



The 'Little Roo' 40m SSB or CW Pocket Receiver

The Little Roo is designed to be a fun to build mini 40m receiver that covers a useful chunk of the 40m SSB sub band or just about all the CW portion of 40m. Typical cover range will be from 7.1MHz to just under 7.185MHz if you build it for SSB use or 7.0MHz to 7.070MHz if you build for CW reception.

The little Roo is built into a mint tin style case and is powered by a standard PP3 9V internal battery, you will just need headphones (or a small speaker) and a wire antenna to start having fun with this super little receiver. There are no surface mount parts to fit but due to the component density on the PCB this may not be suitable for the complete beginner.



First lets check we have all the parts :-

Part	Value	Part	Value
C1	100pf C1B 30pf	R6	10K ¼ Watt (Black, Brown, Orange)
C2	1nf	LOW	20K Blue Trimmer
C3	8.2pf	HIGH	20K Blue Trimmer
C4a	100pf C4B 8.2pf	GAIN	10K Potentiometer
C5	1nf	TUNE	10K Potentiometer
C6	200pf	L1	3.3uH (Brown Body Inductor)
C7	200pf	L2	3.3uH (Brown Body Inductor)
C'x'	47pf (SSB) or 150pf (CW)	L3	3.9uH (Green Body Inductor)
C8	0.1uf	NE602	Can be Alternative (See Instructions)
C9	0.1uf	LM386	Audio Amplifier
C10	0.022uf	78L06	6v Voltage Regulator
C11	0.1uf	D1	BB910 (SSB)/ ISV149 (CW)
C12	0.1uf	D2	1N4148
C13	100uf 16v DC (Min)	D3	Link
C14	10uf 16v DC (Min)	LED	3mm Orange
C15	10uF 16v DC (Min)	SWT	ON/OFF PCB Switch
C16	10uf 16 DC (Min)	AUDIO	3.5mm Jack Socket
C17	0.1uf	BNC	PCB Vertical BNC Socket
C18	0.1uf	PCB	Main PCB Board
C19	0.1uf	PWR	2 Way Terminal Block
R1	10K ¼ Watt (Black, Brown, Orange)	CASE	Mint Tin Case
R2	100K ¼ Watt (Black, Brown, Yellow)	J1	3Pin Header & Jumper
R3	10 ohm ¼ Watt (Black, Brown,Brown)	INSERT	Blank Clear PCB Insulator
R4	10 ohm ¼ Watt (Black, Brown,Brown)	STICKER	Little Roo Case Label
R5	33K ¼ Watt (Orange, Orange,Orange)		



KANGA PRODUCTS

Electronic Kits For The Radio Amateur

To make the Kit easy to build we have broken it down into a number of stages, the parts for each stage are bagged separately.

The Kit can be built for either SSB Sub Band or for CW Sub band, one but NOT both. You need to choose and follow any instructions given for your choice.

Stage 1 The Voltage regulator

Both versions share this stage.

This stages components are:

Part ID	Part	Fitted
R6	10K ¼ Watt (Brown, Black, Orange)	
LOW	20k Blue Trimmer	
HIGH	20K Blue Trimmer	
TUNE	10K Potentiometer (marked 103 on back)	
C19	0.1uf (Marked 104)	
SWT	Mini PCB Power Switch	
PWR	2 Way terminal Block	
78L06	6V Voltage Regulator	
D2	1N4148	
D3	LINK (Off cut from Resistor)	
PP3 Clip	9V Battery Clip	
LED	3mm Orange power on LED	
J1	3 pin Header and 2 way jumper	

Tick off each part as you fit it.

Lets start fitting the resistors

Fit R6 10K resistor (Bands are Brown, Black, Orange)

The resistor can be fitted either way round, make sure it is fitted flush to the board and trim the legs when soldered nice and short (flush) to the board.

When fitted tick it off the table above.

Next let's fit the only capacitor in this stage

C19, this is a yellow small capacitor marked 104 on one side. Again, this can be fitted either way round. Once fitted tick it off the table.



Now let's fit a difficult part of this stage, the vertical mounted diode.

Do not rush ahead and just fit the diode without reading this!

