

# MST



## CONSTRUCTION MANUAL

 [www.ozQRP.com](http://www.ozQRP.com)

# CONTENTS

1	Introduction .....	4
2	Block Diagram .....	5
3	Circuit Description .....	6
3.1	SSB generator.....	6
3.2	Transmit mixer.....	6
3.3	Power amplifier .....	7
3.4	Receive mixer.....	7
3.5	Receive audio.....	8
4	Parts List.....	14
4.1	Main Board Parts.....	14
4.2	Band Specific Parts .....	16
4.3	Off Board Parts.....	16
5	Construction.....	17
5.1	General.....	17
5.2	Construction Steps.....	17
6	Enclosure.....	23
6.1	Rear Panel .....	23
6.2	Front Panel.....	23
6.3	Mounting the PCB.....	23
6.4	Front panel label.....	24
6.5	Fitting the DDS VFO.....	26
7	Wiring up.....	27
8	Testing and alignment .....	29
8.1	General.....	29
8.2	Power on.....	29
8.3	Receive.....	30
8.4	Transmit.....	30
8.5	Carrier frequency adjustment.....	32
9	Operation.....	33

# List of Figures

Figure 1 MST Block diagram.....	5
Figure 2 SSB generator.....	9
Figure 3 Transmit mixer.....	10
Figure 4 Power amplifier.....	11
Figure 5 Receive mixer.....	12
Figure 6 Receive audio.....	13
Figure 7 Component overlay.....	22
Figure 8 Front panel drilling guide.....	25
Figure 9 DDS VFO front panel mounting.....	26
Figure 10 Wiring diagram.....	28
Figure 11 Typical circuit voltages.....	31
Figure 12 Crystal filter response.....	32

# 1 INTRODUCTION

The MST (Minimalist Sideband Transceiver) is an easy to build and fun to use single sideband QRP transceiver for the 40M or 80M amateur bands.

The MST transceiver follows a minimalist design making it inexpensive and simple to build yet still delivering excellent performance and a pleasure to operate.

The MST PCB contains a complete SSB transceiver except for a VFO. You can add a VFO of your own, as long as it has the correct frequency range and drive level, but to obtain best results it should be used with the companion DDS VFO. The DDS VFO offers drift free performance and features a clear backlit LCD display, selectable frequency steps and programmable IF offset.

MST Features:

1. Complete SSB transceiver on a single PCB (just add a VFO).
2. Superhet receiver using a 4 pole 10MHz crystal filter.
3. 5W PEP minimum power output using a rugged power MOSFET output stage.
4. Unwanted sideband suppression is typically 40dB.
5. All spurious transmit outputs below -45dBc.
6. AF and microphone gain controls.
7. Easy to adjust and set up.
8. Front panel LED transmit power and modulation indicator.
9. Plenty of audio output to drive a loudspeaker.
10. High quality double sided PCB with groundplane, solder mask and silk screen.
11. Simple and easy to build using all through hole components.
12. No complicated coil winding required. Uses inexpensive commercial coil assemblies for tuned circuits.

A range of kits containing PCBs and hard to get parts for the MST and the DDS VFO are available from [www.ozQRP.com](http://www.ozQRP.com).

## 2 BLOCK DIAGRAM

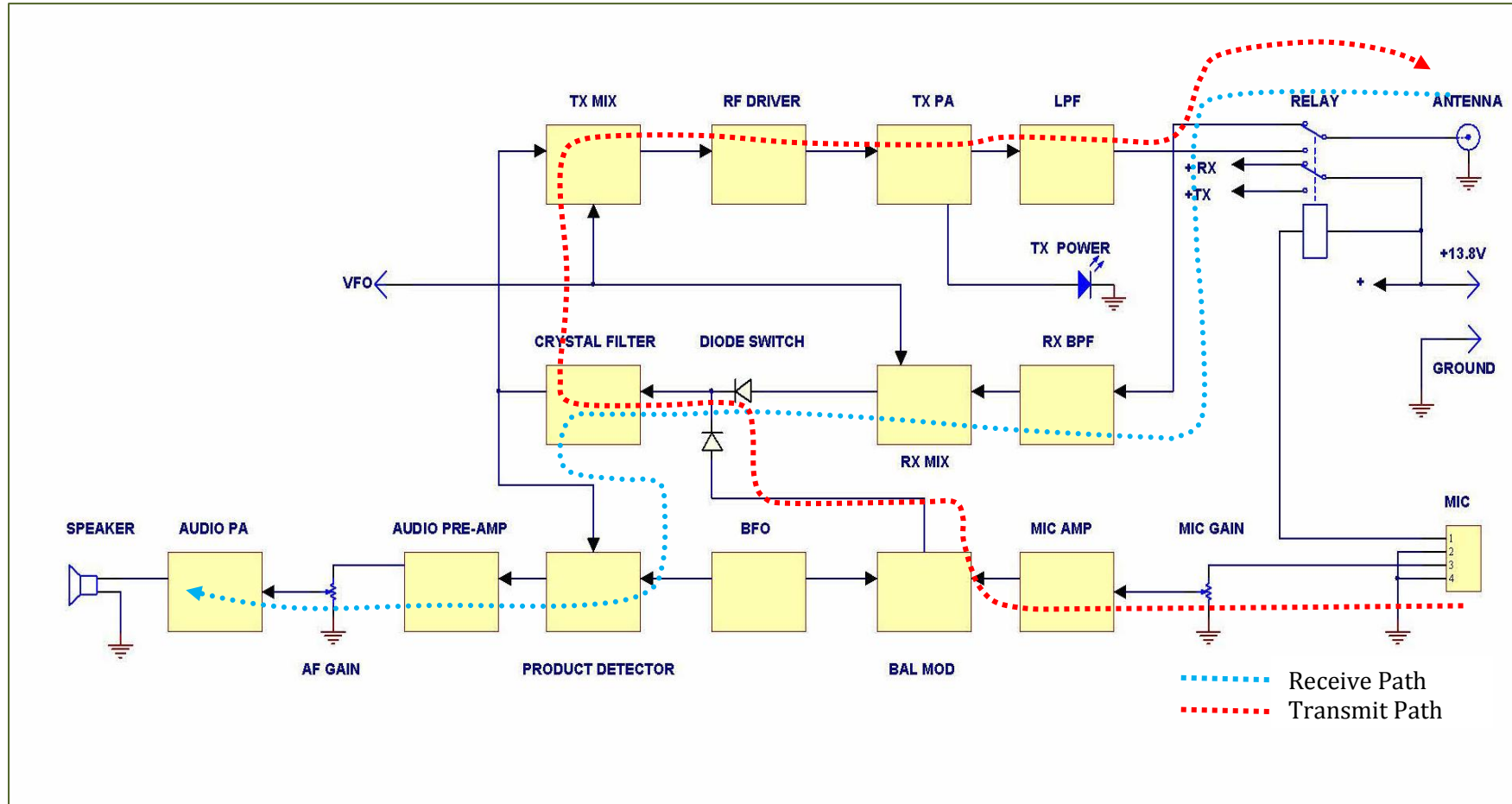


Figure 1 MST Block diagram

## 3 CIRCUIT DESCRIPTION

### 3.1 SSB GENERATOR

Transistor Q1 is configured as a Colpitts oscillator and acts as the 10MHz carrier oscillator in transmit and beat frequency oscillator (BFO) in receive. The frequency of crystal X1 is set by trimmer capacitor TC1 to a frequency slightly above the upper crystal frequency response. This results in LSB being generated. Capacitor C1 is included in the PCB pattern but not used in this application. The power supply to the oscillator is regulated with a 9.1V Zener diode ZD1.

The oscillator feeds buffer stage Q2 to provide a low impedance drive for the balanced modulator. A lower level output of around 500mV pk-pk is obtained at the junction of Q1 emitter resistors to feed the receive product detector.

Transistors Q3 and Q4 are configured as a shunt feedback audio amplifier which has stable gain and low output impedance. The low input impedance makes it ideally suited for low impedance microphones. C16 is included to prevent RF feeding into the amplifier while C19 provides high frequency roll off.

The balanced modulator is a diode switching type and doubly balanced. When the carrier signal is positive diodes D1 and D4 conduct and when it is negative diodes D2 and D3 conduct. The result is that no RF is present at the output transformer T1. If an audio signal is injected into the bridge the balance is upset and a double sideband suppressed carrier signal is produced at the output of T1. Note that capacitor C8 holds the junction of D1 and D2 at ground for RF.

Due to variations in component parameters the balance is not exact and so trimcap TC2 and trimpot VR1 are adjusted to bring the modulator into balance. In practice up to 50dB of carrier suppression can be achieved.

Diode D5 is used as an RF switch. With no DC current flowing the diode is a high impedance to RF. In transmit around 6mA of DC current flows through D5 and it becomes a low impedance path for RF. This feeds the output of the balanced modulator into the crystal filter. In receive D6 performs a similar function.

The crystal filter is a 10MHz 4 pole ladder type using closely matched crystals on the same frequency. Capacitors C11 to C15 are selected to provide a usable 2.5KHz bandwidth. Resistors R9 and R10 terminate the crystal filter in the correct resistance to give low ripple in the pass band.

### 3.2 TRANSMIT MIXER

The transmit mixer is based around U5 a SA612 balanced mixer. The 10MHz LSB signal from the crystal filter is fed single ended into pin1 while the other input on pin 2 is grounded to RF by C65. The VFO signal of around 300mV pk-pk is fed into pin 6. The balanced output of the mixer is fed to transformer T7 which is tuned to the difference of the VFO and carrier frequencies. The transformer used here is actually a 10.7MHz IF transformer with an integral 47pF capacitor. An external capacitor is added to lower the resonate frequency to match the transmit frequency.

A link coupled winding provides a low impedance output for the pre-driver built around transistor Q9. The collector load is another IF transformer (T8) and resonated to the transmit frequency in the same way as T7. In practice the two transformers form a 2 pole band pass filter and adjusted to give the desired passband.

### 3.3 POWER AMPLIFIER

Transmit signal from the pre-driver is applied to the driver stage built around transistor Q6. A BD139 works well here when biased with about 60mA of collector current. The design is well proven using both shunt and series feedback to provide low input and output impedance and good stable gain on the low HF bands.

The power amplifier is an IRF510 MOSFET and has been used in many designs. It is a good candidate for the HF bands and easily provides over 5 Watts PEP of power from a 13.8 V drain supply. The output from the driver is applied across resistor R38 and becomes the AC drive component for Q7 gate. Zener diode ZD4 and trimpot VR2 provides a stable and variable DC gate voltage to place Q7 just into conduction for linear service. There is a short ramp up of the gate voltage when switching to transmit as capacitor C54 charges and is included to provide a smooth gate voltage transition.

The drain load for Q7 is a broadband transformer T6. The turns ratio is chosen for best match and to provide maximum output into a 50 ohm load. The waveform from Q7 can be high in harmonics and so a 5 pole low pass filter is included to reduce the level of harmonic and other spurious energy to an acceptable level. The values of the low pass filter components are band dependant.

When the PTT is operated the transmit/receive relay is energized and the filtered transmit signal is passed to the antenna. When the PTT is not operated the relay switches the antenna through to the receive circuits. The relay also switches power to the transmit and receive sections as required.

As a visual indication of power output and modulation, the transmit signal is sampled, rectified and filtered by R39, D11 and C63. The resultant voltage is buffered by transistor Q8 to drive a front panel LED via current limiting resistor R41.

The power supply is made available at an auxiliary connector and is intended to power a companion VFO.

### 3.4 RECEIVE MIXER

Signals from the antenna are applied to a bandpass filter formed with two transformers T2, T3 and capacitors C21, C22 and C23. The antenna is link coupled to T2 while the output is fed from a tap on the tuned primary winding of T3 to provide proper impedance matching.

The mixer U1 is another SA612. The input is protected with a pair of back to back diodes and fed single ended into pin 1. Pin 2 is grounded to RF by C25. Zener diode ZD2 provides a stabilized 6.8 volt supply. VFO signal is injected into pin 6 at about 300mV pk-pk. The balanced output which is a difference signal at approximately 10MHz is fed to broadband transformer T4. The output of T4 is passed to the crystal filter when DC current flows through R19 and into D6.

### 3.5 RECEIVE AUDIO

The 10MHz SSB intermediate frequency (IF) signal from the crystal filter is applied to the product detector U2. The product detector is formed with another SA612 and mixes the IF signal with the 10MHz BFO signal to produce an audio output. The BFO is adjusted slightly above the crystal filter upper response so that Lower Side Band signals are detected correctly.

A low noise balanced input audio amplifier is formed with U3 a NE5534. A reference supply for the non-inverting input is obtained from the Zener diode stabilized supply for U2. The high frequency response of U3 is limited by C35 and C42, while capacitors C37 and C36 reduce the low frequency response.

The output of U3 is fed to a 1uF coupling capacitor and 1K resistor (R28) to the AF gain potentiometer. When switching to transmit mode MOSFET Q5 is turned on rapidly through D9, charging C45, and shorting the audio line to ground. When returning to receive mode Q5 slowly turns off as C45 discharges via resistor R29. This circuit produces a very nice action eliminating unwanted audio clicks and pops when switching between receive and transmit modes.

