" on the side nearest to the negative lead.

Miscellaneous

Variable Resistors

Variable resistor (multi-turn)

The components supplied do not have offset wires so can be installed in reverse if care is not taken. This will affect the tuning direction of the adjustment screw. **NB: Make sure that the adjusting screw is located so it is towards the left of the board (the Grove connector P4) so that it is in the**

correct orientation. This provides a "natural adjustment" where clockwise turning of the screw gives increasing frequency, increasing Q and increasing RF Gain.

Variable resistor locations are marked on the component side of the PCB with the symbol VR followed by a number: VR1, VR2 etc.

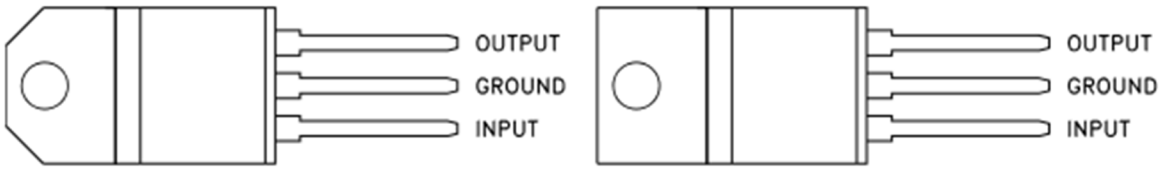
Identification	Quantity	Resistance
VR1, VR2, VR3	3	10k
VR4	1	500Ω

Semiconductors

Remove the Semiconductors from Bag C.

The orientation of the semiconductors must be correct for the receiver to function.

Be careful with the orientation of IC5; the device lays flat on the PCB. The longer flat surface is the back and goes flush with the pcb. The thicker end where the leads project faces upwards away from the pcb. Bend the leads just below the body of the device where they narrow. Hold all three leads in a pair of needle nosed pliers and gently bend so they are at 90 degrees to the body.



Front view of IC5

Insert the M3 screw through the hole in IC5 and the corresponding hole in the board before soldering the leads. The leads may need to be adjusted to get a good alignment between the device and the hole.

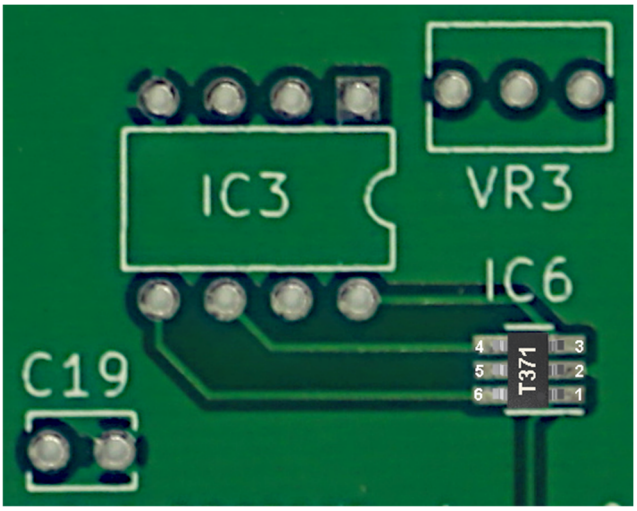
Similarly, ensure that the flats on IC7 and on T1 align with the screen-print outline.

When installing diodes and transistors it is essential that the correct orientation be followed. The diodes with their axial leads look very like resistors but the band on one end of the body indicates the negative (cathode) end and must align with the screen-print on the PCB. Where glass-body diodes have been provided, do not bend the leads whilst holding the diode body. The lead should be held with a pair of fine-nose pliers close to the body to prevent the stress of the bending fracturing the glass.

Identification	Quantity	
IC1	1	TL084 amplifier
IC2	1	LM324N amplifier
IC3	1	Socket only
IC4	1	ICL7662 regulator
IC5	1	+12V regulator, 7812
IC6	1	*LM73 Temperature Sensor (pre-installed)*
IC7	1	+5V regulator, 78L05
T1	1	BC547 transistor
T2	1	*MMBF5484 FET (pre-installed)*
D1, D2, D3	3	1N914 diode
D4	1	1N4002 diode

Semiconductor locations are marked on the component side of the PCB with the symbols IC, T or D for the various components followed by a number.

The image below shows the orientation of IC6, the LM73 temperature sensor, for your reference. Pin 1 is underneath the 'T' of the underlined legend 'T731'.

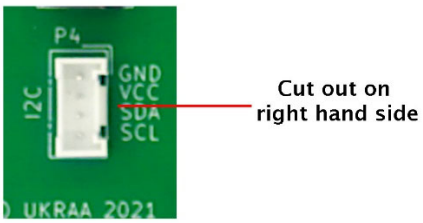


Orientation of LM73

Hardware

Some of the header plugs on the board are polarised and the screen print indicates the position of the raised polarising peg.

