A Surface Mount Signal Tracer

Bill Mooney describes how ultra miniature components can be used to construct a super small circuit test device.

When designing, servicing or just dabbling with electronic circuits, a tracer is a handy gadget to have around. Its built in detector for amplitude modulated (Radio Frequency) RF signals will allow you to trace signals back through the various stages of the circuit and its use of surface mount components introduces this interesting technology.

The circuit is based on the MC34119, a clever little device originally designed for speakers, power amplification and other telephone applications. It draws only 2.7mA quiescent current and will work with a supply as low as 2V. In the signals tracer it will deliver 100mW into an 8Ω speaker using a 3V supply provided by a couple of 1.5V N cells. The chip also has a useful chip-disable (muting) input where a high on pin 1 reduces the standby current to 65μA. Although this is not used in the signal tracer, it may be of interest to those who compulsively scribble circuit diagrams on the back of beer maps and envelopes. The circuit diagram of the tracer is shown in Fig. 1.

The gain of the circuit is determined by a pair of resistors R2 and R3 connected to the inverting input in classical manner. The capacitor C3 provides bass cut and the maximum gain is 48dB at 1kHz but falls off above this due to internal compensation. A lower gain would give a wider frequency response but in this case the small speaker compensates somewhat for the fall off and the tracer needs to be as sensitive as possible. The resistor R3 is a little outside the design range of for this IC and a small offset current must be injected into the negative input to balance the outputs. The 10MΩ resistor, R1, provides this current. Since there is no output capacitor even a small DC difference across the loudspeaker (pins 5 and 8) will result in a heavy increase in quiescent current. For example just 100mV difference between pins 5 and 8 will result in some 12mA extra drain.

The completed signal tracer with insert showing PCB.

To achieve the high input impedance required in such an instrument the system is operated in the less stable non-inverting mode. A value of 100kΩ is adequate and will have minimal effect on any circuit under investigation. The sensitivity control consists of] a 100k potentiometer and audio signals are applied directly to this. For radio frequency work a Schottky diode detector, D1, is switched into the circuit. The resistors R4 and R5 provide a current loop for the diode. A 100pF input capacitor, C5, allows only RF to reach the detector. C6 removes residual RF allowing audio to be picked off through C7. The BAR18 Schottky diode is packaged in an SOT23 surface mount package. It has a very low turn on voltage, staring to conduct well below 100mV but specified as 0.4V at 1mA. A very low junction capacitance allows it to work up to UHF and it will withstand 70V piv. This is a respectable replacement for the germanium device usually employed in this application.If working in high voltage environments such as valve equipment, the working voltage of C5 should be increased to account for the increased strain.

The circuit is fabricated on a thin (1mm double sided PCB measuring 3.8cm by 1.25cm. This is small enough to make a very compact probe using button cell batteries.. However, a more practical approach is to use the N cells which should give a couple of months intermittent operation. The PCB foil pattern in shown in Fig. 2 and the