Audio from the AF gain control is amplified by U4, an LM386, to drive a loudspeaker. U4 is configured for extra gain by including C41 and R26. C40 guards against RF entering the amplifier input, while R30 and C47 help stabilize the output.



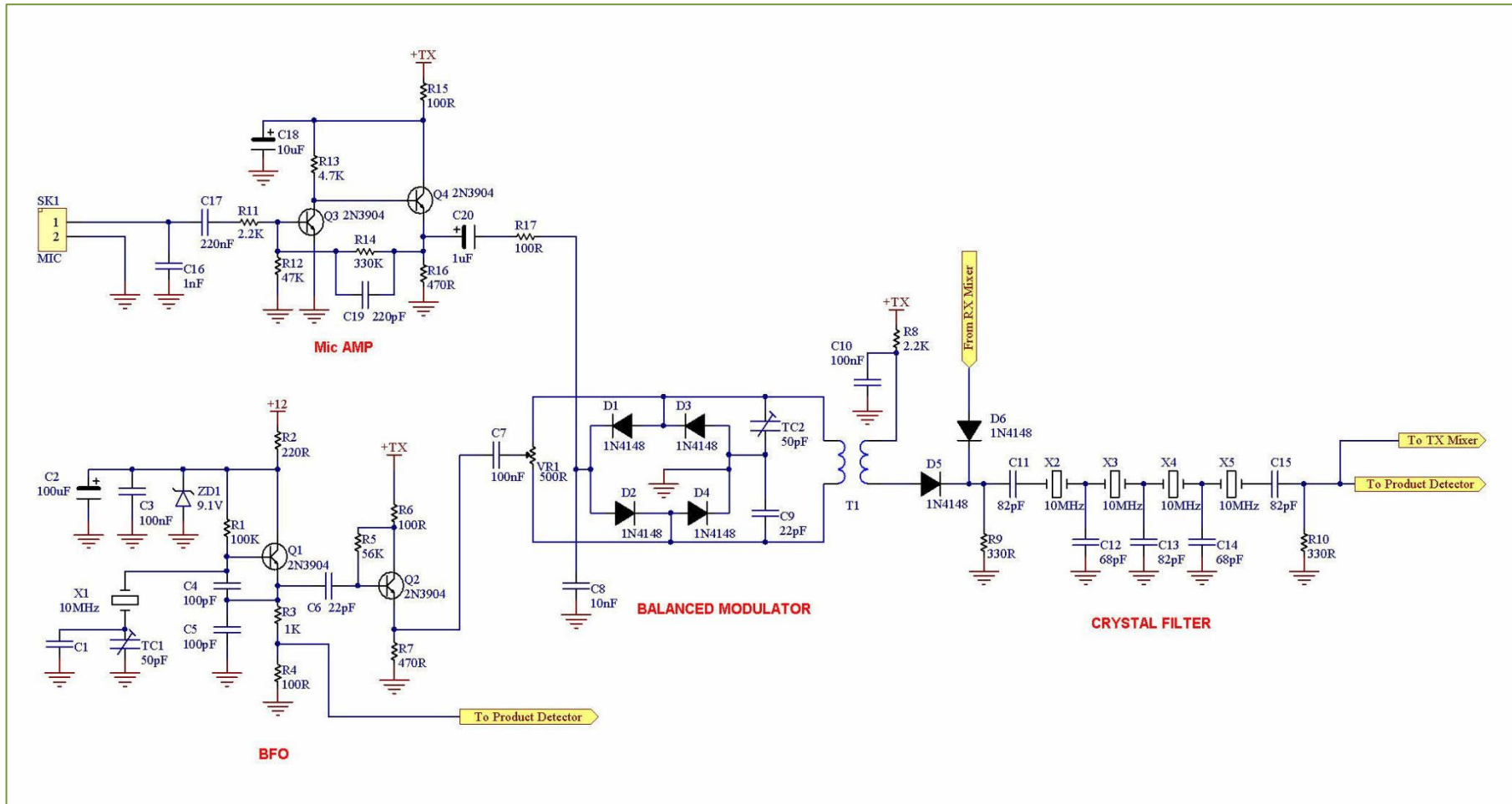


Figure 2 SSB generator

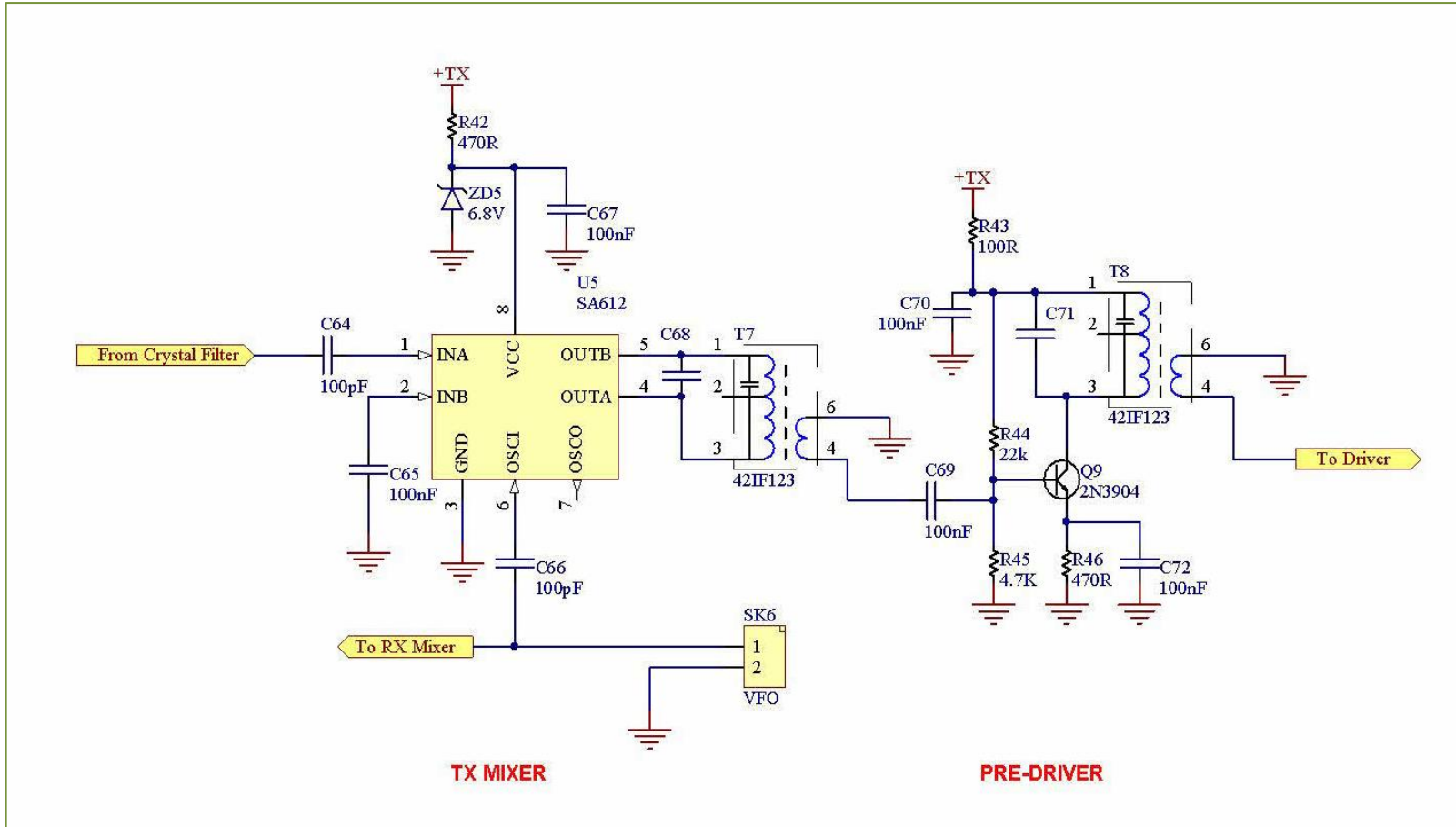


Figure 3 Transmit mixer

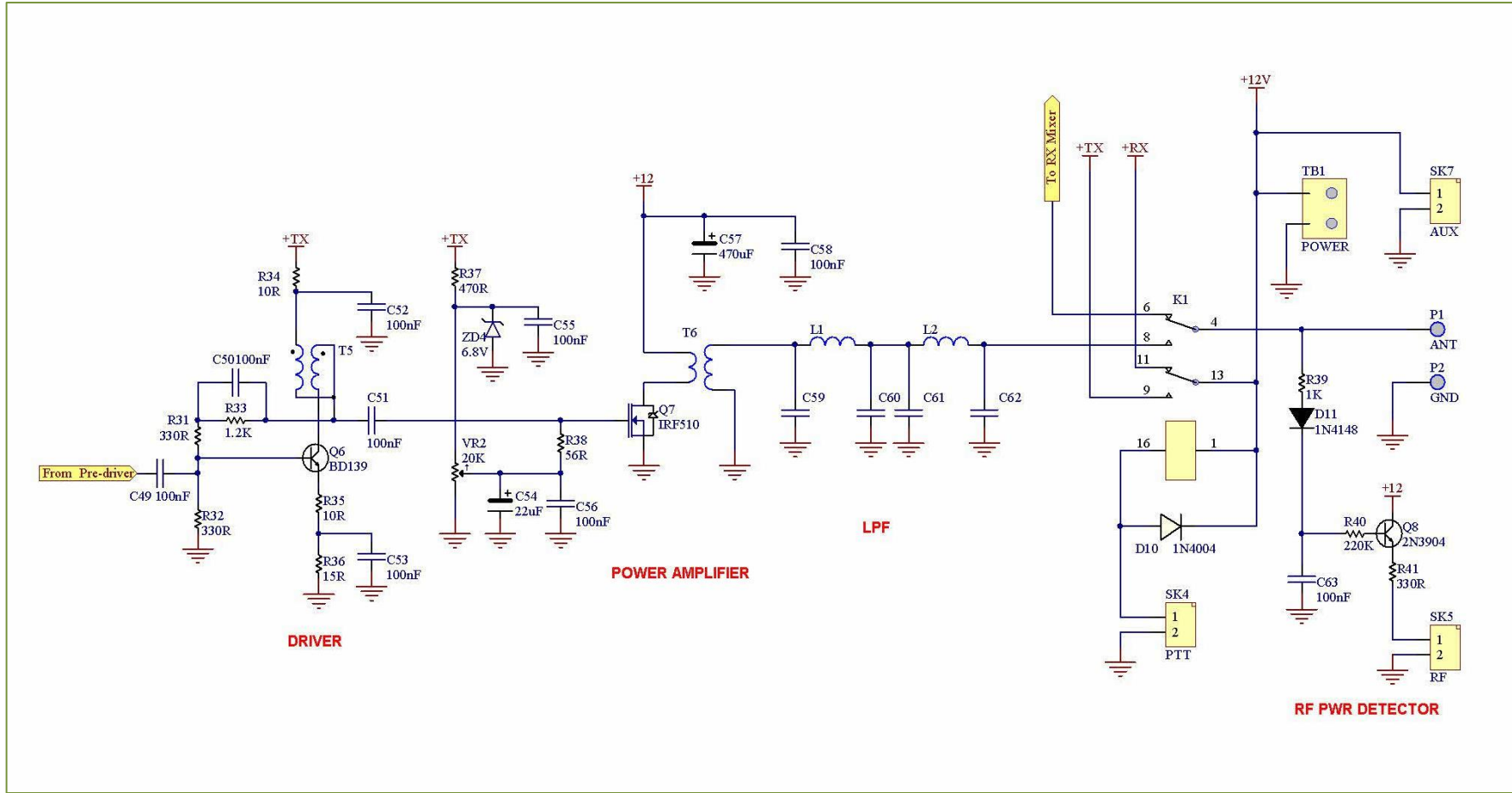


Figure 4 Power amplifier

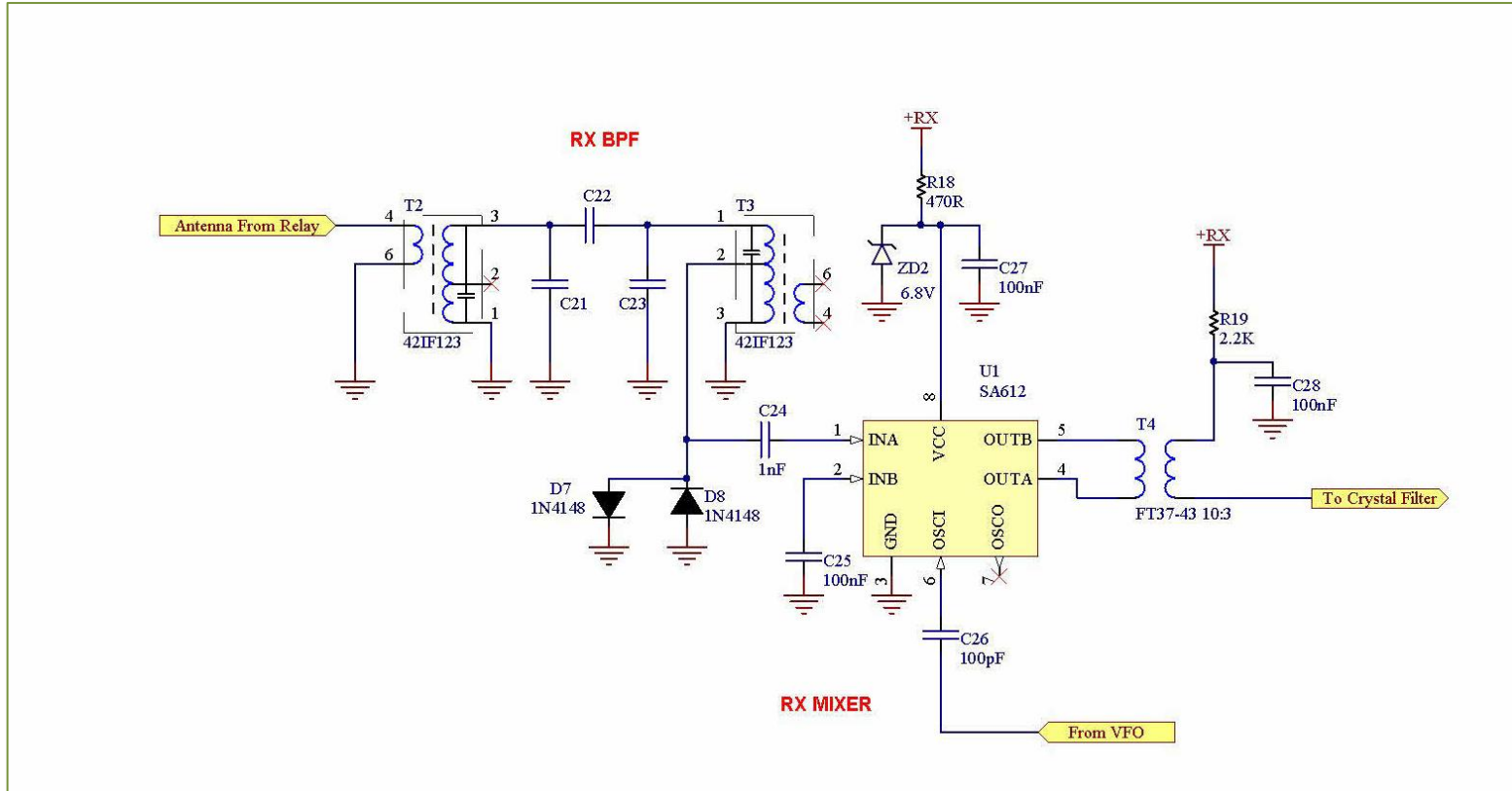


Figure 5 Receive mixer

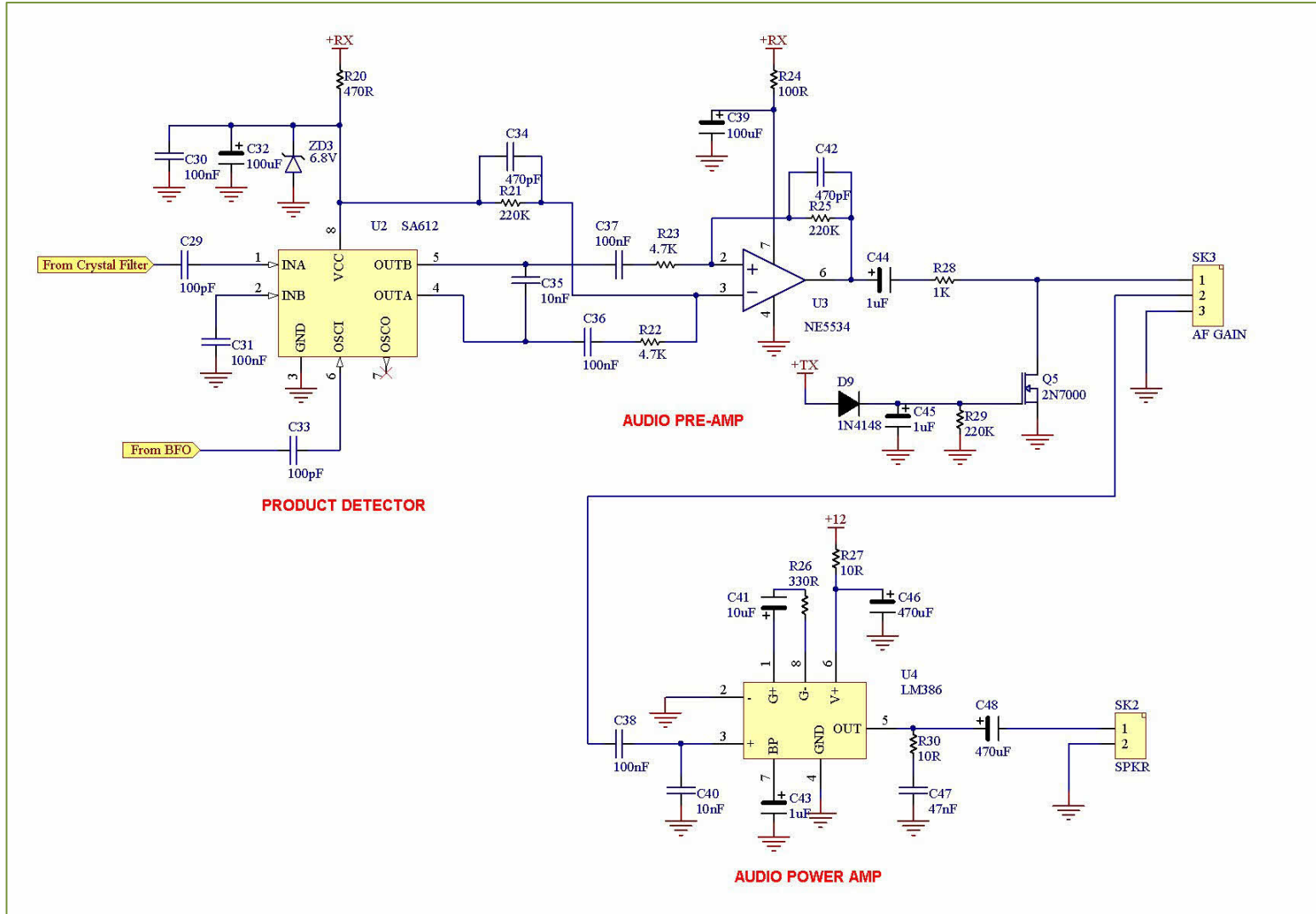


Figure 6 Receive audio

## 4 PARTS LIST

### 4.1 MAIN BOARD PARTS

Quantity	Comment	Designator
2	22pF 50V disc ceramic NPO	C6, C9
2	68pF 50V disc ceramic NPO	C12, C14
3	82pF 50V disc ceramic NPO	C11, C13, C15
7	100pF 50V disc ceramic NPO	C4, C5, C26, C29, C33, C64, C66
1	220pF 50V disc ceramic	C19
2	470pF 50V disc ceramic	C34, C42
2	1nF 50V disc ceramic	C16, C24
1	10nF 50V disc ceramic	C8
2	10nF 63V polyester MKT	C35, C40
1	47nF 63V polyester MKT	C47
3	100nF 63V polyester MKT	C36, C37, C38
22	100nF 50V monolithic ceramic	C3, C7, C10, C25, C27, C28, C30, C31, C49, C50, C51, C52, C53, C55, C56, C58, C63, C65, C67, C69, C70, C72
1	220nF 63V polyester MKT	C17
4	1uF 16V RB electrolytic	C20, C43, C44, C45
2	10uF 16V RB electrolytic	C18, C41
1	22uF 16V RB electrolytic	C54
3	100uF 25V RB electrolytic	C2, C32, C39
3	470uF 25V RB electrolytic	C46, C48, C57
2	50pF ceramic trimcap 0.2" pitch	TC1, TC2
-	See Text	C1
4	10R 1/4W 1% resistor	R27, R30, R34, R35
1	15R 1/4W 1% resistor	R36
1	56R 1/4W 1% resistor	R38
6	100R 1/4W 1% resistor	R4, R6, R15, R17, R24, R43
1	220R 1/4W 1% resistor	R2
6	330R 1/4W 1% resistor	R9, R10, R26, R31, R32, R41
7	470R 1/4W 1% resistor	R7, R16, R18, R20, R37, R42, R46
3	1K 1/4W 1% resistor	R3, R28, R39
1	1.2K 1/4W 1% resistor	R33
3	2.2K 1/4W 1% resistor	R8, R11, R19
4	4.7K 1/4W 1% resistor	R13, R22, R23, R45
1	22k 1/4W 1% resistor	R44
1	47K 1/4W 1% resistor	R12
1	56K 1/4W 1% resistor	R5
1	100K 1/4W 1% resistor	R1
4	220K 1/4W 1% resistor	R21, R25, R29, R40
1	330K 1/4W 1% resistor	R14

Quantity	Comment	Designator
1	500R vertical multi turn trimpot	VR1
1	20K horizontal trimpot	VR2
10	1N4148 signal diode	D1, D2, D3, D4, D5, D6, D7, D8, D9, D11
1	1N4004 1A power diode	D10
1	9.1V 0.5W Zener diode	ZD1
4	6.8V 0.5W Zener diode	ZD2, ZD3, ZD4, ZD5
6	2N3904 NPN transistor	Q1, Q2, Q3, Q4, Q8, Q9
1	2N7000 MOSFET	Q5
1	BD139 NPN transistor	Q6
1	IRF510 Power MOSFET	Q7
3	SA612 RF mixer amp	U1, U2, U5
1	NE5534 low noise op-amp	U3
1	LM386N-4 audio power amp	U4
5	10MHz HC49 crystal	X1, X2, X3, X4, X5
1	DPDT DIP relay 12V coil	K1
1	FT37-43 4 turns: 3 turns 0.5mm	T1
4	42IF123 10.7MHz IF Transformer	T2, T3, T7, T8
1	FT37-43 10 turns: 3 turns 0.5mm	T4
1	FT37-43 8 turns bifilar 0.25mm	T5
1	BN-43-202 2 turns: 5 turns 0.5mm	T6
2	PCB pin 1mm	P1 (ANT), P2 (GND)
6	2 pin 2.54mm pitch header	SK1 (MIC), SK2 (SPKR), SK4 (PTT), SK5 (RF), SK6 (VFO), SK7 (AUX)