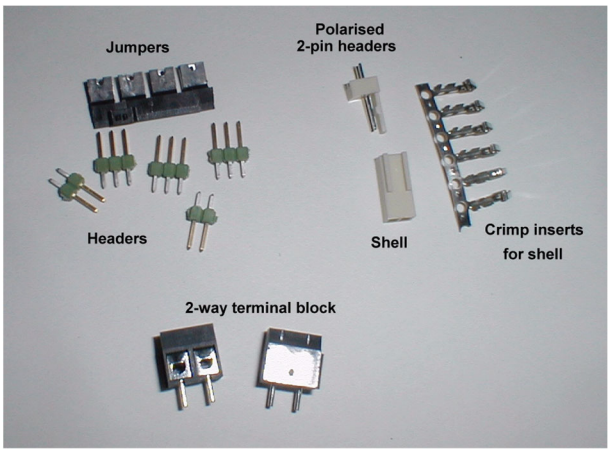
The Grove connector P4 must be installed with the correct polarity. The gap in the housing is on the right hand side facing into the pcb.



Three pairs of crimp terminals are provided for wiring the connectors to the polarised (white) two-pin header plugs. Bare some insulated connecting wire and gently close the tangs on the wire with the pliers. The bared wire length should be cropped so that the outer tangs grip the insulation whilst the ones nearest the socket grip the wire. The end of the wire should not protrude more than 1mm beyond the lower crimp or it will impede the insertion of the header pin. Since it is unlikely you will have the correct crimp tool you should run a small amount of solder into the wire crimp to ensure a good connection. The crimp-socket is then inserted into the shell, but you need to be careful to align the tongue with the slot in the moulded shell. When aligned correctly, push the crimp into the shell until the tongue engages and the wire cannot be pulled out. A damaged wire may be replaced by gently

pushing the tongue in through the slot in the shell, whilst applying a gentle pull to the wire. You may require a new crimp, though with care they can be re-soldered.

The interconnections are covered in the main Manual.



A selection of Hardware

Identification	Quantity	
IC1, IC2	2	14 pin socket
IC3, IC4	2	8 pin socket
P1, P7, P8, P9, P10	5	2-way plug & socket (pairs)
P2	1	2-way terminal block
P5, P12	2	3-way header & links
P11	1	2-way header & link
P6	1	2 x 3-way header & link
P4	1	Grove Connector

Testing

Visual Inspection

- Start with a completed PCB with no ICs installed.
- Check that the appearance of all soldered joints is satisfactory, and there are no wires or pins that have been missed. A correctly made joint should reveal the end of the though-board wire or pin. This must

not be hidden inside a spherical blob of solder. Check for a wetting meniscus on the sides of the through-wire (see diagram).

- Check all electrolytic capacitors and diodes are inserted the right way round. The grey band with a black "-" denotes the negative lead. The band on the diodes should match those shown on the layout diagram.
- Check all the IC sockets are oriented correctly, with the indicator slot in the correct orientation (see the layout diagram).
- Check the orientation of transistor T1 and IC7 (note the position of the "flat" on the moulding).
- Place a jumper on the "PROTECT" side of header P5.
- Place jumpers on the 2 positions on P6 furthest away from IC3 to set A1 and A2 to "0", which provides an address of "1" for the EEPROM.
- Place a jumper on the middle pin of P12 not connected to either of the other P12 pins.
- Place a jumper on P11.

Resistance Tests

It is suggested that any general purpose Digital Multimeter (DMM) can be used for the following tests. Your positive probe should be on the screw nearest to IC5, to ensure the electrolytic capacitor charges with the correct polarity.

- Measure the resistance between the wire-clamping screws of P2. The purpose of this test is to ensure that there is not a dead short on the power rail, which might damage the power supply. The initial reading should be high (10kOhm – several MOhm, depending on the driving voltage of the test meter) and should drift slowly upwards as the large electrolytic capacitor charges up. If the reading is below 5kOhm repeat the visual inspection looking for stray wires or solder splashes between pads. If the reading shows an open circuit, that may be because your DMM uses a very low voltage which may not overcome the diode

forward voltage drop.

Power-up Tests

- Assuming the results of the above tests are satisfactory, insert the ICs taking care to orientate them correctly with regard to the indent on the socket. Check carefully down the side of the IC sockets (magnifying glass) after insertion to ensure that all the IC pins have entered the socket and there are none bent up underneath the package.

- Arrange a 15 Volt DC power source and connect to P2, but connecting the DMM in series with the positive lead to the positive terminal of P2, and select direct current measurement on the 200mA range on the DMM. Apply power and check that the current draw is around 30mA if the LED is not installed and around 40mA if the LED is installed. If the reading is above 50mA or very low remove the power and return to visual inspection of the PCB.

- If the current draw is correct remove the power and disconnect the DMM. Ensure that VR4 is wound fully anti-clockwise (wind 30 turns to be sure, there is no end-stop) and/or check the resistance with a DMM - it should be about 500 Ohms. Re-connect the DMM to read the output level on P10. Place a short across the aerial terminals P1. Re-connect the power leads direct to P2 and apply power. The output level should read:

  - about 0.2 to 0.25 Volts on P10 (2.5 Volt range);
  - about 0.4 to 0.5 Volts on P7 (5 Volt range).

  If the reading is higher or much lower there may be a problem, in which case remove the power and repeat the preliminary tests before proceeding with detailed fault-finding.

Proceed to the "Tuning" stage as described in the VLF Receiver User Manual. This becomes part of the testing because successful tuning indicates correct operation of the unit.

Soldering Guide

Technique

IMPORTANT:

DO NOT leave a soldering iron plugged in for a long time without using it.

NEVER leave a hot soldering iron unattended.