Although we have positions for two diodes, I am going to suggest that you only fit one (D2) and use a wire link made from a resistor off-cut in place of the other diode (D3).

Two diodes will give a few KHz more tuning range at the high frequency end BUT as the battery voltage drops the voltage regulator will not supply a stable voltage to the tuning circuit, this will give an effect like chirp on strong signals. If you power the receiver from an external 12v source then ok two diodes will be fine but if you use this with a standard PP3 9V battery then you will get better quality reception with just one fitted, I will assume you will be fitting just one from this point on.



The diode is a 1N4148 which is one of the most common small signal diodes, there are a few alternatives to this and its possible that you will receive one of these but they look the same and have the same specification too.

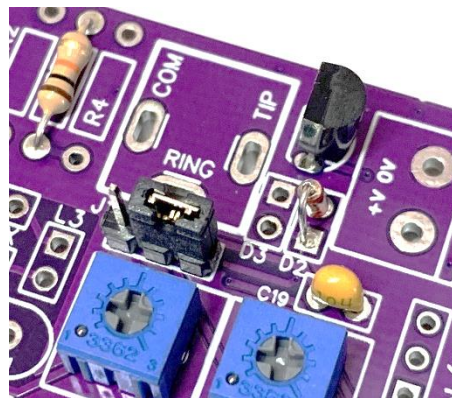
First, we need to bend the legs of the diode as it is fitted vertically.

If you look at the diode you will see one end has a black band.

Bend the leg on this side back so its parallel to the other legs as in the picture here.

The picture on the right shows the location of D2 and D3. The labelling of these locations is small so look closely.

You will see that the two positions for the diodes have a white rectangle around them, in the rectangle you have a square pad and a round pad.



Put the diode into the D2 position so that the leg from the black banded side goes into the SQUARE Pads hole.

Now very important, fit In the D3 position fit a wire link made from a resistor off cut.

Next we can fit the tuning voltage selector headpins and jumper, this is located just above the position for the position for the headphone socket, to the left of the diode you just fitted. Fit this and put the jumper between pins 2 and 3 (more on this later).

Now, we can fit the Voltage Regulator, this is the 3-pin black device that looks like a transistor. But its marked 78L06 on the flats side.

The position for this is just above the diode we have just fitted, the PCB shows the outline of the device when fitted so make sure you insert the regulator as the silk screen shape, it doesn't need to be pushed right down to the PCB so leave a few mm of component leg



above the board when its fitted. Again, once fitted cross it of the stage component list above.

Now we can fit two small blue trimmers.

Both trimmers are the same value so you can use either in any position. Make sure that they are sat nice and flush before soldering.

Insert them into the positions on the board marked LOW and HIGH.

These LOW and HIGH trimmers control the voltage limits applied to the tuning diode and so are part of the Voltage regulation circuit.

Turn both trimmers fully **anticlockwise**.

With the controls set like this you will have maximum tuning range.

At a later stage if you wish to reduce the tuning range you can adjust these to cut the HIGH and the LOW RF frequency limits, if you do make an adjustment later, I suggest adjusting the HIGH trimmer first as its adjustment will change the setting of the LOW trimmer.

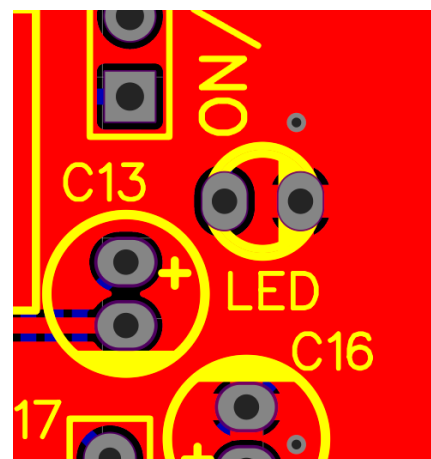
Again, mark these off when fitted.

Just a few more parts left for this stage, Lets fit the small PCB switch. The switch (marked ON/OFF on the PCB) can be fitted either way round, I push the switch right the way down to the PCB but if you wish you can leave it standing higher, it may be easier to get to if you do but I find no problem with it mount fully down.