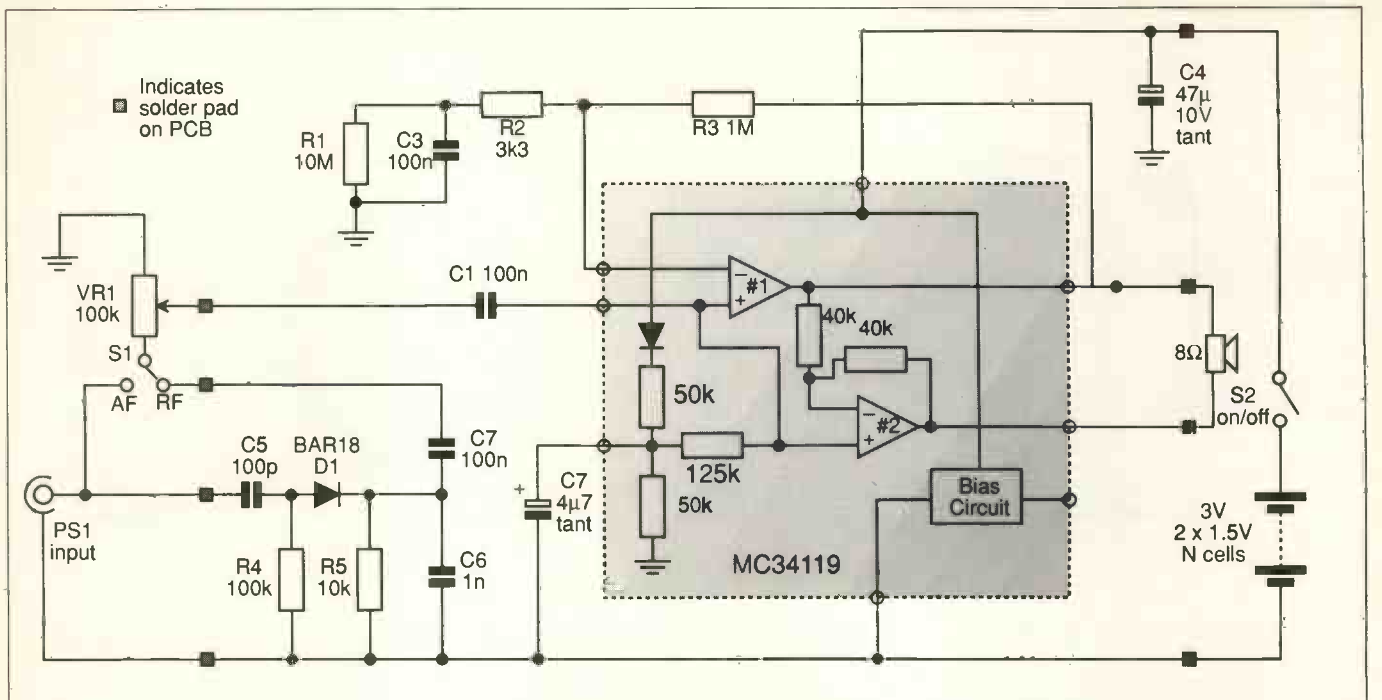


Fig. 1. The main circuit diagram.

component overlay in Fig. 3. The PCB is produced in the conventional manner using UV sensitive spray coating, developing and ferric chloride etching.

When putting the surface mount devices onto the board it is better to start with the larger devices such as the tantalum capacitors. Use the minimum amount of solder and apply the iron for sufficient time to allow it to flow nicely and wet both surfaces. The IC pins can easily be bridged with solder so take care – a solder wick or sucker can be used to remove any excess.

The tantalum capacitors are marked with a band at the positive end. The ceramic chip caps are not marked and should be kept under wraps until required. Pin 1 on the IC is indicated by a flat along one edge. Sometimes a dot is used to indicated pin 1 of SOIC ICs.

The position of the PCB and the various pieces of hardware can be seen in Fig., 4. Mark the battery holes first by making sure that the PCB will slot in as shown underneath S1. Leave a small gap running between the battery holder and the box for the wires running from the solder tags. The pins on the switches, phono socket and sensitivity control all need to be cur back with a side cutters. A length of about 1.5mm should be sufficient. See that the phono socket terminal is clear of the battery connector and likewise the three connections to VR1. Mark the long pin of VR1 before trimming, this is the wiper. The potentiometer specified in a P16 "knob pot" having the resistive

Fig. 2. The PCB (full size).

element in the knob and thus leaving plenty of room inside the project box. Thin 30awg Kynar wire is ideal for all interconnections with a twisted pair for the loudspeaker connections. The battery holders should be wired in series before finally bolting them into place. The M2 countersunk bolts for the battery holder should be cut to length and filed level with the nuts otherwise the cells will not fit into