- Clean the tip of the iron by wiping it on the damp sponge. It should be "tinned" with a very thin coating of solder. The sponge will remove excess and any "slag".

- Place a small amount of solder on the tip, this should NOT form a big blob and is NOT for making the joint but will help to convey the heat of the iron to the joint.

- Hold soldering iron firmly down against the copper track and against the side of the component lead so that the track and component lead are BOTH heated (about two seconds).

- After the lead and the contact have heated, apply the solder to the lead and copper track (NOT directly to the iron) until the solder melts and flows around the lead and into the contact.

- Remove the iron. This should leave a gently dished meniscus like water in a glass showing that the solder has "wetted" the joint and thus made good electrical contact. A lumpy or "crystalline" look to the joint suggests that the joint is "dry" and will not make a reliable electrical contact. In such case it should be carefully reheated with the iron and perhaps a very small about of solder until the required meniscus is formed. The inability to form this meniscus easily suggests that the iron may not be hot enough, or possibly is not "tinned" properly. Do not apply so much solder that it runs across the PCB surface.

- Allow the joint to cool and inspect the joint.

- Trim the excess lead wire just above the solder joint.

- If too much solder has been fed into the joint, and it resembles a blob, the excess may be removed with the solder-wick or solder-sucker. The joint can then be inspected for correct wetting under a magnifying glass.

Some Further Advice

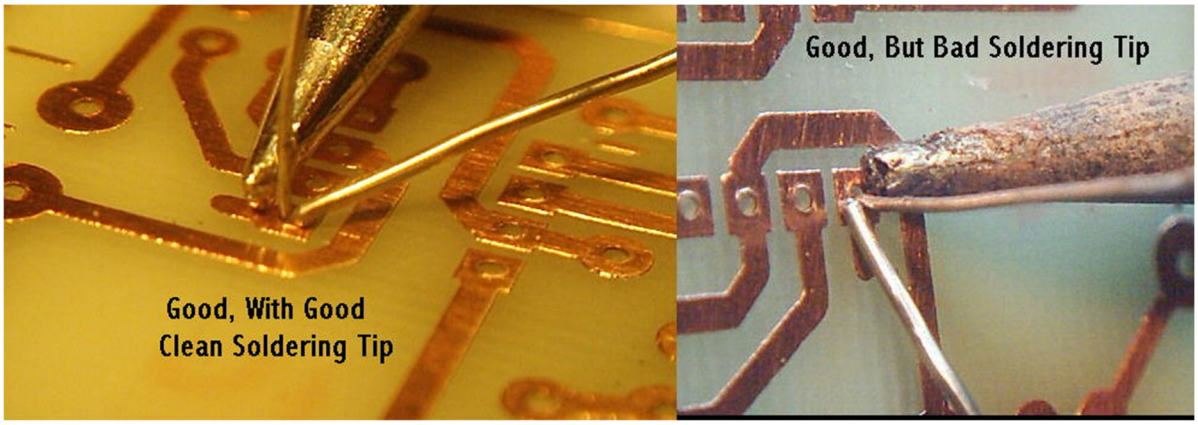
It is recommended that you install the resistors first. The easiest way is to take a bag containing resistors with the same value (you may wish to check the colour code against the table at the rear of the Manual), select one and identify on the PCB the position corresponding to the first resistor identity on the bag label e.g. R1 and, after dressing the leads, install in the appropriate holes. Be careful to correctly identify the resistor values.

Multi-pin components can easily not be fitted snug to the board, because they have a tendency to slip out slightly when the board is turned over for soldering. If you do not use a jig (see later section for a description of a simple aid), solder just one pin then inspect the top side of the board to check that the component is fully home against the board surface. The polarised header plugs are easy to mount slightly skewed, but correct neat assembly will enhance your satisfaction with the finished unit. If you use the foam aid to hold the components, it is suggested that you place a jumper on the unpolarised jumper pins or they just perforate the foam and are not held tight.

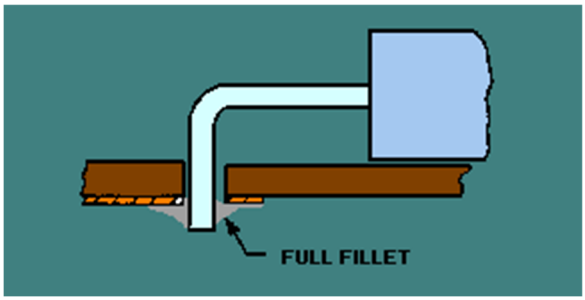
It is possible that some components may be supplied with slightly different lead spacing specification. The lead spacing can be altered carefully with a pair of narrow-nose pliers. Try not to bend the lead too close to the component body because it is possible to damage the component. The glass encapsulated diodes are the most sensitive to this form of damage. The wires should not be bent closer than about 2mm to the body of the component to prevent fracturing the lead-to-glass seal.