1	3 pin 2.54mm pitch header	SK3 (AF GAIN)
1	2 way term block 5.08mm pitch	TB1 (POWER)
2	15mm long 3mm dia heat shrink	
	0.5mm enamelled copper wire	
	0.25mm enamelled copper wire	

## 4.2 BAND SPECIFIC PARTS

Quantity	40M	80M	Designator
1	6.8pF 50V disc ceramic NPO	33pF 50V disc ceramic NPO	C22
4	56pF 50V disc ceramic NPO	390pF 50V disc ceramic	C21, C23, C68, C71
4	470pF 100V monolithic ceramic COG	820pF 100V monolithic ceramic COG	C59, C60, C61, C62
2	1.1uH T50-2 15 turns 0.5mm	2.2uH T50-2 21 turns 0.5mm	L1, L2

## 4.3 OFF BOARD PARTS

Quantity	Comment
1	Plastic instrument case. 200mm x 155mm x 65mm with aluminium panels. <a href="http://www.altronics.com.au">www.altronics.com.au</a> H0480F or equivalent.
1	red binding post
1	black binding post
1	SO239 panel mount socket
3	knobs
2	10K log pot 16mm
1	5mm amber LED and bezel
1	microphone socket
1	front panel label
1	loudspeaker 8 ohm 67mm square or equivalent
4	4g x 6mm self tapping screws
8	3mm x 6mm screws
4	3mm x 10mm screws
4	3mm nuts
1	solder lug
4	12mm long 3mm threaded nylon spacer
4	10mm long 3mm threaded nylon spacer
4	3mm x 16mm countersink screws
8	2 pin 2.54mm pitch header plugs
1	3 pin 2.54mm pitch header plug
	hookup wire



## 5 CONSTRUCTION

### 5.1 GENERAL

The MST is built on a high quality fiberglass PCB. The PCB is double sided with tracks on the bottom side with the top side being a continuous ground plane. The holes are plated through and so it is not necessary to solder both sides to make connections. To assist construction the component overlay is screen printed on the top side and a solder mask is included to guard against solder bridges.

The ground plane is substantial and can sink quite a bit of heat from low wattage soldering irons so ensure you use a good quality iron that can sustain the power required. You may find that sometimes solder doesn't appear to flow through to the top side. This is not necessarily a problem because the plated through holes make a connection to the top side automatically.

Another point to consider is that plated through holes consume more solder than non-plated holes and makes it more difficult to remove components.

**The lesson is to double check the values and orientation of components before installation.**

There isn't a 'best' scheme for loading the components. If desired you can build sections at a time and test them out, but it is not necessary and in any case some sections rely on others before they will operate. The suggested procedure is to load the smaller components first and then work upwards.

### 5.2 CONSTRUCTION STEPS

**Refer to the parts list and Figure 7 when installing the components.**

#### **Step 1: Resistors**

Install and solder a few at a time. It is easier and less confusing to install a group with the same value rather than to cover a section of the PCB with mixed values. If in any doubt about reading resistor values measure them with a multimeter before soldering.

