Overview

The Rev3.02 is a high quality phase-locked-loop (PLL) based FM transmitter designed to operate in the frequency range 87.5 to 108 MHz. Unlike many cheaper designs, the Rev3.02 uses a half-frequency oscillator, meaning that even when amplified to high powers, the impact of any radio frequency (RF) feedback is minimised, improving stability and reducing buzz.

The output stage will produce around 500 milliWatts with a 12 Volt supply rising to 1 Watt with a maximum 15 Volt supply across the whole frequency range. The circuit uses off-the-shelf components meaning that any repairs or replacements can be made from readily available parts.

Compared to many other similar transmitters, the Rev3.02 uses a very low PLL loop frequency. This means that the PLL does not attempt to 'correct' bass modulation and together with the careful design of the circuit, means that the frequency response is flat to from around 1 Hz to over 100 kHz, ensuring very clean and accurate modulation.

An out-of-lock power down circuit ensures that there is no output from the transmitter until the PLL is locked, preventing emissions on frequencies other than that to which it is set (which can be disastrous for high power amplifiers when connected to a narrow-band antenna).

The output of the transmitter is filtered by a 7 pole low pass filter, reducing all spurious outputs to at least 60 dB below the main carrier.

Many of the components associated with the transmitter have already been surface mounted onto the high quality printed circuit board, making construction faster, simpler and making performance more reliable.

Please read the instructions in full before beginning construction (especially noting the fitting of VC1).

Important Notice: A licence is required to connect this transmitter to an antenna. Please ensure that you have the necessary authorisations before using this device with an antenna.

Construction

To construct the transmitter you will need:

- A soldering iron, preferably with a narrow tip
- Solder
- A pair of snips or side cutters to trim component leads once devices are soldered to the board
- A 4mm former for winding coils (the smooth end of a 4mm drill bit is ideal for this)

Check that you have all the parts by comparing those supplied with the parts list and familiarise yourself with which components are which.

In general, when loading components into a PCB, the principle used is to start with those with the shortest profile (i.e. resistors) and end with those with the tallest profile (i.e. electrolytic capacitors). All components should be mounted as flush to the PCB as possible.

Start by fitting all of the resistors. Help with identifying resistor colour codes can be found online at:

- www.wirelesswaffle.com/resistors/

At the beginning, take care not to damage any of the surface mount devices (SMD). Once you have a few resistors soldered to the board, when turning the board upside down to solder new components, there will be far less chance of damaging any of the SMD.

Next fit the two varicap diodes, DV1 and DV2, the crystal (X1) and the four TO-92 style transistors (Q2, 3, 4 and 6). Note the orientation of the transistors as marked on the board and mount them as close to the board as possible. Q3 and Q6 can be pushed all the way into the board such that none of the metal of their legs appears on the top.

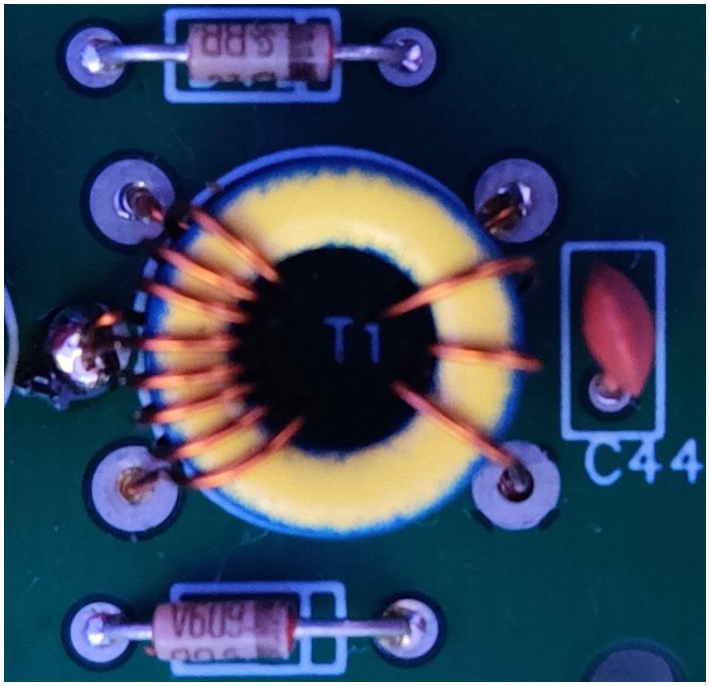
The small ceramic capacitors should be installed next (not the tall electrolytics), and inductor L4 which looks like a slightly thicker resistor.