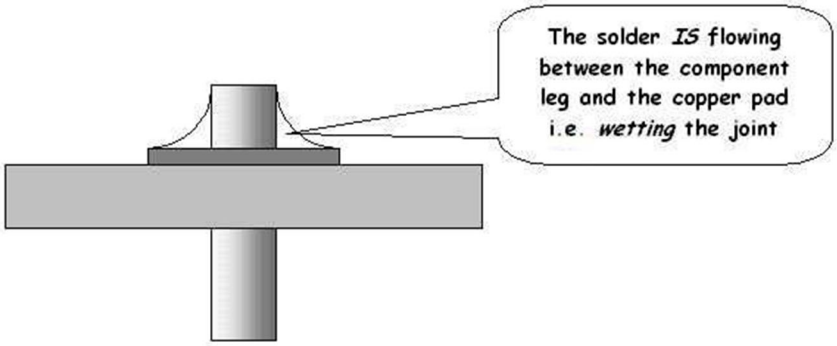
Examples of Soldered Joints

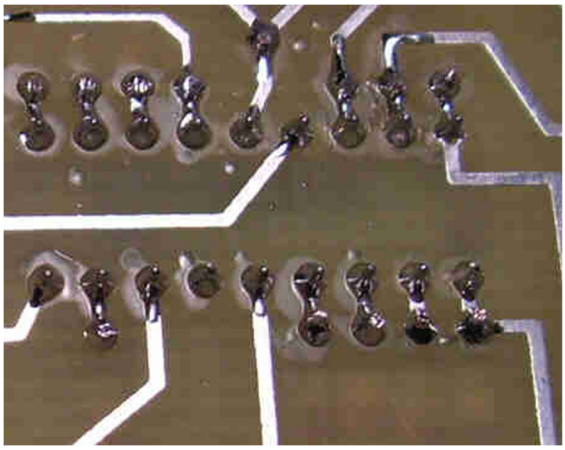
Making good soldered joints is vital to the correct operation of this unit. The pictures below provide a guide to help you inspect your work and correctly assess the solder quality.



The joint should look like this:

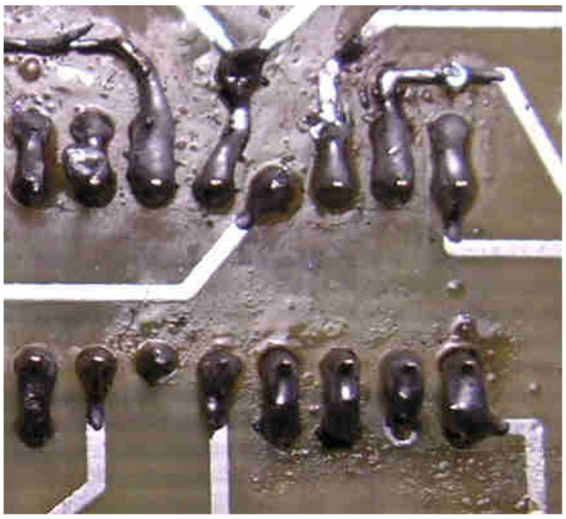


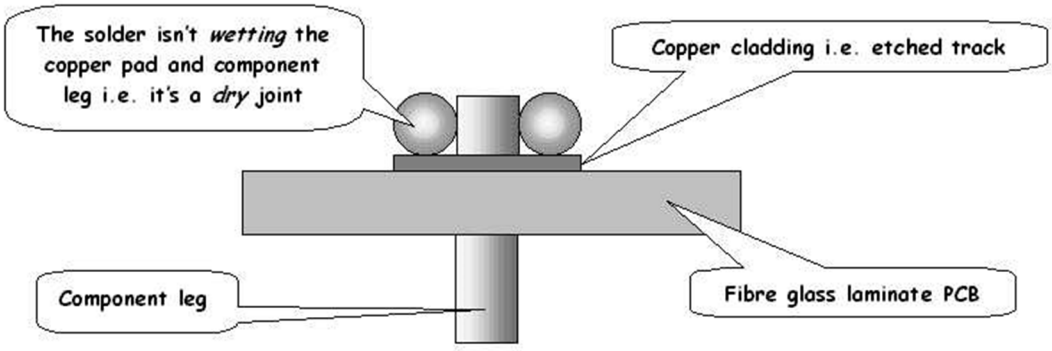




A good soldered joint

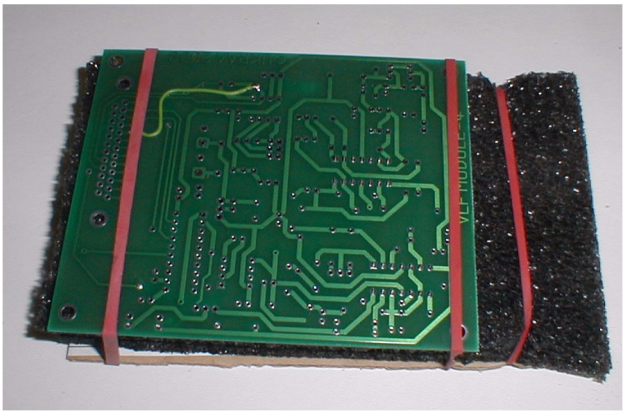
A bad joint looks like this:



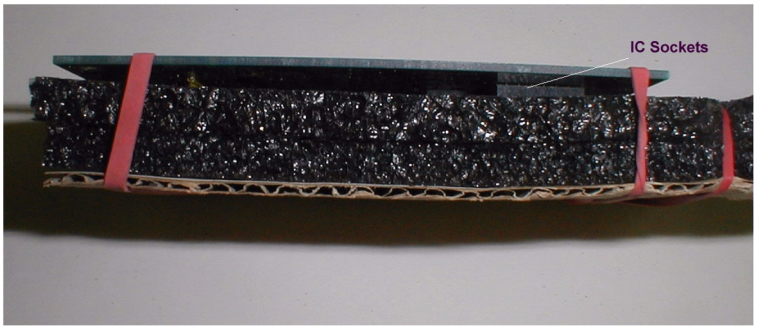


Simple Assembly Frame

The insertion of wire-ended components is fairly straight-forward in that they can be retained in position, whilst turning the board for soldering, by splaying the excess leads out. Other components, like headers, and sockets, with more solid connections are more difficult. Professional assemblers use a frame, which allows all the components to be inserted and held on position against the PCB by a foam block whilst the frame is rotated to offer up the solder side of the board. A very simple version of this may be easily made with some foam, a piece of thick cardboard, or hardboard, and a couple of elastic bands. The only caution is that the pins of the headers may get hot during soldering and some foam may melt at quite low temperatures. The easy way to avoid problems like this is to insert the jumper on the component side of the header pins before strapping the foam over them. Using this method you can insert several components and solder them in position secure in the knowledge that they will stay properly seated against the board. Just be careful not to touch the elastic bands with the soldering iron or everything may fly apart!



Soldering Assembly Frame



Soldering Assembly Frame (side view)

Cleaning the Circuit Board

You may wish to remove any solder flux residues left after soldering. There are products available for this purpose, as liquids or aerosols. Depending on the solder you have used, it may also be possible to use water as the solvent, alternatively cellulose thinners can be used for a faster effect. Scrubbing with a stiff brush (even an old toothbrush) may be all that is required.

Correcting Mistakes

If you have to remove a faulty or misplaced through-hole component, remember the PCB is worth more than the component. Chop up the component usually through the body, to leave as long a lead as possible and enable you to remove the leads individually. This can be done usually without getting the board too hot for the through-hole to be damaged. However, this often leaves you with a solder filled hole. There are all kinds of solder suckers, but these can be difficult to use particularly on ground-plane holes and sometimes difficult to keep tinned and keep the joint hot enough. The following technique can be successful if done carefully.

Work on the component side of the board. Since you pulled the legs out that way you should have a dome of solder on that side whereas the solder surface in the hole on the other side may be below the board surface. A fairly large chisel bit is best for the following operation rather than a conical point.

Take some solder-braid (use good quality Multicore) if you are right-handed, place the reel or the slack on the right of the PCB and hold the end in your left hand. Place the last used part of the braid, yes that bit filled with solder, over the hole to be cleared, fill the soldering iron bit with a blob of new solder and place it on top of the braid and over the hole. When you see the solder start to flow pull the braid away across the hole under the iron keeping the iron over the hole. The solder should "wick out" of the hole and fill the braid as you move it away. Stop pulling when no more solder enters the braid and quickly remove the iron. When you remove the braid and iron you should now see a clear hole.

The secret is to use plenty of new solder to engulf the old solder in a pool that you then draw into the new wick. The new wick appearing over the hole tends to keep it cooler as well. If you work from the component side of the board there is less chance of damaging tracks if you do get things too hot. If it doesn't work first time allow the hole to cool for a bit before trying again. Be careful when using the hole again and inserting a component - remember it is slightly smaller than original because there will be a thin film of solder over the inside. If you do damage the pad on the component side it does not matter too much because the connection will be made on the solder side of the board and it can easily be patched if necessary.