Pass the pigtailed through from the top and bend out slightly underneath to hold them in place. Turn the PCB over and press down slightly to make them rest against the surface and solder. Cut off the excess pigtail with side cutters.

#### **Step 2: Diodes**

Note the positive or cathode end before installation. The small Zener diodes look like signal diodes so make sure you don't get them mixed up.

#### **Step 3: Non-polarized capacitors**

Note the various types used. Ceramic disc, ceramic monolithic and polyester MKT. These are non-polarized and can go in either way.

#### Step 4: Trimmer capacitors and trimpots

The type specified for the trimmer capacitor is quite small and has one lead electrically connected to the screwdriver adjustment slot. Use a multimeter to determine this pin and solder to the hole in the PCB connected to the ground plane.

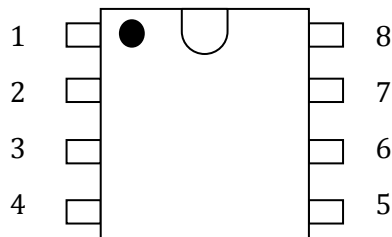
Note that the carrier balance trimpot is a multi-turn vertical mount while the bias trimpot is a horizontal mount type.

#### Step 5: Transistors

The 2N3904 transistors and 2N7000 MOSFET are orientated to match the screen silk component overlay. The BD139 is installed so that the metal side of the body faces inwards towards the centre of the PCB. Leave the IRF510 power MOSFET installation till later.

#### Step 6: Integrated Circuits

All ICs are 8 pin DIP types and have either a dot above pin 1 or a notch at the top between pin 1 and pin 8. This is shown in the top view diagram below. Double check they are installed in the correct orientation.



#### Step 7: Electrolytic capacitors

These are polarized and must go in the correct way. The component overlay has a '+' mark to indicate the positive lead.

#### Step 8: Connectors

There are a number of connectors fitted to the PCB to provide a neat finished product. They also allow easy removal of the PCB if required. The 2.54mm pitch connectors have a vertical polarizing piece and the connectors are installed with this piece towards the centre of the PCB.

The power connector is a 2 way terminal block and the terminal openings face towards the edge of the PCB.


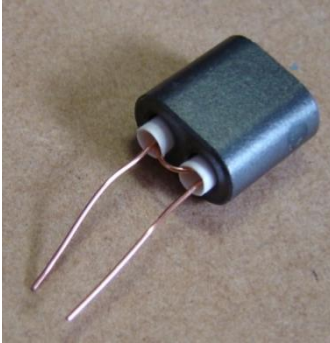
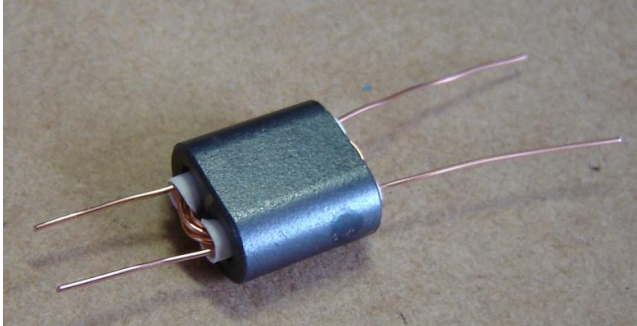
The antenna and its ground connection are soldered to a pair of 1mm PCB pins. This is done to ensure a positive low resistance connection.

#### Step 9: IF transformers and relay


The four IF transformers can only go in one way and so are simply fitted into the holes in the PCB and soldered. The relay is inserted into the board and soldered.

## Step 10: Coils

### Output transformer T6

<p>T6 is wound on a 13mm two hole ferrite balun former type BN-43-202. The edges of the holes are quite sharp and can scrape the enamel off the wires. To reduce scraping and the risk of shorts, cut two lengths of 3mm heat shrink and feed into the balun former holes.</p>	
<p>Wind the 2 turn primary with 0.5mm enamelled copper wire. The primary winding connects to Q7 drain. A turn is where the wire passes up through one hole and down the other.</p>	
<p>Starting at the opposite end wind the 5 turn secondary in the same manner. Use 0.5mm enamelled copper wire. The secondary is the output and connects to the LPF. Scrape the enamel off the ends of the wires and tin with solder. Check with a multimeter that there are no shorts between the windings before installing in the PCB.</p>	

### Low Pass Filter Coils L1 and L2

<p>Wind the turns shown below for the required band using 0.5mm enamelled copper wire on a T50-2 toroid.</p> <p>40M: 15 turns. 80M: 21 turns</p> <p>Note the direction of winding as this makes for a neater alignment of the toroid in the PCB.</p> <p>Scrape the enamel off the ends of the wires and tin with solder before installing in the PCB.</p>	
---	--

## Transformers T1 and T4

Use a FT37-43 ferrite toroid and 0.5mm enamelled copper wire.

T1: 4 turn primary (balanced modulator), 3 turn secondary (output)

T4: 10 turn primary (U1), 3 turn secondary (output)

Scrape the enamel off the ends of the wires and tin with solder before installing in the PCB.



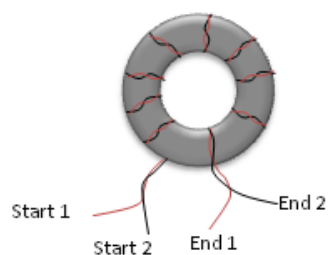
## Bifilar transformer T5

Use a FT37-43 ferrite toroid and 0.25mm enamelled copper wire.



Take two 300mm lengths lay them parallel and then twist together until there are about 3 twists per centimetre. A vice and battery drill with a hook shaped bit make this job easy.

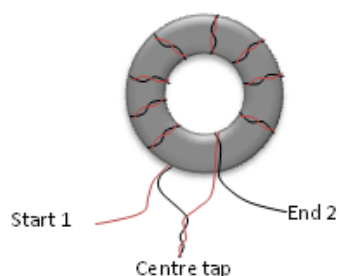
Wind on 8 turns. Scrape some enamel off the ends and use a multimeter to find the start and end of each winding.



Take the start of one winding and the end of the other winding and twist together to form the centre tap.

Trim the leads with sidecutters and tin with solder before installing. Push the wires through the holes in the PCB and sit the toroid so that it rests against the surface of the board.

Ensure the two-wire centre tap goes to the middle hole in the PCB overlay for T5.



### **Step 11: Crystals**

The crystals are mounted close to the PCB but not touching it. This is to avoid the metal case of the crystal shorting pads underneath. A space of about 1mm is all that's needed. Once the crystals for the filter are installed solder a wire across the top of the metal cases and down to a spare pad in the PCB groundplane set aside for this purpose. This will help keep out noise and resist widening of the filter skirts.

### **Step 12: Antenna wires**

The antenna wires are hard to solder to the PCB pins once the PCB is installed in the case so solder short lengths to the ANT and GND pins now.

### **Step 13: Power MOSFET**

The reason this is left to last is simply because it is near the edge and may get bent and possibly damaged as the PCB is turned over and back as other components are being installed.

Mount the device with about 5mm of lead length between the PCB and body.

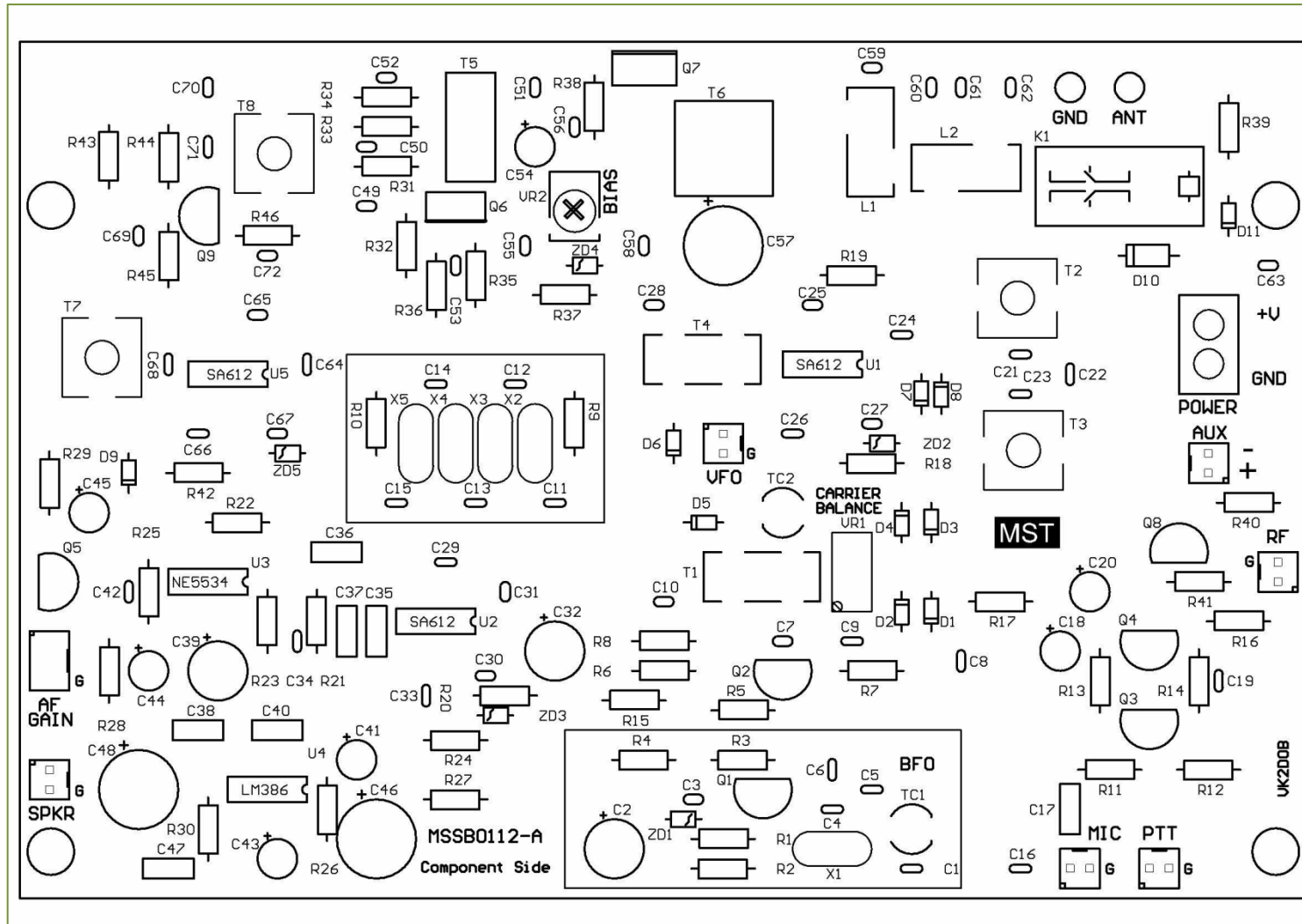


Figure 7 Component overlay



## 6 ENCLOSURE

Now that your MST board is completed you'll want to put it in an enclosure and try it out. The PCB was designed to fit inside a plastic instrument enclosure 200mm wide by 155mm deep by 65mm high. A suitable type is catalogue number H0480F available from [www.altronics.com.au](http://www.altronics.com.au). It can be purchased with aluminium panels or you can easily make them yourself from 1.5 or 2mm thick aluminium sheet. The PCB mounting holes match plastic pillars on the bottom of the enclosure and it will be necessary to remove unwanted pillars using either a large pair of sidecutters or carefully with a large drill. Of course any other enclosure that accommodates the PCB will be suitable, although if using the companion DDS VFO it will need to have similar minimum dimensions.

Before installing the board in the enclosure, carefully look for errors, such as components in the wrong way and solder bridges between tracks. The risk of solder bridges is greatly reduced due to the solder mask, but check anyway. A few moments spent here is cheap insurance against big problems later on.

One of the more common problems is poor solder joints with enamel covered wire. Some types when soldered will melt the enamel but most will not, so it is important to remove the enamel off the ends of the wires with sandpaper or a sharp knife before soldering. Double check connections with a multimeter.

### 6.1 REAR PANEL

The rear panel only contains the SO239 antenna socket and two binding post to connect the power supply. As the rear panel layout is straightforward no layout drawing is supplied, however remember to locate the antenna socket near the PCB pins so the wires are kept short.

### 6.2 FRONT PANEL

The front panel is designed to accommodate the DDS VFO. It requires a number of holes and cut-outs to be made as shown in Figure 8. The microphone hole is dimensioned to fit a standard 4 pin connector however this may need changing to match your own.

There are two options for the VFO mounting holes. Make them countersink to have a flush surface and no holes in the label, or if you are happy to have screw heads showing on the front leave them straight drilled.

### 6.3 MOUNTING THE PCB

Temporarily put the rear panel in place and sit the PCB on the mounting pillars. If needed bend the leads of Q7 slightly so that the metal tab is resting parallel against the rear panel. Use a sharp pencil or scribe and run around the inside of the hole in the tab to mark the position for the mounting hole on the rear panel. Remove the rear panel and drill a 3mm hole ensuring that you remove all burrs.

Screw the PCB to the mounting pillars using small self tapping screws. Slide an insulating washer between the rear panel and Q7 tab. Insert a plastic bush into Q7 tab then pass a 3mm screw from the rear through the bush and screw on a nut and tighten. Check with a multimeter that there is no electrical connection between the rear panel and Q7 tab.