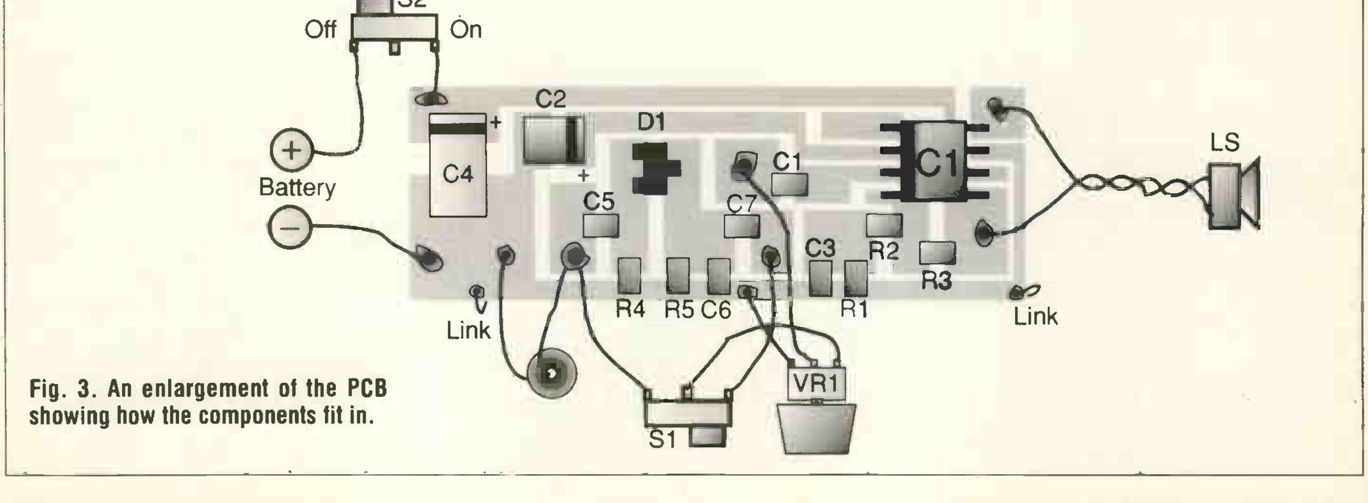


Fig. 3. An enlargement of the PCB showing how the components fit in.

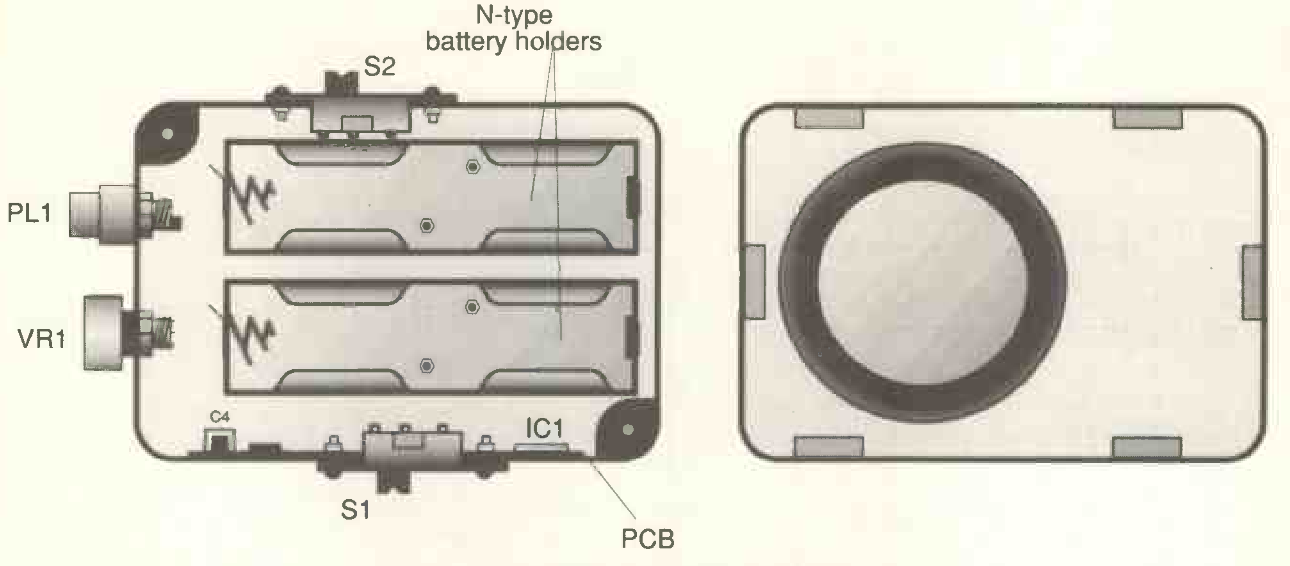
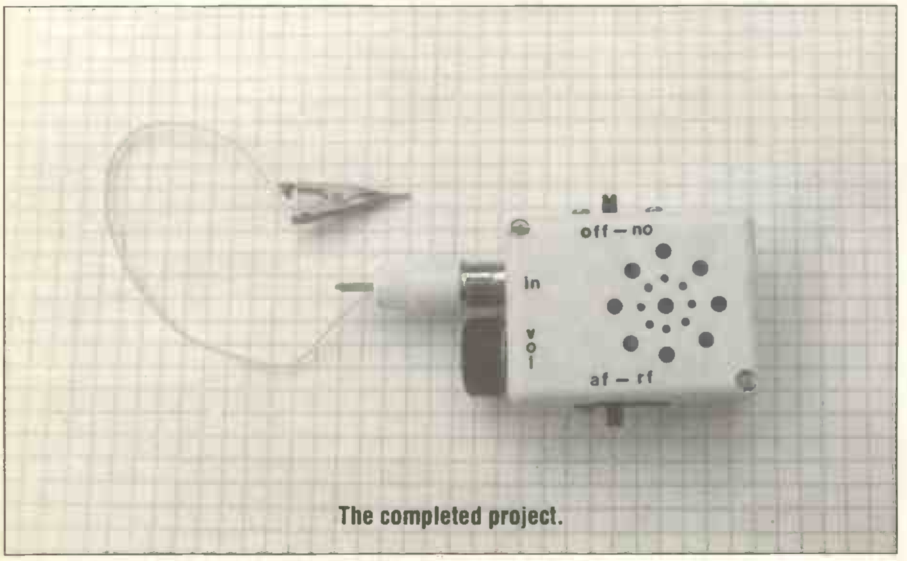