Appendix 1 - Component List

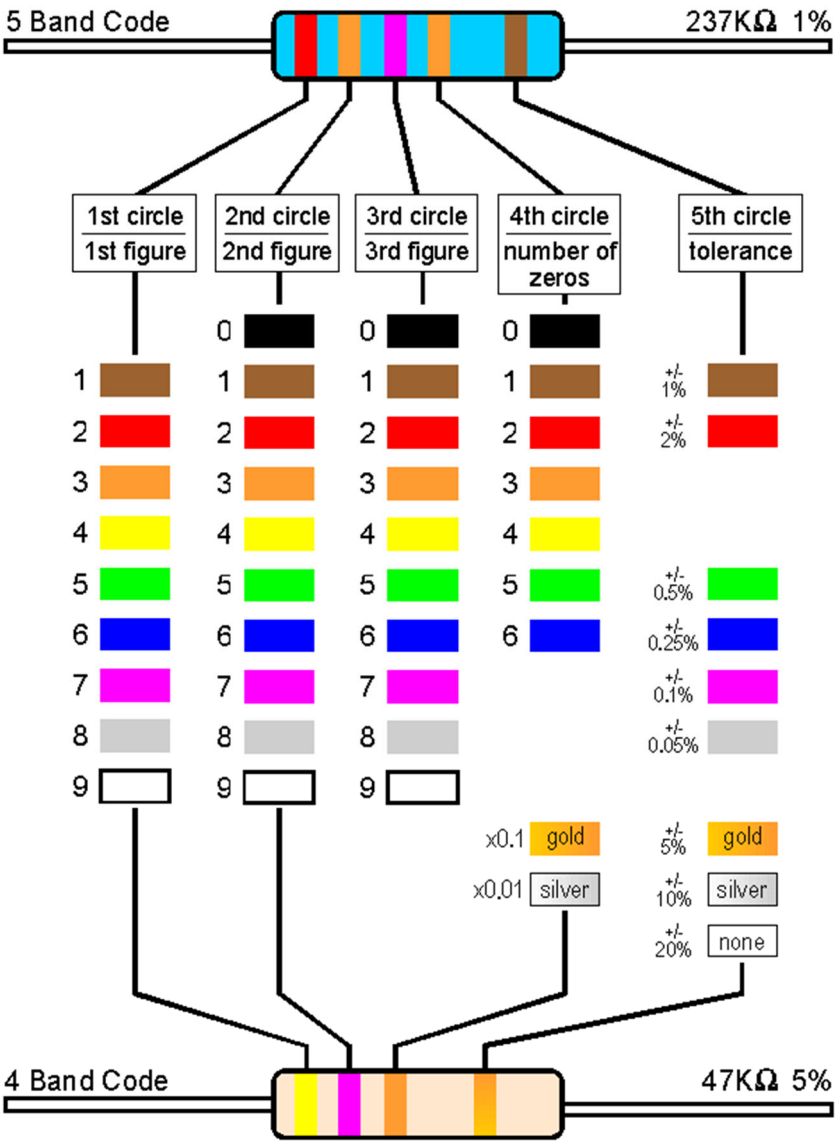
Identification	Qty	Component
Bag A - Resistors		
R1, R2, R5, R6, R7, R8, R9, R10, R11a, R12, R16, R19, R20, R21, R22, R23, R24	17	10k
R3, R17	2	1M
R4	1	1k
R18	1	100k
R13	1	8.2k
R14, R15	2	47k
VR1, VR2, VR3	3	10k 30 turn variable resistor
VR4	1	500R 30 turn variable resistor
Bag B - Capacitors		
C12	1	1nF Ceramic disc
C4, C5	2	1n5 1% Polystyrene
C3, C13	2	10nF Ceramic disc
C1, C2, C8, C14, C16, C17, C18, C19	8	100nF Ceramic multilayer
C15	1	10nF Polyester
C7, C10, C20	3	10uF 63*V Electrolytic * Shown as 16V on circuit diagram, rated voltage of component supplied may vary.
C6, C11	2	100uF 16V Electrolytic
C9	1	220uF 35V Electrolytic

Bag C – Semiconductors		
IC1	1	TL084 amplifier
IC2	1	LM324N amplifier
IC4	1	ICL7662 negative voltage converter
IC5	1	+12V regulator, 7812
IC6	1	*LM73 Temperature Sensor (pre-installed)*
IC7	1	+5V regulator, 78L05
T1	1	BC547 transistor
T2	1	*MMBF5484 FET (pre-installed)*
D1, D2, D3	3	1N914 diode
D4	1	1N4002 diode
Bag D – Connectors		
IC1, IC2	2	14 pin socket
IC3, IC4	2	8 pin socket
P1, P7, P8, P9, P10	5	2-way plug & socket (pairs)
P2	1	2-way terminal block
P5, P12	2	3-way header & links
P11	1	2-way header & link
P6	1	2 x 3-way header & link
P4	1	Grove Connector
M3 Screws, Feet & nut.		

Appendix 2 Recommended Assembly Order

Identification	Quantity	Item	Check
D1, D2, D3	3	1N914 diode	
R1, R2, R5 – R10, R11a, R12, R16, R19, R20, R21, R22, R23, R24	17	10k	
R3, R17	2	1M	
R4	1	1k	
R18	1	100k	
R13	1	8.2k	
R14, R15	2	47k	
D4	1	1N4002 diode	
C1, C2, C8, C14, C16, C17, C18, C19	8	100nF Ceramic multilayer	
C12	1	1nF Ceramic disc	
C3, C13	2	10nF Ceramic disc	
IC1, IC2	2	14 pin socket	
IC3, IC4	2	8 pin socket	
C4, C5	2	1n5 1% Polystyrene	
IC7	1	+5V regulator, 78L05	
T1	1	BC547 transistor	
P4	1	Grove Connector	
IC5	1	+12V regulator, 7812	
P2	1	2 way terminal block	
P3, P5	2	3 way header & links	
P11	1	2 way header & link	
P6	1	2 x 3 way header & link	
C15	1	10nF Polyester	
P1, P7, P8, P9, P10	5	2 way plug & socket (pairs)	
VR1, VR2, VR3	3	10k variable resistor	
VR4	1	500Ω variable resistor	
C7, C10, C20	3	10uF 63V Electrolytic	
C6, C11	2	100uF 16V Electrolytic	
C9	1	220uF 35V Electrolytic	
Feet	4	Feet	
Tested Label	1	Label	