## 6.4 FRONT PANEL LABEL

A front panel label will add a professional look to the completed transceiver. You can design one yourself using a graphic design software package or you can simply download a completed sample from [www.ozQRP.com](http://www.ozQRP.com). If you plan to design your own then use the dimensions shown in Figure 8 as a guide. Once you have either your own file or the downloaded file follow the procedure below to produce the label:

1. Print the file using a colour inkjet printer onto an A4 sheet of high quality photo paper.
2. Using a sharp hobby knife and steel rule as a guide cut out the inside area of the LCD window. Touch up any bare paper edges with a black permanent marker.
3. Laminate the whole sheet using an A4 laminator. If you don't have your own then this can be done quite cheaply at large stationary stores. The exciting thing about this is that the LCD cutout becomes a transparent window.
4. Using the hobby knife cutout the other holes in the label.
5. Cut around the border of the label so that it is the same size as the front panel.
6. Place the label on the front panel and install the pots, LED bezel and microphone socket. You can use some spray adhesive to secure the label to the front panel if it tends to lift off, although if using the recommended case it will stay in place once the enclosure is assembled.



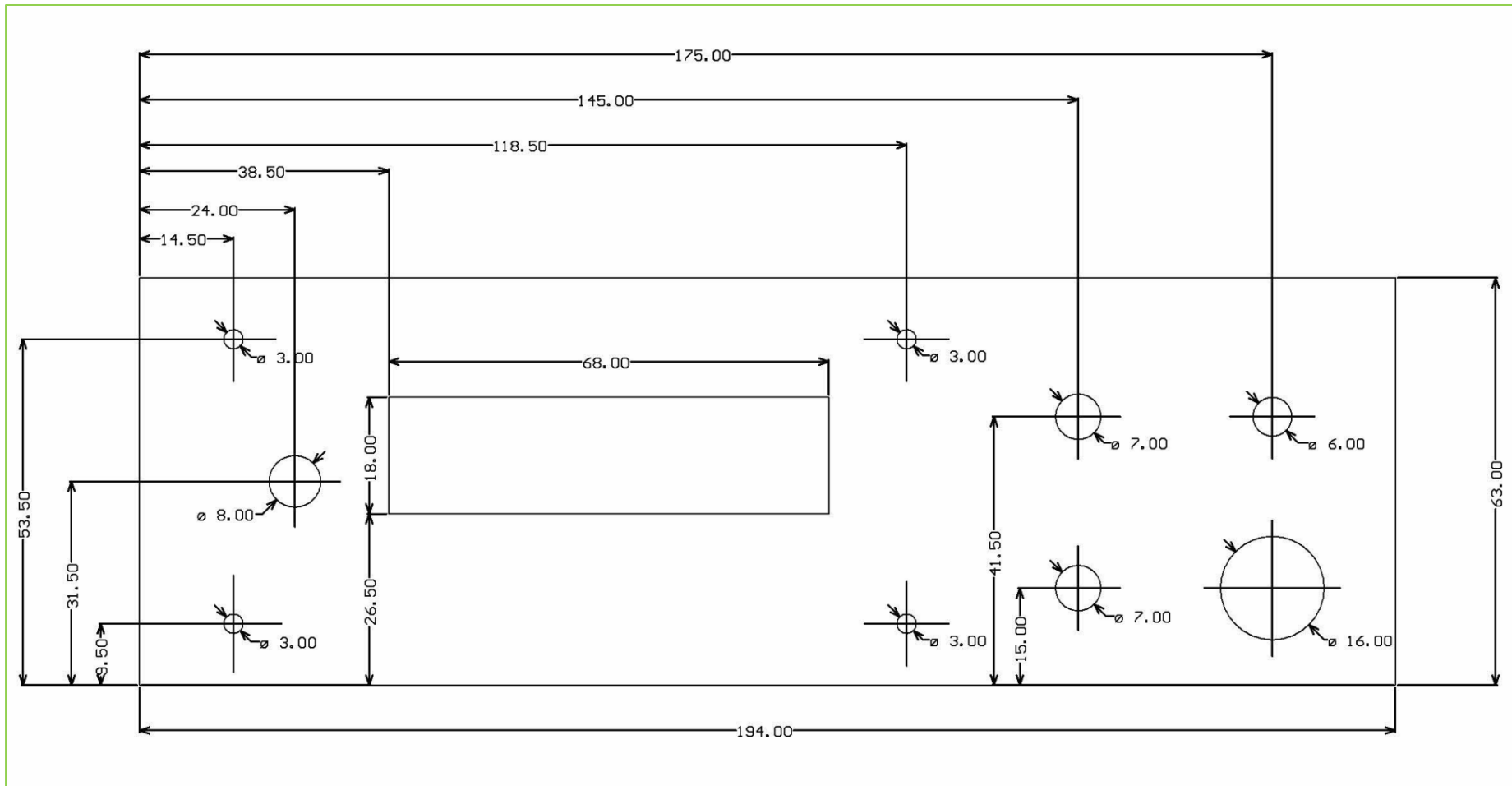


Figure 8 Front panel drilling guide

## 6.5 FITTING THE DDS VFO

The DDS VFO is simply screwed to the inside of the front panel. The DDS VFO PCB is set back from the panel using threaded spacers to allow room for the LCD. If you use the specified LCD the distance will be 22mm and is made up of a 10mm spacer and 12mm spacer.

A countersink screw is used on the front panel to provide a flush surface for the front panel label to sit. If preferred round head screws can be used which means that holes need to be made in the label and also the heads will be visible from the front.

If a LCD is used that does not incorporate a backlight it will not be as high and so the length of the screws and spacers need to be reduced accordingly.

The general mounting arrangement is shown in Figure 9.

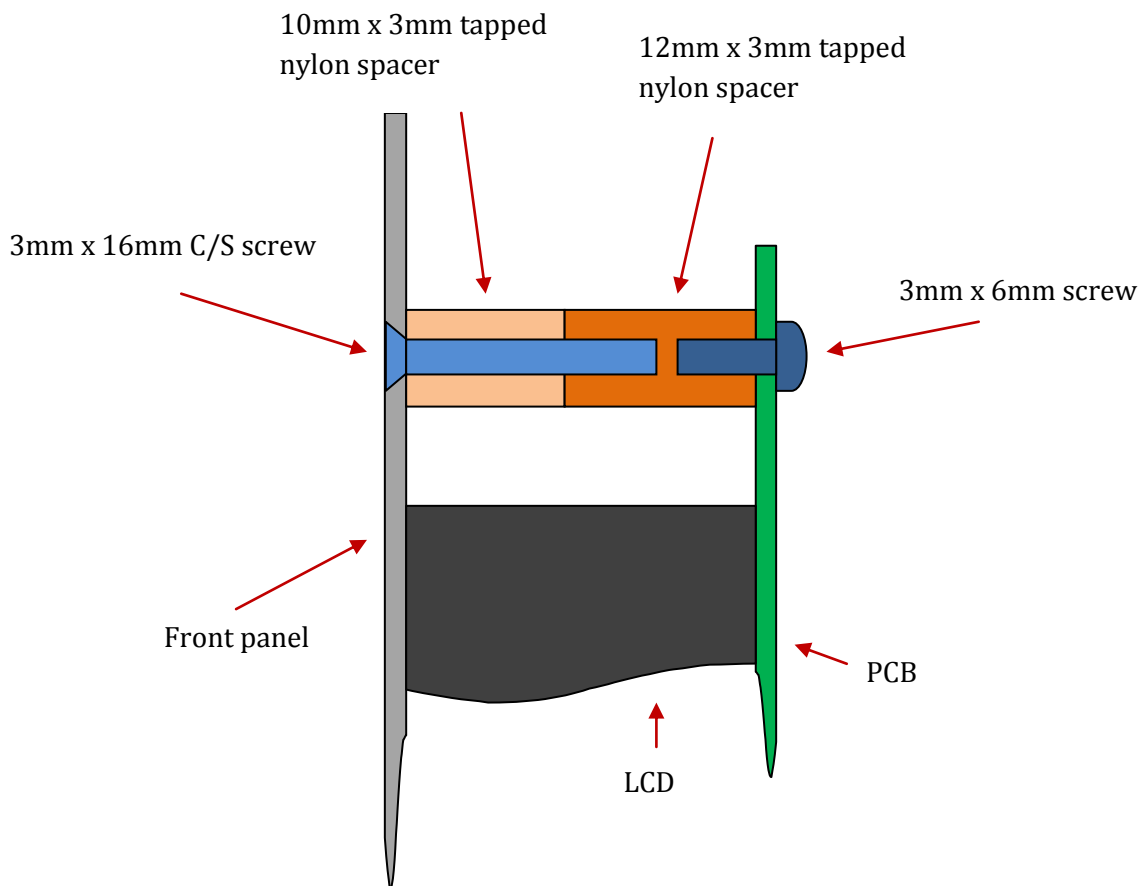


Figure 9 DDS VFO front panel mounting

## 7 WIRING UP

The wiring diagram is shown in Figure 10. Use light duty hookup wire of different colours for each connection and twist the wires together. It was found that shielded cables were not required.

Most PCB connections utilize 2.54mm pitch polarized headers and mating plugs. These are inexpensive, give a professional looking appearance and allow easy disconnection and removal of the PCB if required. The plugs are comprised of a plastic housing and crimp contacts which slide and click in place. If you don't have a suitable crimper it is best to squeeze the contact wings around the wire with long nose pliers to hold in place and then solder. Be careful not to use too much heat and solder as it will make it difficult to insert the contact into the housing.

Not that the microphone socket PTT wires go directly to the PCB but the audio wires connect to the mic gain pot before going to the PCB.

The power supply connection uses a screw terminal block because of the higher currents employed.

The antenna connection uses short wires soldered between the antenna socket and PCB pins to give a low resistance connection. You will also need to install a solder tag under one of the screws holding the antenna socket for the earth wire.

The loudspeaker is mounted on the underside of the enclosure lid and held in place with 3mm countersunk screws and nuts. A series of holes needs to be drilled in the lid to act as a speaker grill.

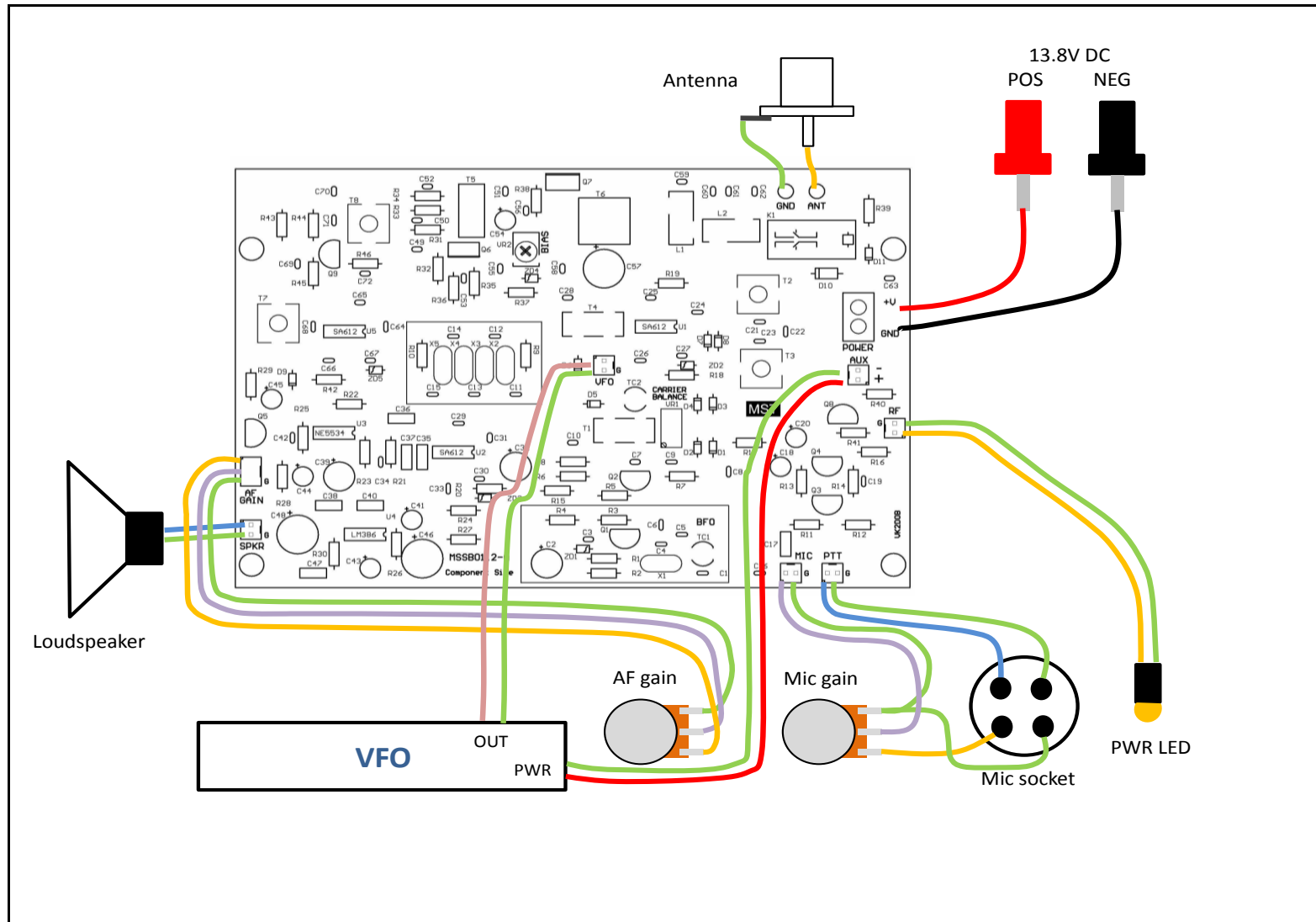


Figure 10 Wiring diagram

## 8 TESTING AND ALIGNMENT

### 8.1 GENERAL

A suggested list of basic tools and accessories to undertake testing is shown below.