The four 16-pin integrated circuits (ICs) should be mounted next (U2, 3, 4 and 5). Take careful notice of the orientation of these devices. U3 and 4 are mounted facing the opposite direction to U2 and U5. The ICs should be oriented such that the dot or semi-circle at one end of the chip is at the same end as the dot marked on the PCB. These devices can be sensitive to static so it is a good practice to handle the ICs as little as possible, or use static handling precautions if you have them.

Transformer T1 is critical to the correct functioning of the transmitter and careful attention must be paid to winding it. Begin by taking the thinner enamelled copper wire provided and folding it over on itself in the middle (creating a tight 'U' turn). Strip the enamel from the centre of the wire where it has been folded and tin it with solder.

Starting from the centre point you have created, tightly wind 4 turns in each direction around the toroid core, such that you end up with a total of 8 turns, with the centre you formed in the very middle (i.e. 8 turns centre tapped). Cut off excess wire leaving enough to solder the ends to the PCB. Remove the enamel from the ends left on the core and tin these with solder to make them easier to mount on the PCB.

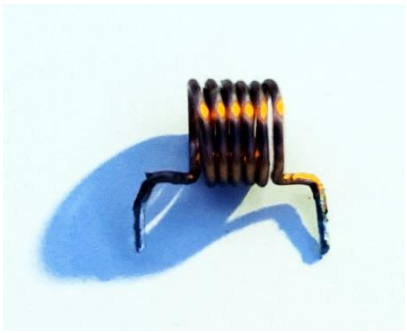
The secondary of the transformer is 3 turns wound on the opposite side of the toroidal core to the winding you just made (use a piece of the remaining thinner copper wire). It does not matter whether these 3 turns are wound in the same direction as the previous winding or not. Again, remove the enamel and tin the ends of the winding ready for soldering to the PCB. The completed transformer, when mounted on the PCB should appear as in the picture below, with 8 turns showing on the main winding and 3 on the secondary.



Next fit the three light emitting diodes (LEDs) again carefully noting their orientation. There is a small 'flat' on one side of the LED and this should align with the flat drawn on the PCB. It does not matter which colour is used for which LED.

Fit the 10 way DIP switch such that the word 'ON' appears towards the outer edge of the PCB.

It is now time to wind the five coils. Three of these (L1, 2 and 5) have 6 turns, and two (L3 and L6) have 5 turns. All of the coils should be wound tightly on a 4mm former (such as the shaft of a 4mm drill bit, or a piece of 4mm wooden dowel). Leave enough additional wire at the end of each coil to form legs that can be used to mount them on the board. Remember to strip the insulation off each end of the enamelled wire so that it can be soldered. The finished coils should look something like the picture below (the one shown has 6 turns).



Fit the electrolytic capacitors, being careful to ensure that they are correctly orientated with the negative (marked on the side of the capacitors with a stripe) fitted such they are in-line with the solid bar marked on the PCB.

Next fit the voltage regulator (U1) which should be mounted so that the metal tag faces towards the edge of the board.

Fit transistor Q5 (2N4427). This should be mounted so that it is flush with the top of the PCB. This transistor needs to be fitted with the supplied heatsink. The easiest way to do this is that once the transistor is firmly mounted on the board, the heatsink can be prised ever so slightly open using a screwdriver and slipped gently over the transistor. The screwdriver can then be removed such that the heatsink now clasps tightly against the transistor.