Appendix 3 – Resistor Colour Codes



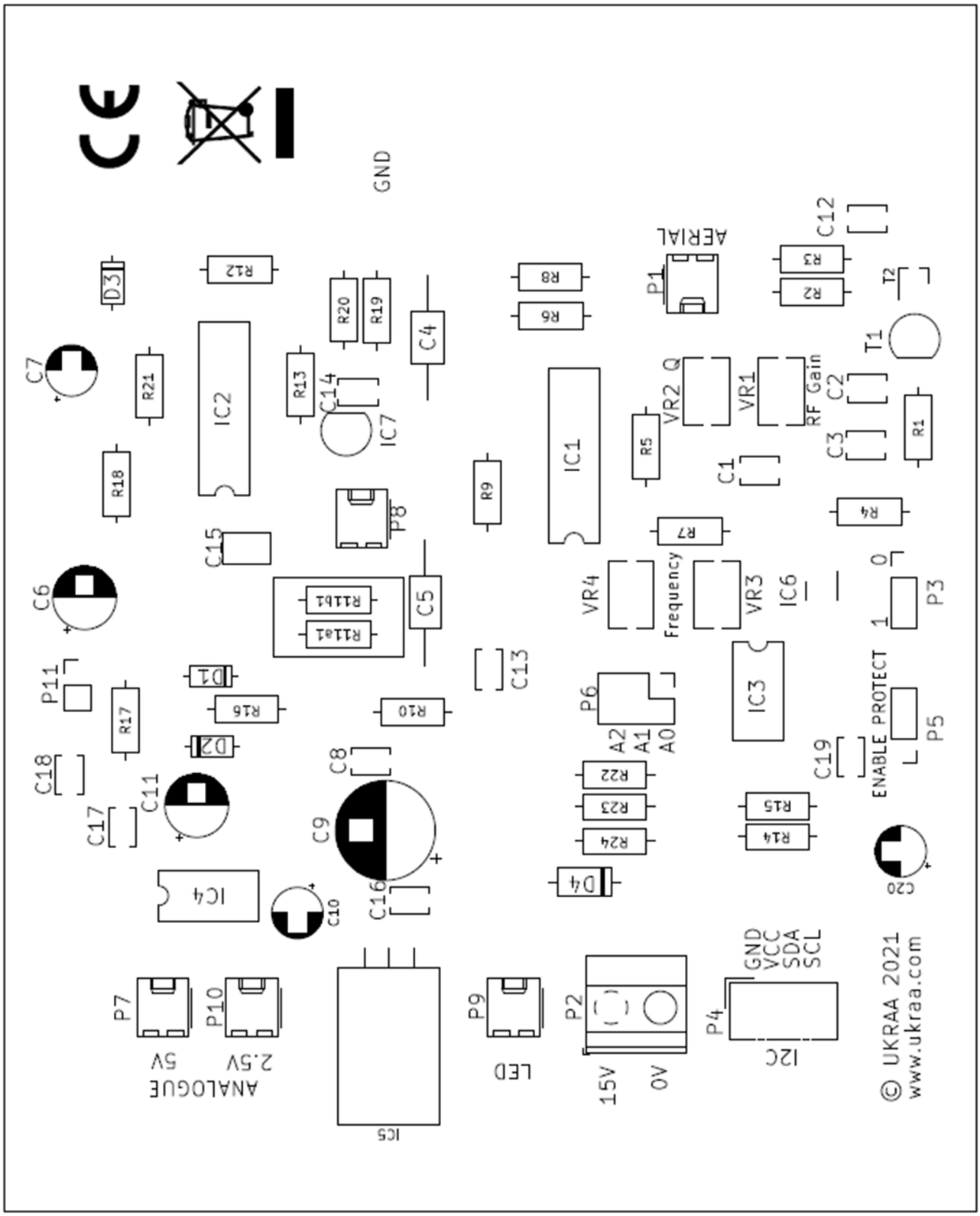
Resistor Colour Code Chart

The following table indicates the values of the resistors:

Identification	Quantity	Resistance	Colour Code First 4 colours
R1, R2, R5, R6, R7, R8, R9, R10, R11a, R12, R16, R19, R20, R21, R22, R23, R24	17	10k	Brown, Black, Black, Red
R3, R17	2	1M	Brown, Black, Black, Yellow
R4	1	1k	Brown, Black, Black, Brown
R18	1	100k	Brown, Black, Black, Orange
R13	1	8.2k	Grey, Red, Black, Brown
R14, R15	2	47k	Yellow, Violet, Black, Red