- Digital multimeter.
- 50 ohm dummy load capable of dissipating at least 5 Watts.
- QRP wattmeter or oscilloscope.
- Power supply capable of 13.8V DC at 2 Amps.
- Small adjustment screwdriver.
- An audio signal generator.
- An RF signal generator.
- A frequency counter.

**Obviously an external VFO is required for testing and operation and it is assumed during the rest of this section that the MST board is being used with the DDS VFO which is already configured and fully operational.**

**Figure 11 lists typical voltages at various points around the board and can be used to verify operation.**

### 8.2 POWER ON

- Temporarily remove the power connector for the VFO so that the MST board current can be measured alone.
- Using a small screwdriver:
  - Rotate the bias trimpot VR2 fully ant-clockwise so that the gate bias voltage is zero volts.
  - Set the trimmer capacitors to halfway.
  - Set carrier balance trimpot VR1 to halfway by counting turns. You should hear a slight click as you reach the end.
- Connect the power supply to the board terminals. **Double check the polarity is correct before applying power.**
- If your power supply does not include a current meter, connect a multimeter in series with the power supply positive lead and set to measure current.
- Connect a 50 Ohm dummy load to the antenna socket.
- Turn on the power supply and note the supply current. It should be around 70mA. If it's far from this turn off immediately and look for problems.
- The relay should be de-energized and the board in receive mode. To verify that there are no obvious problems do a quick probe around the board with a multimeter and check the DC receive voltages as shown in Figure 11.
- Plug in the VFO power connector. Check the power supply current. If using the DDS VFO it should rise to around 150mA but this is highly dependent on the type of LCD used and the backlight current.

### 8.3 RECEIVE

- Set the VFO to a frequency in the middle of the band you will be using.
- Turn the AF gain control to halfway. You should hear some low level hiss come from the speaker indicating that the audio stages are working.
- Connect a frequency counter at the emitter of transistor Q2. Adjust trimmer capacitor TC1 until the frequency is 10.000MHz. This is the carrier (BFO) frequency and is placed on the upper skirt of the crystal filter to produce a Lower Sideband signal.
- Remove the dummy load and apply a moderate RF signal to the antenna connector and tune the VFO until a clear tone is heard in the speaker. Carefully adjust the slugs in coils T2 and T3 for maximum volume. As you approach maximum you will need to turn the AF gain down and probably the RF signal level as well to avoid overload. You can experiment with the position of the two slugs to obtain a specific bandwidth, but in general, peaking at the middle of the band will be satisfactory.

### 8.4 TRANSMIT

- Remove the RF signal source and reconnect the dummy load. It will be an advantage to have either a QRP power meter or oscilloscope to obtain an indication of power output.
- Turn the mic gain control fully anticlockwise.
- Briefly switch to transmit mode by shorting the PTT contacts. Check the power supply current. This is the quiescent transmit current and should be about 100mA higher than the receive current. Anything far from this indicates a problem and should be investigated.
- Operate the PTT and using a screwdriver slowly rotate the bias trimpot clockwise. The power supply current should rise gradually and smoothly. Keep increasing until the power supply current is 150mA higher than quiescent transmit current. This increase is a result of the current being drawn by the output MOSFET as it starts to conduct and move into linear operation.
- Adjust carrier balance controls TC2 and VR1 for minimum power output. There is some interaction so you will need to go back and forwards to obtain maximum balance.
- Apply an audio signal generator to the mic socket set to 1KHz at around 100mV. Operate the PTT and slowly increase the mic gain while monitoring the power output. When around 1 Watt is indicated carefully adjust the slugs in coils T7 and T8 for maximum output. You may need to reduce the mic gain to find the peak. Once the coils are peaked operate the mic gain up and down while monitoring the power output. The power output should change smoothly and you should be able to easily achieve 5 Watts output.
- Take note of the RF power LED and verify that it illuminates and changes intensity with varying power output.
- Plug in a microphone and verify operation. You will now be able to monitor yourself with a receiver placed nearby using a small length of wire as an antenna.

Location	V DC Receive	V DC Transmit
U1 pin 8	+6.8V	0V
U1 pin 5	+5.6V	0V
U2 pin 8	+6.8V	0V
U2 pin 5	+5.6V	0V
U3 pin 7	+13.3V	0V
U3 pin 6	+6.7V	0V
U4 pin 6	+13.7V	0V
U4 pin 5	+6.7V	0V
Q1 collector	9V	9V
Q1 emitter	5.6V	5.6V
Q2 emitter	0V	6V
Q4 emitter	0V	7.2V
Q3 collector	0V	8V
U5 pin 8	0V	+6.8V
U5 pin 5	0V	+5.6V
Q9 emitter	0V	1.5V
Q6 emitter	0V	1.5V
Q7 gate	0V	3.9V

**Note readings taken with a power supply voltage of 13.8V DC.**

**Figure 11 Typical circuit voltages.**

## 8.5 CARRIER FREQUENCY ADJUSTMENT

The suggested carrier frequency is 10.000MHz which places it just above the passband and on the upper skirt of the crystal filter frequency response. The frequency response shown in Figure 12 was obtained from a completed prototype MST board. Note the passband is approximately 2.5KHz.

Due to tolerances in components the response of the crystal filter in your board may be shifted slightly to that shown here. This can result in one of the following.

1. If the response is shifted higher in frequency to the right, the carrier frequency is within the pass-band and the upper sideband is not suppressed sufficiently.
2. If the response is shifted lower in frequency to the left, the carrier frequency is too far away from the pass-band and results in 'thin' sounding voice due to the lack of low frequencies.

If you feel this is the case then the carrier frequency may need adjusting. There are a number of ways to go about this but the easiest is to listen to a good strength station of known quality while gently adjusting trimmer capacitor TC1. Adjust until the received audio sounds natural and the balance of high and low audio frequencies is correct.

Alternatively monitor yourself on a nearby receiver while speaking and adjust the carrier frequency until you sound normal. You can also switch between sidebands on the receiver to hear the reduction in the upper sideband. Note that you will need to adjust the VFO frequency as you make the adjustment as the received signal will be moved off frequency by an amount equal to the change in the carrier frequency.

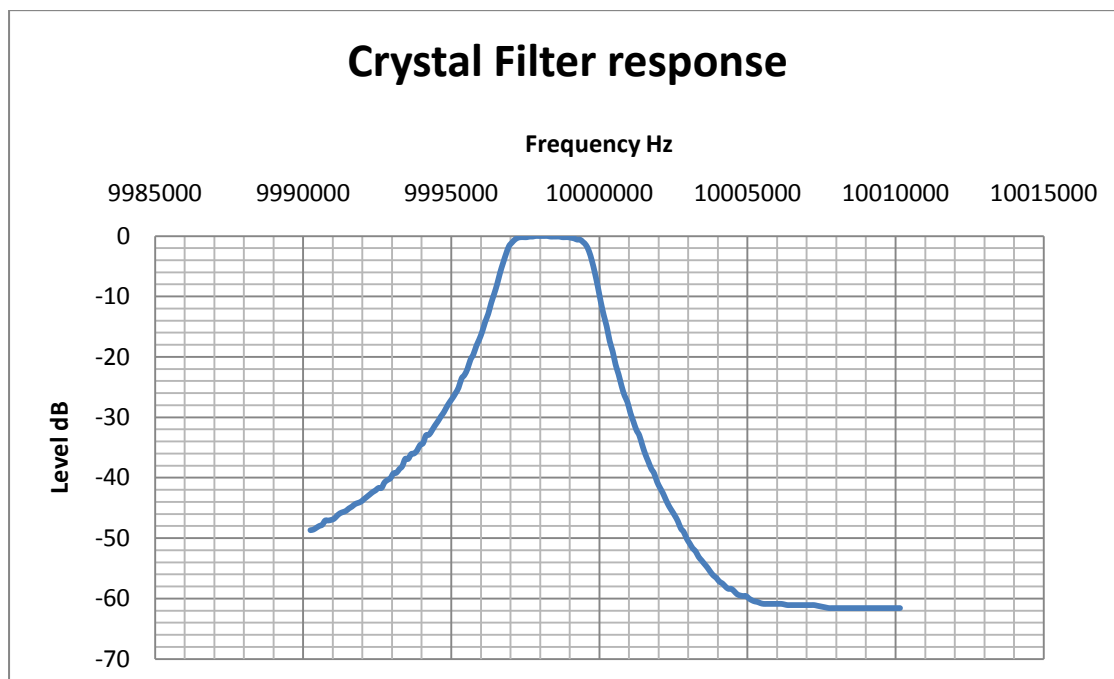


Figure 12 Crystal filter response

If you move the carrier frequency off 10.000MHz and are using the companion DDS VFO then you will need to measure and re-program the IF frequency to ensure the display reads accurately. See the DDS VFO construction manual for details.



## 9 OPERATION

Operating the MST is very easy as there are no complicated controls and software menus to navigate. It's simply a matter of connecting a power supply, microphone and antenna and talking. Operation of the DDS VFO is covered in the DDS VFO construction manual.

In receive mode apart from the VFO there is only the AF gain control to adjust. The MST does not incorporate automatic gain control (AGC) and so you will need to adjust the AF gain control to even out signals. In practice though you'll find that you won't be continually twiddling the knob and it will only be when stations are either very weak or very strong that you will find it necessary to make changes.

The MST as described does not include a headphone socket. If you add one yourself make sure that you include a series limiting resistor to reduce the output of the LM386 otherwise you will find audio peaks quite uncomfortable.

To transmit press the PTT button on your microphone and talk. The front panel RF LED indicator will illuminate on voice peaks. It can also be used as a rough guide to power output as the current through the LED, and therefore its brightness, is dependent on the peak RF output. If the LED is illuminated at a constant level when talking, it indicates that you are driving the transmitter too hard and causing clipping of the RF signal.

**Overdriving will create distortion and excessive harmonic generation and must be avoided.**

To check your signal either connect the MST to a dummy load and monitor yourself with headphones on a nearby receiver, or have a friend that lives close by listen to your signal. The idea is to increase the mic gain progressively while sweeping across your transmission looking for distortion and unwanted spurious byproducts across the band. Set the mic gain control just below the point where these are noticeable.

Congratulations your new SSB QRP transceiver is ready to put on the air.

Have fun!

# DDS VFO



## CONSTRUCTION MANUAL

 [www.ozQRP.com](http://www.ozQRP.com)

# **Important**

## **Please read before starting assembly**

### **STATIC PRECAUTION**

**The DDS VFO kit contains the following components which can be damaged by static discharge:**

- **The DDS chip which is pre-soldered to the PCB and shipped in an anti-static bag.**
- **The microcontroller which is pre-programmed and shipped in an anti-static bag.**

**Do not remove the devices from their protective anti-static bags until you have taken precautions against static discharge.**

- **If possible use an anti-static wrist strap and conductive mat. These can be purchased readily from electronic retailers.**
- **If these are not available then at least ensure you have discharged yourself by touching an earthed metal surface.**
- **Try not to directly touch the pins of the devices.**
- **Ensure your soldering iron has an earthed tip.**

# Contents

1	Introduction.....	4
2	Circuit Description .....	5
3	Parts List.....	7
4	Construction.....	8
4.1	General.....	8
4.2	Construction Steps.....	8
5	Testing.....	11
6	Configuration.....	11
6.1	Understanding options .....	11
6.2	Procedure .....	12
7	Operation.....	13
8	Options .....	14

# List of Figures

Figure 1	Circuit diagram.....	6
Figure 2	Component overlay .....	10
Figure 3	Typical LCD operating display .....	13

# 1 INTRODUCTION

The DDS VFO is designed as a companion for the MST transceiver board although it could be used with most other Superhet or Direct Conversion radios. It is simple to use having only one control yet offers many advanced features. A short form kit is available containing a PCB with the DDS chip already loaded as well as the hard to get parts. You only need to add a few inexpensive components and an hour or so of time and your new DDS VFO is ready to go.