Finally, install the three 2-pin headers used for the power supply, audio in, and RF out.

AT THIS POINT DO NOT FIT VC1.

You should now have mounted all the components on the PCB (with the exception of VC1).

Testing

Connect a 50 Ohm load to the RF port of the transmitter and connect a 12V source to the 12V input. Set all the DIP switches to 'OFF' - this will stop the PLL locking and allow voltages to be measured without any RF output being generated. 12 Volts should be applied to the pin nearest the '+12V' label and ground should be applied to the other. Be very careful with this as connecting power the wrong way around could damage the components on the PCB.

Briefly apply power and check that the '+8V' LED lights.

If the '+8V' LED does not light check the polarity of the power supply (and of the LED!) If these are correct, it is possible that there is a short-circuit somewhere on the board. Check the soldering carefully, especially between the pins of Q3 and Q6 which are very close together. If any shorts are found, clear them and try applying power again.

Measure the voltage on the +8V rail (i.e. on pin 16 of U2) which should measure nearer to 9 Volts. The +5V rail (i.e. on pin 16 of U4) should measure between 5.5 and 6 Volts.

Once it is clear that power is reaching the circuit (i.e. the '+8V' LED lights when power is attached) and the voltages are correct, set all of the switches on the DIP switch to 'OFF' with the exception of switch 10 (the opposite end of where R4 is fitted, and adjacent to U4), which should be set to 'ON'. This sets the transmitter to a frequency of 102.4 MHz for testing purposes.

Apply power again and the 'LOCK' LED should eventually light. It can take up to 10 seconds from the application of power to the 'LOCK' LED lighting. If it has not lit after 15 seconds, it is likely that there is a fault on the board. Go to the 'Troubleshooting' section if this is the case.

In addition to the 'LOCK' LED lighting, the 'PWR' LED may also light, though it is often dimmer than the other two LEDs.

The purpose of the three LEDs is as follows:

- The '+8V' LED indicates that the board's regulated 8V supply is correctly functioning.
- The 'LOCK' LED indicates that the PLL has locked on to the selected frequency.
- The 'PWR' LED monitors the current through the power transistor (Q5) and indicates that RF power is being produced.

It is now time to select the frequency you wish to set the transmitter to. If this is 103 MHz or below, now is the time to install VC1. **If you wish to use frequencies from 103 to 108 MHz, do not install VC1.**

The positions of the DIP switches required to set the transmitter to a particular frequency can be found at:

- www.zynq.uk/tx/

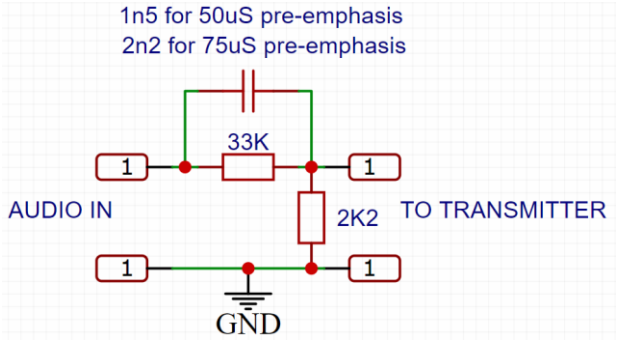
When you have set the switches to the correct positions for the frequency you wish to use, re-apply the 12V power source. It is possible that at this point, the 'LOCK' LED may not now light. To correct this, gradually rotate VC1 using a suitably sized screwdriver with a non-metallic (or insulated) tip, and at some point the 'LOCK' LED should illuminate.

To get the optimum modulation performance, measure the voltage at either end of R9 (this is roughly in the centre of the board, next to the tall 100 uF electrolytic capacitor). Slowly and gradually rotate VC1 until the voltage when the circuit is locked is around 6 Volts (one volt either way will not matter).