Now let's fit the power on LED. This is a small 3mm orange LED, you will notice that it has 2 legs of different lengths. The shorter leg must be inserted into the hole on the right-hand side of the PCB symbol.

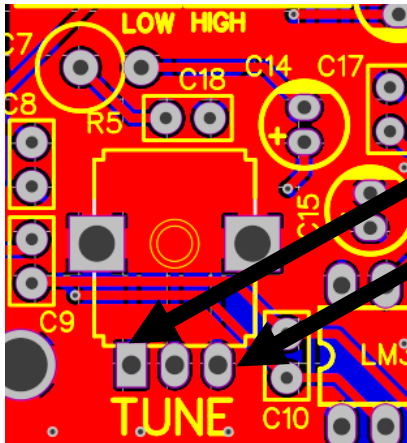
The next part for this stage is the two-way terminal block which is located just above the power switch position.

Make sure when you fit it so that the cable entry points are on the right-hand side as we look down on the board, that's where the battery clip will connect to when we are ready to use our receiver.





That almost completes the first stage, if you want to do a very basic check now then fit the battery clip to the terminal block (make sure you get the polarity right!) and slide the power switch to the ON position, you should get the LED on. If you have a test meter you can place the probes across the tuning control pads and check the voltage. It should be around 6.6V



Look across these pads for the test voltage (6.6v approx.)

Once you're happy fit the tuning control which is a 10K potentiometer, there are two potentiometers in the kit but they are NOT the same, on the back of them (the bottom) you will see a marking, it will say either B1-03 or B1-02, we need the B1-03. Solder in place and also solder the two mounting lugs to hold the control on the PCB.

This completes Stage one build.



Stage 2 The Audio Amplifier

This stage is also the same for both versions.

This stages components are:-

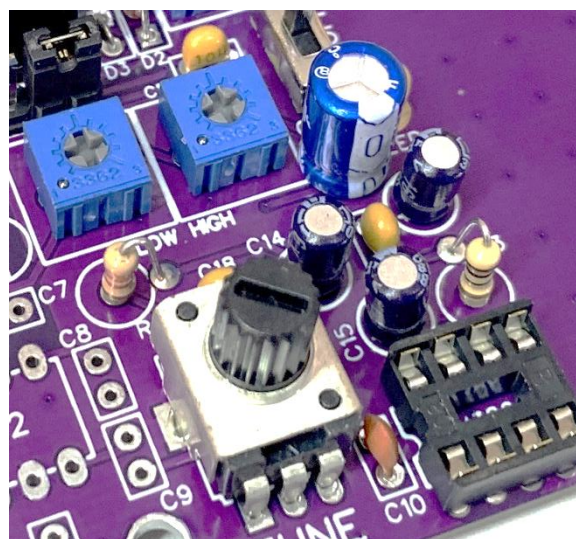
Part ID	Part	Fitted
C10	0.022uf (marked 223)	
C13	100uf 16v	
C14	10uf 16v	
C15	10uf 16v	
C16	10uf 16v	
C17	0.1uf (marked 104)	
C18	0.1uf (marked 104)	
R3	10 Ohm (Brown, Black, Black)	
R4	10 Ohm (Brown, Black, Black)	
R5	33K (Orange, Orange, Orange)	
DIP8	DIP 8 Way IC socket for LM386 (See Notes)	
LM386	Audio Amplifier (See Notes)	
AUDIO SKT	3.5mm Vertical Audio Socket	

****Note****

The LM386, its socket and other parts for the next stage are in a separate little RED anti-Static bag with the kit, remove the chip and socket but put the other parts on one side for the next stage.

Tick off the parts as you fit them.

The audio amplifier in the 'Little Roo' uses a well know and reliable audio amplifier circuit. We have also incorporated a simple but effective hiss reduction technique.



This stage will drive a small loudspeaker if you wish or standard stereo headphones.

First I suggest fitting the 8 way IC socket, the socket has a small notch on one edge, align this edge with silk screen layout for the socket.

Next fit the resistors

R3, R4, and R5. Make sure you double check the value of each part before you fit it, resistors can be fitted either way round. Note R3 and R5 are fitted vertical on the PCB to save space.



Next The capacitors.