DDS VFO Features:

1. Stable frequency – no more drift!
2. Simple to operate using a single rotary encoder control with press button switch.
3. Bright 16 character by 2 line LCD display. Can also be used with a non-backlight display to reduce current consumption if needed.
4. Output frequency range of 1MHz to 10MHz.
5. Selectable 1KHz, 100Hz or 10Hz frequency steps.
6. Tuning lock control.
7. Displays power supply voltage.
8. Programmable IF frequency. Ensures the displayed frequency is the same as the transmitted frequency.
9. Programmable IF offsets:
  - Minus – DDS output frequency is the IF frequency minus the displayed frequency.
  - Plus - DDS output frequency is the IF frequency plus the displayed frequency.
  - Rev - DDS output frequency is the displayed frequency minus the IF frequency.
  - None – DDS output frequency is the same as the displayed frequency. Used for frequency generator applications or with Direct Conversion receivers.
10. Programmable option to save the current frequency and step into memory. These are re-loaded at power up.
11. Draws only around 80mA at 13.8V DC (with LCD backlight).
12. Spurious outputs at least -45dB.
13. Typically 300mV pk-pk sine wave output (200 ohms).
14. High quality double sided PCB with ground plane, plated through holes, solder mask and silk screen.
15. Simple and easy to build using through hole components apart from one SMD chip. To assist constructors the PCB comes shipped with the SMD chip already installed.

PCBs and a range of kits containing hard to get parts for the DDS VFO and the MST SSB transceiver are available from [www.ozQRP.com](http://www.ozQRP.com).

## 2 CIRCUIT DESCRIPTION

Integrated circuit U2 is an Analog Devices AD9834 Direct Digital Synthesis (DDS) chip and forms the heart of the DDS VFO. A 50MHz crystal oscillator module supplies the master clock and gives usable output frequencies up to a quarter of that value or 12.5MHz.

The DDS is controlled by a 3 wire serial data bus from microcontroller U1. The microcontroller is an Atmel ATmega8 and runs an internal 8MHz clock which avoids the use and expense of an external crystal.

A rotary encoder produces quadrature signals when rotated and one line is fed into the microcontroller interrupt input and becomes the reference input. Once an interrupt is detected the state of the other input is read and the direction determined. The rotary encoder also incorporates a press button switch and is used for a number of functions as described later. U1 incorporates pull-up resistors for the encoder inputs so they are normally held high and capacitors C7 and C8 filter out switch bounce.

The microcontroller has an internal analog to digital converter (ADC) which uses the 5 volt rail as a reference. The ADC measures the incoming power supply rail via a resistive attenuator formed with R5 and R6. The calculated value is displayed on the LCD during use. Two other inputs, one analog (A) and the second digital (D), have been included in the PCB layout but are not implemented in the current software release. The PCB also includes an ISP connector to allow future software upgrades without having to remove the chip from its socket.

Most of the remaining pins of U1 are dedicated to driving the LCD in 4 bit mode. R9 sets the backlight current and may be left out if a non-backlight LCD is used. VR1 adjusts the LCD contrast and normally only needs to be set once at the testing stage.

The outputs from the DDS chip are current sources and the output is converted to a voltage by the 200 ohm resistors R12 and R13. The full scale value of the current and hence the pk-pk output voltage is set by the 5.6K ohm resistor R11. The output is passed through a simple 10MHz Butterworth low pass filter to attenuate unwanted high frequency components.

The incoming power supply is regulated to 5V by a 7805 1A regulator which mounts flat on the PCB to act as a heatsink.

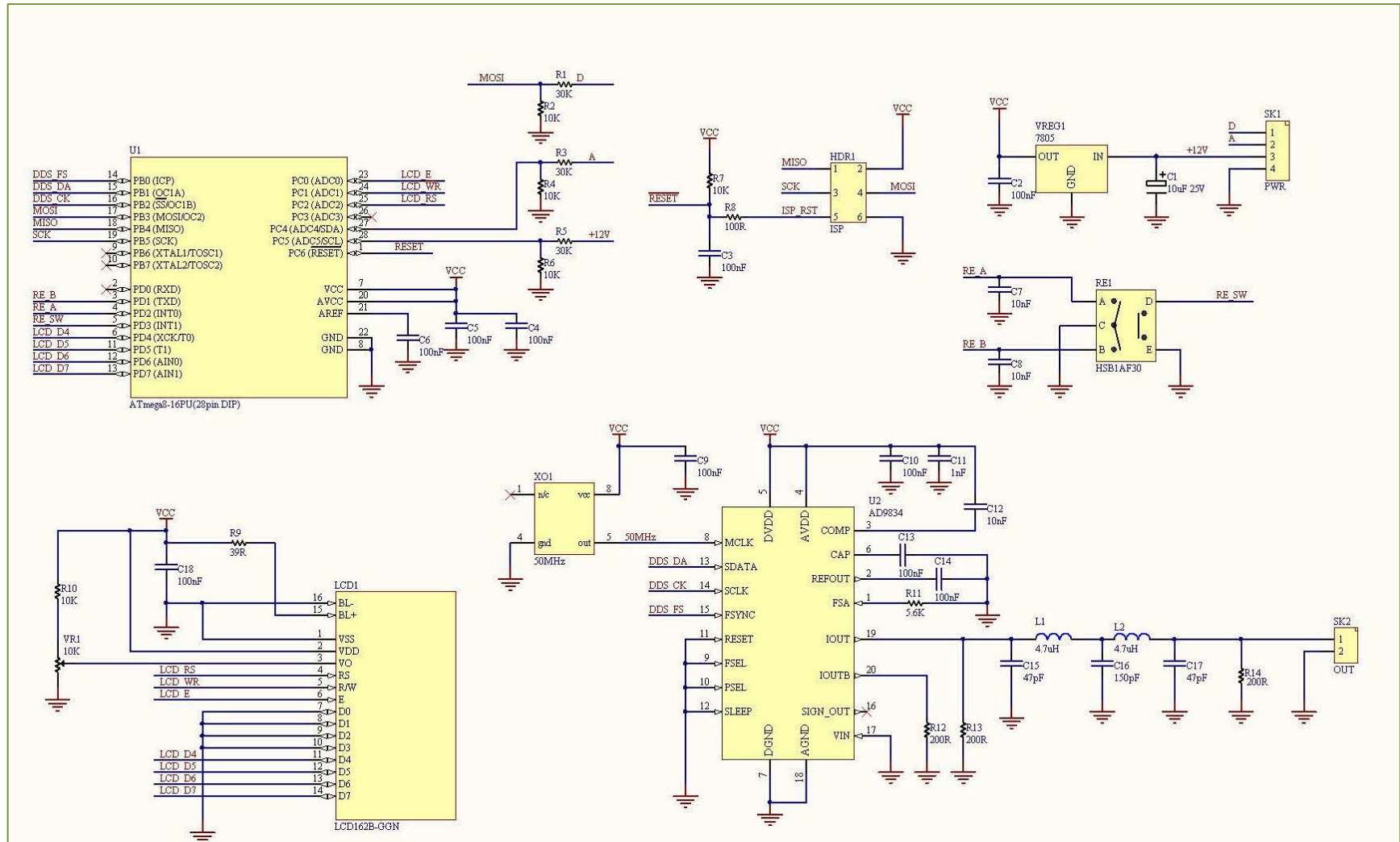


Figure 1 Circuit diagram

### 3 PARTS LIST

Quantity	Comment	Designator
2	47pF 50V disc ceramic	C15, C17
1	150pF 50V disc ceramic	C16
1	1nF 50V disc ceramic	C11
3	10nF 50V monolithic ceramic	C7, C8, C12
10	100nF 50V monolithic ceramic	C2, C3, C4, C5, C6, C9, C10, C13, C14, C18
1	10uF 25V RB electrolytic	C1
1	39R 1/4W 1% resistor	R9
1	100R 1/4W 1% resistor	R8
3	200R 1/4W 1% resistor	R12, R13, R14
1	5.6K 1/4W 1% resistor	R11
5	10K 1/4W 1% resistor	R2, R4, R6, R7, R10
3	30K 1/4W 1% resistor	R1, R3, R5
1	10K horizontal trimpot	VR1
1	ATmega8-16PU 28pin DIP	U1
1	AD9834BRUZ DDS 20 pin TSSOP	U2
1	7805 5V regulator TO220	VREG1
1	50MHz crystal oscillator module - half size	XO1
1	16 x 2 LCD with backlight. 80mm x 36mm. Winstar WH1602B or equivalent.	LCD1
2	4.7uH moulded RF choke	L1, L2
1	3 pin x 2 row 2.54mm pitch header	HDR1 (ISP)
1	11mm Rotary Encoder. BI Technologies EN11-HSB1AF30 or equivalent	RE1
1	4 pin 2.54mm pitch header RA	SK1 (PWR)
1	2 pin 2.54mm pitch header RA	SK2 (OUT)
4	12mm long 3mm nylon spacer	
4	2.5mm x 20mm screws	
4	2.5mm nuts	
1	16 way SIL male pin header	
1	16 way SIL female pin header	



## 4 CONSTRUCTION

### 4.1 GENERAL

The DDS VFO is built on a high quality fiberglass PCB. The PCB is doubled sided with tracks on both sides along with a ground plane. The holes are plated through and so it is not necessary to solder both sides to make connections. To assist construction the component overlay is screen printed on both sides and a solder mask is included to guard against solder bridges.

The ground plane is substantial and can sink quite a bit of heat from low wattage soldering irons so ensure you use a good quality iron that can sustain the power required. You may find that sometimes solder doesn't appear to flow through to the top side. This is not necessarily a problem because the plated through holes make a connection to the top side automatically.

Another point to consider is that plated through holes consume more solder than non-plated holes and makes it more difficult to remove components.

**The lesson is to double check the values and orientation of components before installation.**

There isn't a 'best' scheme for loading the components, however, the suggested procedure is to load the smaller components first and then work upwards.

### 4.2 CONSTRUCTION STEPS

**Refer to the parts list and Figure 2 when installing the components.**

#### **Step 1: Resistors**

**If in doubt check the values with a multimeter before soldering.** Pass the pigtails through from the top and bend out slightly underneath to hold them in place. Turn the PCB over and press down slightly to make them rest against the surface and then solder. Cut off the excess pigtail with side cutters. The trimpot can also be installed at this stage. R14 is not used.

#### **Step 2: RF chokes**

These are not polarized and can go in either way. Make sure they sit flat against the board before soldering.

#### **Step 3: Capacitors**

Install the capacitors with minimal lead length. The electrolytic is the only polarized capacitor and the positive lead points towards the edge of the PCB.

#### **Step 4: IC socket**

The IC socket is installed next with the notch facing the right side of the PCB.

#### **Step 5: Crystal oscillator module**

Ensure pin 1 of the oscillator module faces the bottom left of the PCB.

#### **Step 6: 5 volt regulator**

The regulator lays flat against the PCB so needs the leads bent at right angle to fit through the holes in the PCB. Insert into the PCB holes and secure with a 3mm screw and nut before soldering the leads.

#### **Step 7: Connectors**

The Output connector is a 2 pin right angle header while the Power connector is a 4 pin right angle header. Push them into the PCB so they lay flat against the board and then solder.

The ISP header is an un-shrouded 3 pin x 2 row header and the shorter pins are inserted into the PCB and soldered.

The LCD connector is a 16 way male pin header and is mounted on the reverse side with the short ends of the pins located in the PCB. Ensure it is pushed in completely and solder the end pins first and check it is at right angle to the PCB before soldering the remaining pins.

#### **Step 8: Rotary encoder**

The rotary encoder is mounted on the reverse side. Insert into the holes and check that it is at right angle to the board before soldering. It is not necessary to completely fill the mechanical mounting holes in the PCB with solder.

#### **Step 9: Microcontroller**

Remove the microcontroller from the bag and insert into the IC socket ensuring pin 1 points to the top right of the PCB towards the contrast control. It may be necessary to squeeze the rows of pins slightly together to get it to fit into the socket.

#### **Step 10: LCD**

Solder a 16 way female pin header to the rear side of the LCD PCB.

The LCD can then be mounted onto the DDS VFO and mate with the 16 way male pin header.

Place four 12mm long nylon spacers between the LCD and DDS VFO PCB at each corner and secure the assembly with 2.5mm screws and nuts.