Fig. 4. How everything fits into the case.

the holder correctly. The M2 round head bolts for the switches also need to be trimmed to length. The 'U; shaped channels for the two switches to drop into can be made with a sharp scalpel and filed square as the [plastic is easily worked. The switch mounting plates remain on the outside of the box leaving more room inside and allowing the batteries to be replaced easily.

Solder some lengths of connecting wire to the PCB and slide it into place. Now trim the leads to the shortest length which will allow the board to be slid out slightly to make measurements. Take care with the Kynar wire as it will not take too much flexing and is liable to break at solder joints. The links between the rear of the board and the ground track provides screening. Input wires should be kept away from output wires for maximum stability. A dab of blue-tack will hold the PCB in place.

The next step is to check everything very carefully. Are the batteries wired correctly? and the tantalum capacitors the right way around?> Are there any solder bridges? If all looks well apply power but monitor the drain current with the sensitivity control set to minimum. The unit should draw 2.7mA ideally but up to 5mA is acceptable. The difference is due to a slight imbalance of the output amplifiers which could be trimmed by setting R1 to actually achieve 2.7mA. Check the audio function first by poking around inside a radio receiver or applying a 1kHz signal to the input.



The completed project.

A simple input probe can be made up from an inch or so of 18swg wire soldered to the centre of a phono plug and fired to a point. A few inches of flexible wire soldered to the phono plug screen with a small crocodile clip at the end provides a ground connection. ■

Components

Resistors

0805 chip resistor 2%

R1	10M
R2	3k3
R3	1M
R4	100k
R5	10k
VR1	100k P16 type knob pot

Capacitors

C1, C3, C7	100nF 0805 chip capacitor X7R
C2	4.7µF 16V tantalum
C4	47µF 10V tantalum
C5	100pF 0805 chip capacitor COG
C6	1nF 0805 chip capacitor COG

Semiconductors

D1	BAR18 Schottky diode SOT23 package
IC1	MC34119 SO8 package

Hardware

S1, S2	miniature slide switch
Sk1	phono socket

2 x flat based N type battery holder
Project box type BIA 50x37x24mm small mountable white plastic box
8Ω miniature dynamic loudspeaker CS29B type