This stage has both standard ceramic capacitors and larger electrolytic types

First fit C17 and C18 which are both 0.1uf Yellow capacitors marked 104 on one side.

Next fit C10 which is a 0.022uf capacitor marked 223

Now let's fit the electrolytic capacitors

C13 is a 100uf capacitor with a minimum voltage rating of 16v DC, these capacitors must be fitted the correct way round. If you look at the body of these capacitors you will see a white or grey strip down the body on one side, this is the negative side. The negative leg is shorter than the other.

The long leg is the positive leg and this MUST be fitted so it goes into the hole with the '+' sign next to it

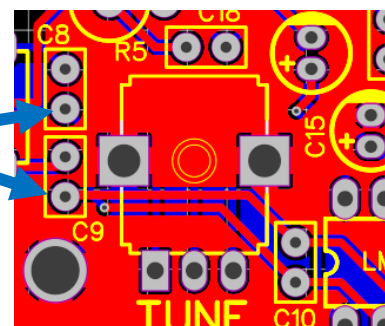
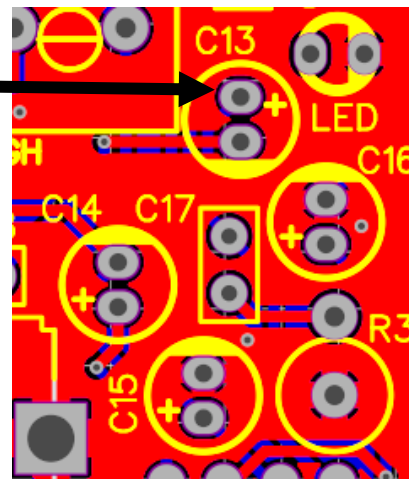
The same rule applies to all the other electrolytic capacitors too. Fit C14, C15, and C16.

Finally fit the vertical audio headphone socket, this can only fit one way. Just make sure it's pushed fully down onto the PCB before soldering.

If you wish we can now test the audio stage. To do so plug the LM386 chip into the 8 way socket, make sure it's the right way round. The small dot on top of the IC shows pin 1, the chip should be fitted so that dot is nearest the markings for C10 on the board.

Temporarily connect the battery for the next test.

Now plug in the headphones and turn on the receiver, you should see the orange LED light and hear a faint hiss in the headphones. The amplifier doesn't produce lots of hiss has it's really designed not to! so you may want to touch a metal screwdriver to the bottom pads for C8 and C9. You should hear a Buzz now if you touch the screwdrivers blade. If so, great the audio amplifier is working! Turn off the receiver and we can now move onto the next stage.





Stage 3 The Mixer

The mixer is the heart of the radio.

IMPORTANT

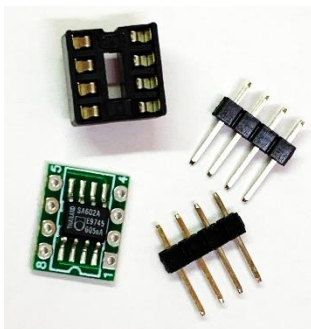
This stage that will have different parts depending on which version you are building so take care to follow the correct instructions.

The main chip that does all work is the NE602, there are a few variations on this that will all work the same. I now have to supply you a Surface mounted version as supplies of this chip are currently very difficult to obtain. Don't worry, I have preinstalled it onto a little adapter board. You will need to solder pins to the adapter board but then you can just treat it the same as the normal chip.

This stage components are :-

Parts ID	Part	Fitted
R1	10k ¼ Watt (marked Brown, Black, Orange)	
R2	100K ¼ Watt (marked Brown, Black, Yellow)	
C6	200pf (marked 201)	
C7	200pf (marked 201)	
C8	0.1uf (marked 104)	
C9	0.1uf (marked 104)	
CX	47pf (marked 470) and 150pf (marked 151)	
C11	0.1uf (marked 104)	
C12	0.1uf (marked 104)	
Y1	7.2MHz resonator	
D1	BB910 or IVS149 Varicap Diode	
L3	3.9uH Green moulded inductor	
DIP8	8 way IC socket	
NE602	Mixer chip (see Instructions below)	

Preparing the Mixer chip adapter board.