The 'PWR' LED should be lit (though note it is often not as bright as the other two). You can carefully compress or expand the turns of inductor L5 to maximise the brightness of the 'PWR' LED. This will maximise the output power of the transmitter.

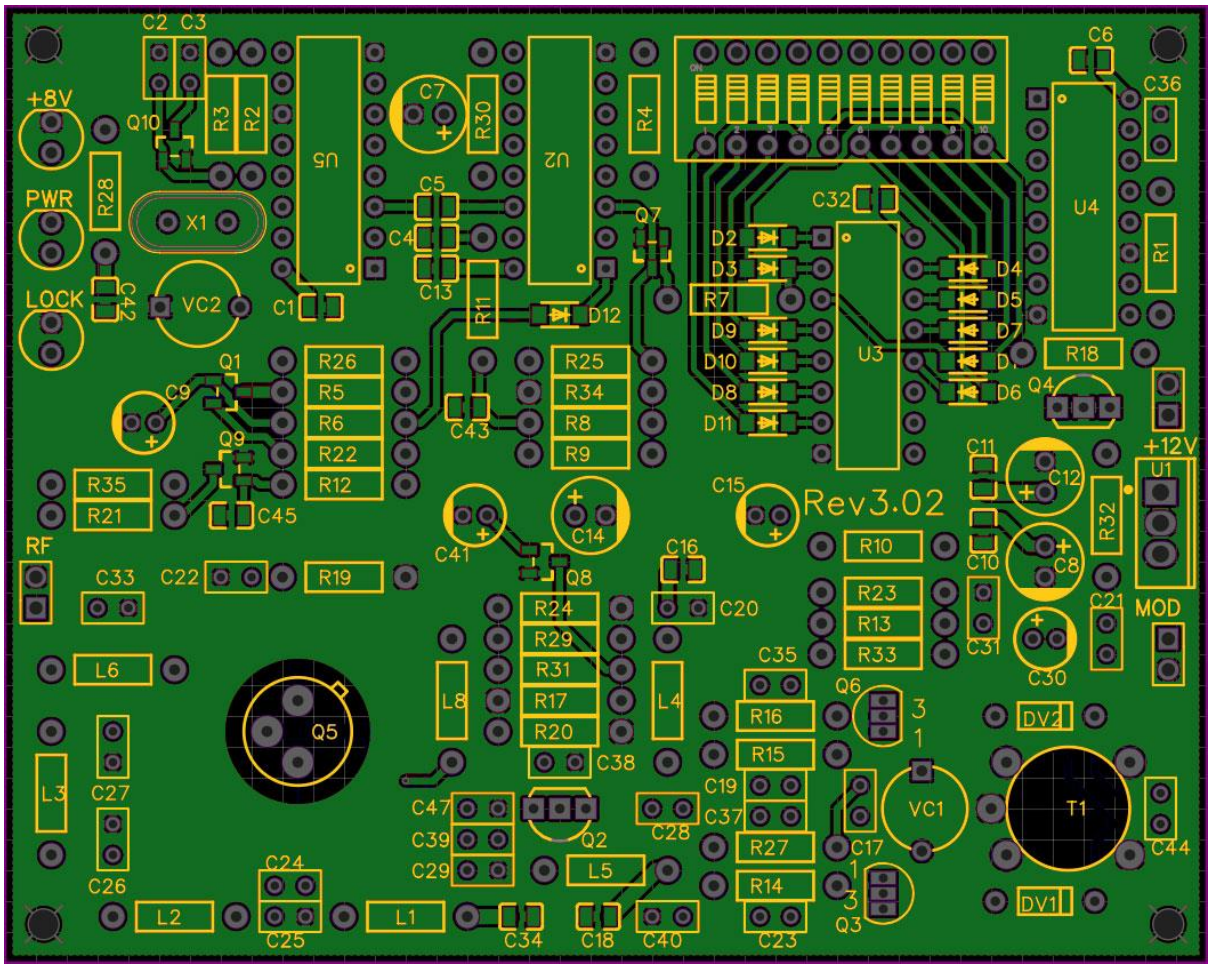
No other adjustments are necessary. The output power of the transmitter should be over 500 mW for a 12 Volt supply. With a 15 Volt supply the output power should reach 1 Watt. Do not exceed 15 Volts on the power supply.