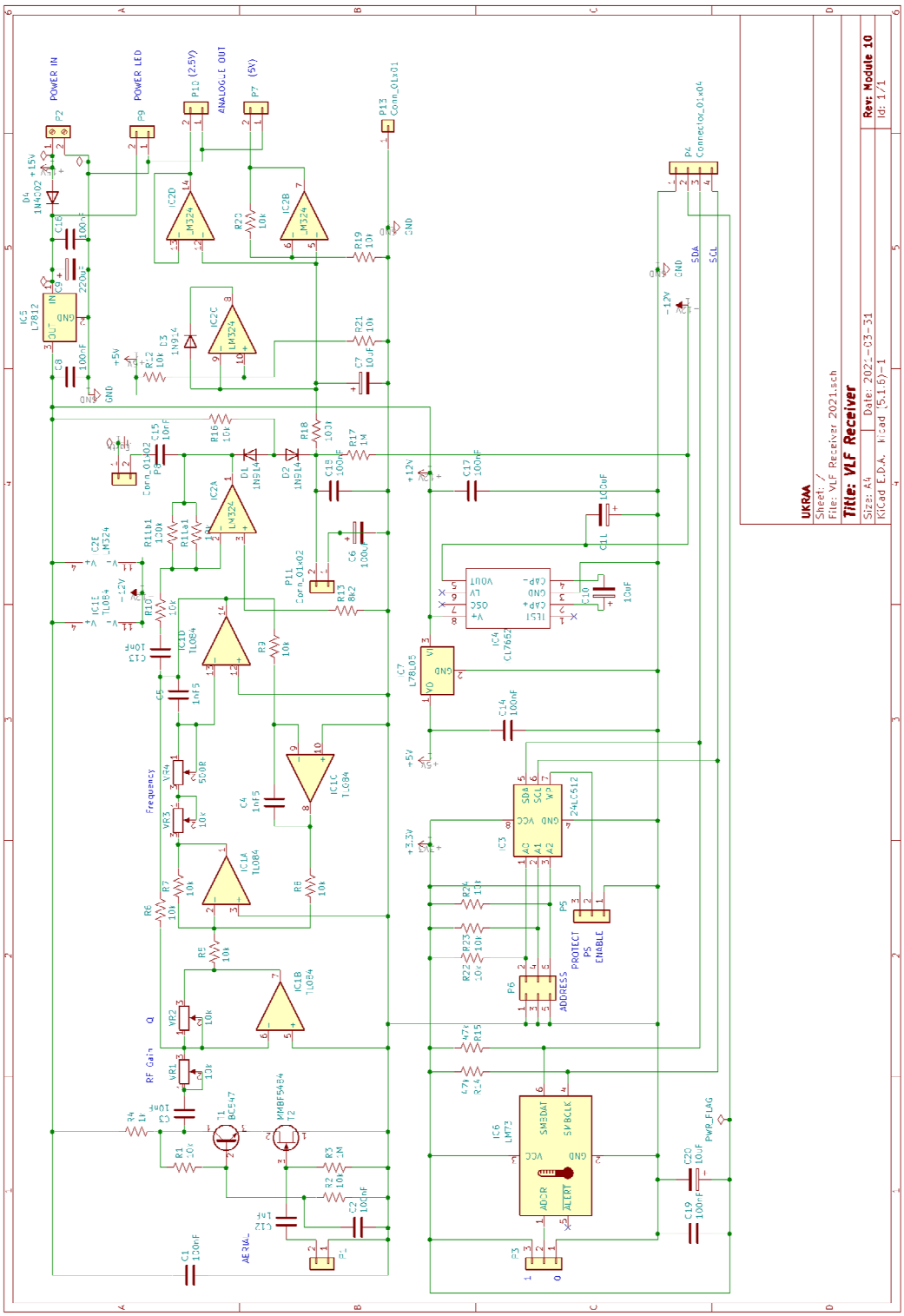
Note that the colours are always read with the first colour nearest the edge of the component as shown above. This is not easy to determine with the resistors supplied but they are all five-band 1% tolerance so will all have a brown ring at one end. Sometime it will be necessary to use the resistance range of your multi-meter to gain confidence you have the right value.

Appendix 4 – VLF Receiver PCB Layout



VLF Receiver Component Overlay

Appendix 5 – VLF Receiver Circuit Diagram



Appendix 6 – Regulatory Compliance

RoHS

The Directive on the restriction of the use of certain hazardous substances in electrical and electronic equipment 2002/95/EC, (commonly referred to as the Restriction of Hazardous Substances Directive or RoHS) was adopted in February 2003 by the European Union. The RoHS directive took effect on 2006 July 1, and is required to be enforced and become law in each member state. This directive restricts the use of six hazardous materials in the manufacture of various types of electronic and electrical equipment. In speech, RoHS is often spelled out, or pronounced "rosh".

The above paragraph was taken from the Wikipedia essay on RoHS.

The RoHS Directive restricts the use of the following six hazardous substances in electronic and electrical equipment products falling within the Directive:

1. Lead
2. Mercury
3. Cadmium
4. Hexavalent chromium
5. Polybrominated biphenyls
6. Polybrominated diphenyl ethers

UKRAA confirms that the suppliers of the components and materials used in the UKRAA VLF Receiver have stated that such components and materials are RoHS compliant and that reasonable steps have been taken to confirm these statements.

WEEE

RoHS is closely linked with the Waste Electrical and Electronic Equipment Directive (WEEE) 2002/96/EC that sets collection, recycling and recovery targets for electrical goods and is part of a legislative initiative to solve the problem of huge amounts of toxic e-waste.

The Waste Electrical and Electronic Equipment (WEEE) Directive is designed to ensure the efficient collection and recycling of electrical and electronic equipment at end-of-life. If a customer purchases a new product from UKRAA which falls within the WEEE Directive to replace an existing one (of similar function to the one that has been sold) and intends to dispose of the existing one, then the customer can request that we take back the existing product and deal with the costs and logistics of recycling it. Any customer wishing to take advantage of this facility should contact us. Provided that the existing product comes within the scope of the WEEE Directive, we will make arrangements for its return or collection and will deal with its disposal.

Contacts

The UK Radio Astronomy Association

Springfield
Rookery Hill
Ashtead
Surrey
KT21 1HY

A Charitable Incorporated Organisation.

Registered Charity in England and Wales No. 1123866.

E-mail: info@ukraa.com

Website: www.ukraa.com

Telephone: 01372 279066

BAA Radio Astronomy Group

Website: www.britastro.org/radio

Charitable Incorporated Organisation

Registered UK Charity No. 1123866