The parts for this will be left in that little Red antistatic bag you opened in the last stage.

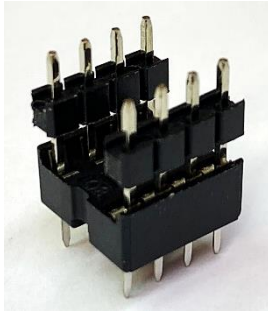
First let's fit the 8-way IC socket. Align the socket to match the outline on the PCB.

Now I suggest that you push the two 4 way pin strips into the socket, the long side of the pins should be in the socket.



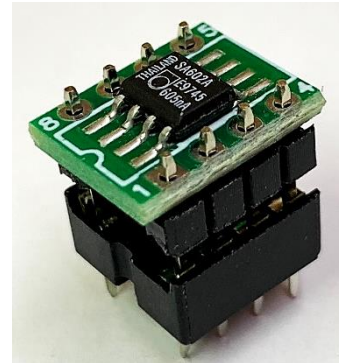
KANGA PRODUCTS

Electronic Kits For The Radio Amateur



Now drop the adapter board on top of the pins. Take care and solder each pin to the adapter board. Now you can treat the assembly the same as a standard chip. Make sure that the 'chip' is plugged into the socket the right way round, if you look at the top of the chips board you will see the printed

outline of a standard chip with a notch at one end, this end should be nearest to the notch on the socket.



Well done, that's the hardest part done.

Again, when a part is fitted tick it off the list

Much of the mixer is still the same regardless on version you're building.

Now let's fit the two resistors R1 and R2.

These can be fitted either way round.

R1 is the 10k resistor (coloured Brown, Black, Orange)

R2 is a 100K resistor (Coloured Brown, Black, Yellow)

Next we need to fit a few capacitors

C8, C9, C11, and C12 are all the same value, 0.1uf they are marked 104 on one side.

Now the next capacitors control the frequency range we can achieve with the ceramic resonator you have been provide with. It's important that you stick to these values for the coverage we have specified.

Let's fit these parts.

C6 is a 200pf capacitor marked 201

C7 is a 200pf capacitor marked 201

IMPORTANT SSB or CW part follows!

SSB Only

Cx is critical in controlling the frequency we are receiving, if you are building this for SSB use then Cx is 47pf (Marked 470).

CW Only

If your building this for CW use then Cx needs to be 150pf (Marked 151) .

Both capacitors are with the kit.



All the capacitors in the RF side of the mixer (all the ones below 0.1uf in value) should be NPO/COG types if you ever change them.

Now we need to fit a few more parts of different values for either a SSB radio or a CW radio.

Follow the section that matches your build requirements.

SSB Build ONLY

Now we will fit L3, For the frequency coverage we have stated you will need to use the 3.9uh green moulded inductor. Using higher value inductance here will lower the bottom received frequency but at the risk of making the drift a problem, feel free to experiment if you so wish, values of around 3-15uH will give interesting results.

Next fit the diode D1, this is a special diode, it is a varcap diode so it acts as the tuning element in the radio, when we change the voltage applied to this diode by the tune control we get a change in capacitance that 'pulls' the frequency of the VFO.

This diode looks like the voltage regulator we fitted before but only have two legs.

One side of the diode is flat, that side will have BB910 printed on it. This flat side MUST be nearest to the position for the resonator Y1.

CW Build only

For L3 we only need to install a short link wire made from a off cut of wire from a resistor. The supplied 4.7uH inductor is NOT needed for this version.

Now the CW version uses a different vari-cap diode, we have also supplied you with a ISV149 diode, this again is a 2 pin diode and looks much slimmer than the other SSB diode. You will see that it has a large flat side. That side will have ISV149 written on it. This larger flat side MUST be nearest to the position for Y1.

Now back to the common to both version parts.

Now we can fit the ceramic resonator Y1

This has three legs, most resonators have only two and we are only using two in this circuit, the third leg will just be used to give more stability to the mounting of this part and although it is soldered to a pad it is not connected electrically.

The resonator can be fitted either way round.

That Completes the wiring of the Mixer.