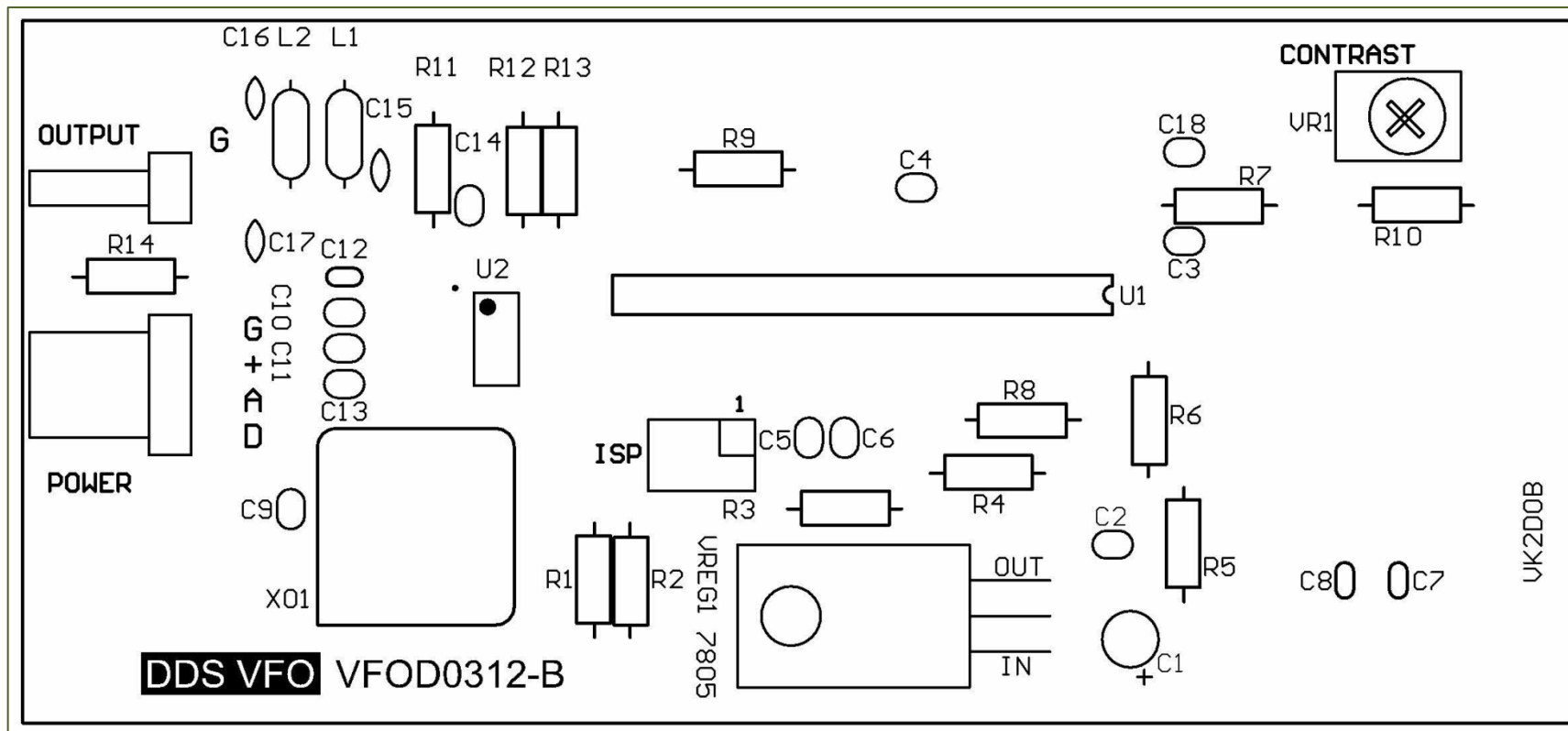


Figure 2 Component overlay

## 5 TESTING

Before applying power check the board over one more time. Look for solder bridges and components in the wrong way. A moment spent here may save a lot of frustrating time later on.

Once you are satisfied connect the DDS VFO to a power supply between 8 and 15V DC. If the power supply has current limiting set this to about 200mA.

Apply power and check that the LCD backlight turns on. Also check with a multimeter that the output of the regulator is between 4.75 and 5.25V DC. The power supply current should be around 80mA, but this will depend on the exact LCD you have installed and it's backlight characteristics.

**If the readings are way outside this range turn off immediately and look for problems.**

At this stage you may or may not see anything on the LCD. Adjust the contrast trimpot until the characters are clear and easy to read.

Turn the power off then on again and check that the first thing displayed is the firmware version number. This is then followed by the operating display.

As the DDS VFO has not been configured the LCD will display default settings. These are Transmit frequency of 7.1MHz and a 1KHz Step. The default IF frequency is 10MHz with a Minus offset so using an oscilloscope or a frequency counter verify that the DDS is outputting a 2.9MHz sine wave signal of around 300mV pk-pk.

## 6 CONFIGURATION

### 6.1 UNDERSTANDING OPTIONS

#### **IF offset**

Consider a transmitter using a Superhet mixing arrangement. The transmit frequency is a result of the IF frequency being mixed with a VFO frequency. If for example the IF carrier frequency is 10MHz and the desired transmit frequency is 7.1MHz then the VFO frequency can be either:

1.  $10 - 7.1 = 2.9\text{MHz}$  using a Minus offset
2.  $10 + 7.1 = 17.1\text{MHz}$  using a Plus offset

It is important to understand that when the DDS VFO is configured with an IF offset the frequency displayed on the LCD is not equal to the DDS output frequency.

If, however, the VFO DDS is to be used as a general purpose signal generator or with a Direct Conversion receiver, the IF offset can be disabled. In this case the displayed frequency will be the same as the DDS output frequency.

## TX frequency

The TX (Transmit) frequency is the initial frequency displayed on the LCD at power up. The VFO DDS firmware uses this value and the IF carrier frequency as well as the IF offset to compute the actual DDS output frequency.

## IF frequency

To generate an accurate output frequency using the equations above, the IF carrier frequency must be measured and the value entered at configuration. If not the displayed frequency may not match the transmit frequency.

## 30 second frequency save

This option enables the automatic saving of the current frequency and step in EEPROM so that it is recalled the next time the DDS VFO is powered on. This overrides the programmed TX frequency. The EEPROM has a maximum specification of 100,000 write/erase cycles, so to protect it from excessive writes, the save is performed only once per frequency change and only after a 30 second idle period has elapsed. If the encoder is operated so that there is less than 30 seconds between changes no saves will be performed.

## 6.2 PROCEDURE

To program the IF frequency, the Transmit frequency and the IF offset type use the following procedure:

1. Hold down the encoder button and apply power to the DDS VFO.
2. Wait until the LCD displays Configuration (about 3 seconds) and then release the button.
3. The IF offset screen will be displayed. Pressing the encoder button for less than 1 second will toggle through the options. Once completed press the encoder button for longer than 1 second and release.
4. The IF frequency screen will be displayed and the cursor will be positioned on the 10MHz digit. Rotating the encoder clockwise increments the digit while rotating the encoder anti-clockwise decrements the digit. Pressing the encoder button for less than 1 second will step the cursor onto the next digit. Once completed press the encoder button for longer than 1 second and release. **Note that if the IF offset has been disabled this screen will be bypassed as it becomes irrelevant.**
5. The Transmit frequency screen will be displayed and the cursor will be positioned on the 10MHz digit. Rotating the encoder clockwise increments the digit while rotating the encoder anti-clockwise decrements the digit. Pressing the encoder button for less than 1 second will step the cursor onto the next digit. Once completed press the encoder button for longer than 1 second and release.
6. The 30 second save frequency screen will be displayed. Pressing the encoder button for less than 1 second will toggle between the Yes and No options. Once completed press the encoder button for longer than 1 second and release.
7. The DDS VFO will then store the values in non-volatile memory and return to normal operation.

## 7 OPERATION

### Power on

When the DDS VFO is powered on the version number is displayed briefly. This is followed by the operating display shown in Figure 3.

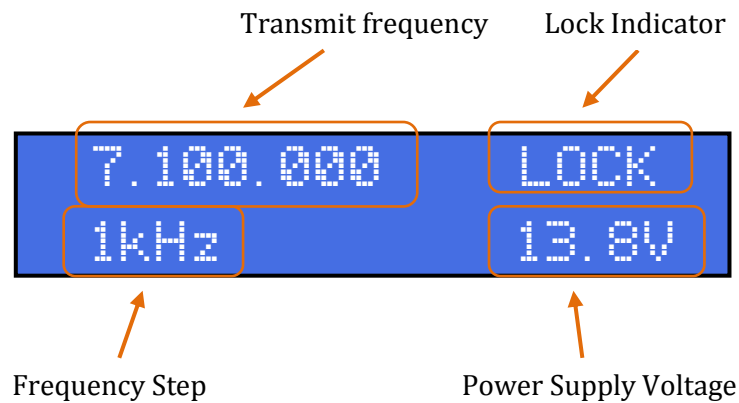


Figure 3 Typical LCD operating display

### Tuning

To tune the DDS VFO rotate the encoder clockwise to raise the frequency and counter-clockwise to lower the frequency. The frequency will change by an amount equal to the Step value.

### Frequency steps

To change the current frequency step press the encoder push button for less than 1 second and release. Each press cycles through the available steps which are 1KHz, 100HZ and 10Hz.

### Tuning Lock

To lock the encoder from changing frequency press the encoder push button for more than 1 second. A lock message will be displayed on the upper right of the LCD.

To unlock press the encoder push button for less than 1 second.

### Voltage Display

The power supply voltage is displayed in the lower right hand side of the LCD. This is updated every 500mS. If the voltage falls below 8V a low voltage warning message is displayed to indicate the power supply is too low for proper operation.

## 8 OPTIONS

### LCD

The DDS VFO operates best with a backlit display. The characters are bright and clear and readable in dim environments. The backlight does consume extra current and if minimizing current consumption is a priority then a non-backlit LCD can be used instead. In this case resistor R9 can be left off the board, although leaving it in will not cause any issues.

If an LCD with backlight is used, but is not the same as listed in the parts list, the backlight may require a different value of current limiting resistor. Check the LCD data sheet before applying power to determine the correct resistor value and change R9 if needed.

A non-backlit display will also not be as deep and this needs to be taken into account when mounting to the front panel.

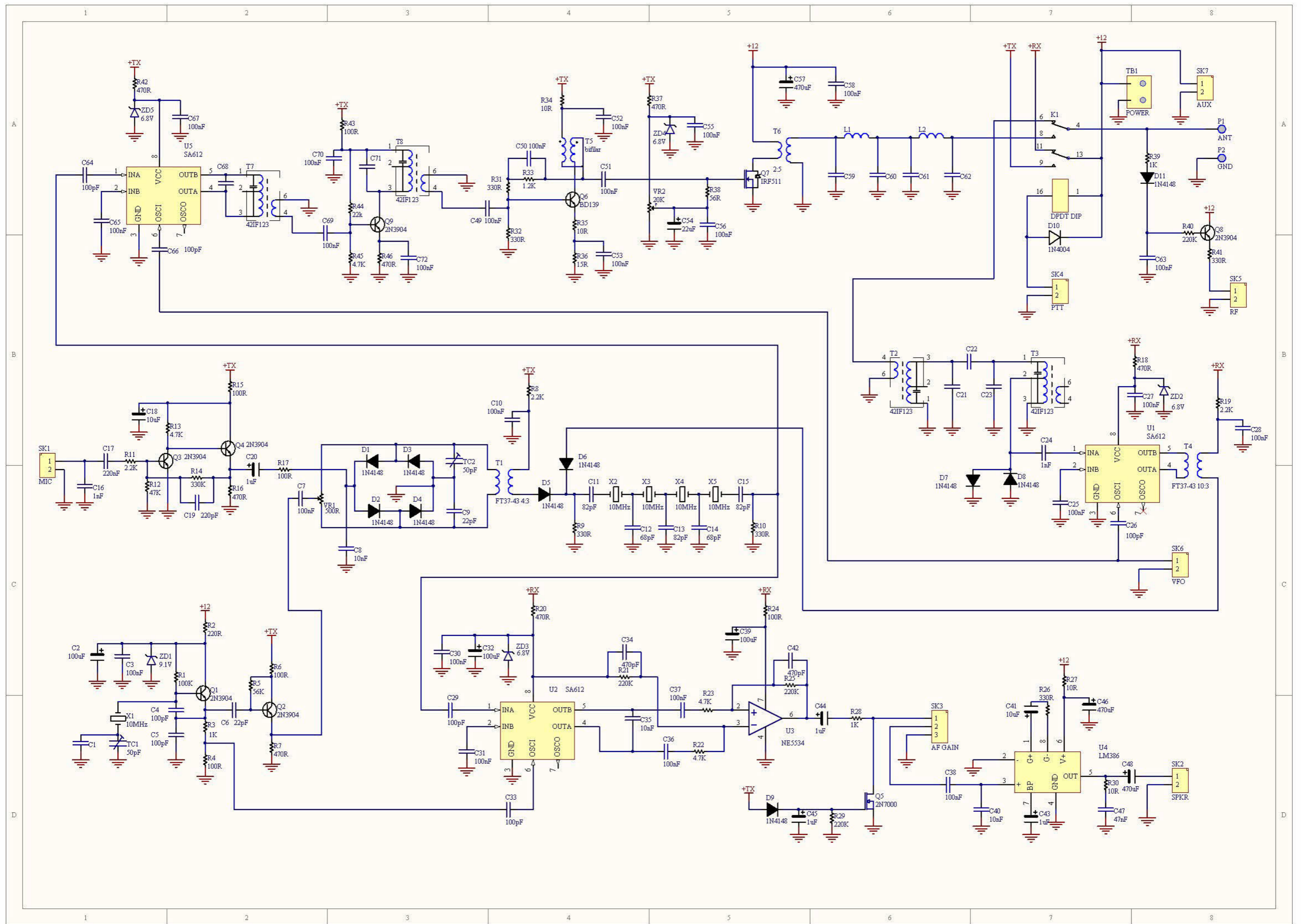
### Additional inputs

The PCB includes two additional inputs. One is intended to measure an analog voltage and the other is configured as a digital input. Both are able to handle a voltage between 0 and 20V DC maximum. The current firmware release does not utilize these inputs and so when making the 4 pin power cable these pins can be left unterminated.

### Firmware upgrades

The PCB contains a functioning 6 pin ISP header to allow future firmware upgrades.







# MST400 SSB TRANSCEIVER

TUNE



AF GAIN



TX



MIC GAIN



MIC



# MST800 SSB TRANSCEIVER

TUNE



AF GAIN



TX



MIC GAIN



MIC