The modulation input to the transmitter is very sensitive, only a few hundred milliVolts of audio are required for full deviation. Note that the input is not pre-emphasised and is intended for a pre-emphasised input (or a stereo MPX input). If pre-emphasis is required, please use the following circuit at the audio input to the transmitter (components not supplied).



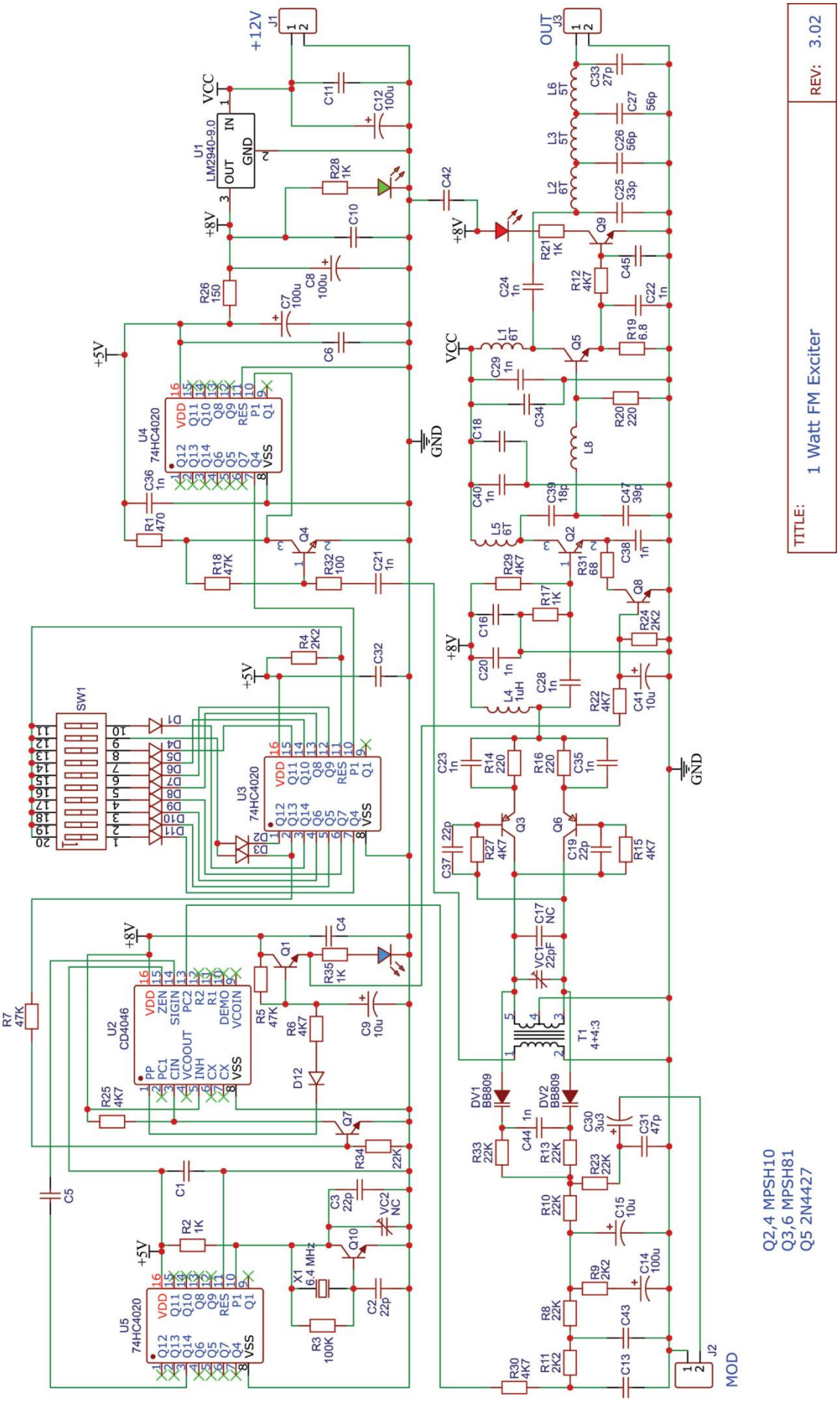
PCB Layout

The layout of the PCB is replicated below for reference (as installing the components can cover up their label on the PCB). Note that in the case +12V and RF, the square hole is 'live' and the round hole is 'ground'. This is the other way around for the modulation, where the square hold is 'live'.



Circuit Diagram

The circuit diagram of the transmitter is provided on the next page. There are a number of components marked 'NC' (not connected). These are not necessary for the correct functioning of the transmitter over its normal frequency range, and are only needed when the unit is used for special purposes. The values of components which are pre-installed on the PCB (i.e. the SMD devices) are not shown.


TITLE:
1 Watt FM Exciter
REV: 3.02
Q2,4 MPSH10
Q3,6 MPSH81
Q5 2N4427

Troubleshooting

If the +8V LED is lit, and the +8V and +5V power supply rails are at the correct voltages, check the voltages at the following points which should be in the following range.