The final stage is next

Stage 4 The Band Pass Filter

This is a very important part to get right, the airwaves are full of signals right the way across the RF spectrum, they would all be fed into the mixer without the band pass filter and the weak amateur signals we want to hear would be wiped out without some good filtering here.

We are using some higher 'Q' moulded inductors here that form a good narrow bandwidth filter for our 40m receiver. We are using good quality NPO capacitors too and we have found the bandpass filter to be very good even with all the QRM from strong European broadcast stations,

Let's look at the parts for this stage :-

Part ID	Part	Fitted
C1A & C1B	C1A 100pf (marked 101) C1B 30pf (Brown with purple tip)	
C2	1nf (marked 102)	
C3	8.2pf (marked 8.2)	
C4A & C4B	C4a 100pf (marked 101) C4B 8.2pf	
C5	1nf (marked 102)	
L1	3.3uH	
L2	3.3uH	
Gain Control	1K Potentiometer (marked B1-02 on the back)	
BNC Socket	PCB Mount vertical Socket	

Mark parts off list as you fit them.

Lets start with the capacitors

C1A/B, C2, C3, C4A/B, and C5 make sure that you fit the right value in the right place, double check their value from the table above.

Next fit the two inductors. The inductors should be fitted flush with the PCB and not sticking up.

Now fit the gain control, this is a 1K potentiometer that looks the same as the Tune control but will have B1-02 printed on the bottom.

Finally fit the vertical BNC socket. This Socket is like a BIG heatsink, it may be difficult to solder so make sure your soldering iron is up to full temperature before soldering it and expect to have to leave your iron on the 4 earth lugs for at least 10 secs per joint to get the solder to flow. Remember, that connector will be **very hot** for a while afterwards so be careful not to touch it till it cools.



That completes the final stage.

Now don't rush to power it up, lets double check for any dry joints or solder bridges between pins. If it looks good it's time to fit the mixer chip the NE602. Make sure that you fit it the right way round.

Now connect an antenna to the BNC socket, adjust the gain control fully clockwise. Insert the headphones or even connect a small speaker. Now turn on the Little Roo.

Normally you will instantly here band noise, adjust the tuning control from one end of its travel to the other, fully anticlockwise you will typically be around 7.090Mhz (SSB Build) or about 6.990Mhz (CW Build), turn the tuning control fully clockwise and you will be around 7.185Mhz(SSB Build) or 7.070Mhz(CW Build). You should be able to find a number of stations if the time of day is allowing reasonable propagation to your area.

If you wish you can leave things just as they are but you may wish (if your building the CW version) to adjust the bottom frequency up to the 40m bottom frequency of 7Mhz. To do so adjust the LOW blue trimmer to bring the bottom frequency up to 7Mhz, that will be about 3rd of a turn in clockwise direction but for now you can test the radio as is.

Now the 'Little Roo' is a Direct conversion receiver (DC Receiver) , like ALL DC receivers the receiver will pickup both sidebands of a transmission but you will only be able to understand one side in SSB use, normally on 40m that will be lower sideband, you will find it better to start at the high frequency end of the tuning range and tune down, you will come across the LSB signal first which you should be able to tune into and understand, care while tuning will be needed as the correct voice pitch can be tricky to find at first. If you have built this for CW use then like in the SSB version you will hear a CW signal both sides of the carrier frequency, tune to which ever one has less QRM is the best option. The 'little Roo' has good stability but you may find you need a couple of minutes of operation before the VFO reaches its best stability.

I find after a few minutes of use that I can listen to a net without the constant need to touch up the tuning on ever over.

When your happy that the radio is working correctly then put the supplied clear plastic insulation card in the bottom of the 'mint' tin case, drop the main receiver board in there on top of the insulator. I trim the 9v battery lead now so I don't have a lot of wire floating around the case, I suggest about 1 ½ inch's (about 40mm for you youngsters). I have put a little bit of double-sided tape or 'blue tack' (not much or you will never get it off!) on the back of my battery and placed it on the right side of the board in the case. The lid of the case should close without problem.

You have a 'Little Roo' sticker that can now be applied to the top of the case. Take care and take your time with doing this as you want to get a good finish.

I hope you enjoyed building the little Roo and find it works well for you. 73 Paul