Measurement Point	Expected Voltage
U2, Pin 3	2.0 - 7.0 V
U3, Pin 13	1.5 - 4.5 V
U4, Pin 10	2.5 - 3.5 V
U4, Pin 7	2.5 - 3.5 V
U5, Pin 10	2.5 - 3.5 V
U5, Pin 3	2.5 - 3.5 V

If any of these voltages are incorrect, please carefully check that all resistor values are correct and that there are no short-circuits on the board.

Parts List

Note that parts that are already installed (SMD), or not required, are not shown in the list below.

Capacitors	Type	Quantity
C2, C3, C19, C37	22pF	4
C7,C8,C12,C14	100uF	4
C9,C15,C41	10uF	3
C20,C21,C22,C23,C24,C28,C29,C35,C36,C38,C40,C44	1nF	12
C25	33pF	1
C26,C27	56pF	2
C31	47pF	1
C30	3.3uF	1
C33	27pF	1
C39	18pF	1
C47	39pF	1
VC1 (only fitted for frequencies below 103 MHz)	22pF	1

Semiconductors	Type	Quantity
DV1,DV2	BB609A	2
Q2,Q4	MPSH10	2
Q3,Q6	MPSH81	2
Q5	2N4427	1
LED1,LED2,LED3	5mm, any colour	3
U1	LM2940-9.0	1
U2	CD4046	1
U3,U4,U5	74HC4020	3

Inductors	Type	Quantity
L1,L5	6T, 4mm ID (0.56mm wire)	2
L2,L3,L6	5T, 4mm ID (0.56mm wire)	2
L4	1uH	1
T1	4+4:3 on Toroid (0.35mm wire)	1

Resistors	Type	Quantity
R1	470	1
R2,R17,R21,R28,R35	1K	5
R3	100K	1
R4,R9,R11,R24	2K2	4
R5,R7,R18	47K	3
R6,R12,R15,R22,R25,R27,R29,R30	4K7	8
R8,R10,R13,R23,R33,R34	22K	6
R14,R16,R20	220	3
R19	6.8	1
R26	150	1
R31	68	1
R32	100	1

Miscellaneous	Type	Quantity
X1	6.4 MHz Crystal	1
J1,J2,J3	2 pin header	3
SW1	10 Way DIP Switch	1