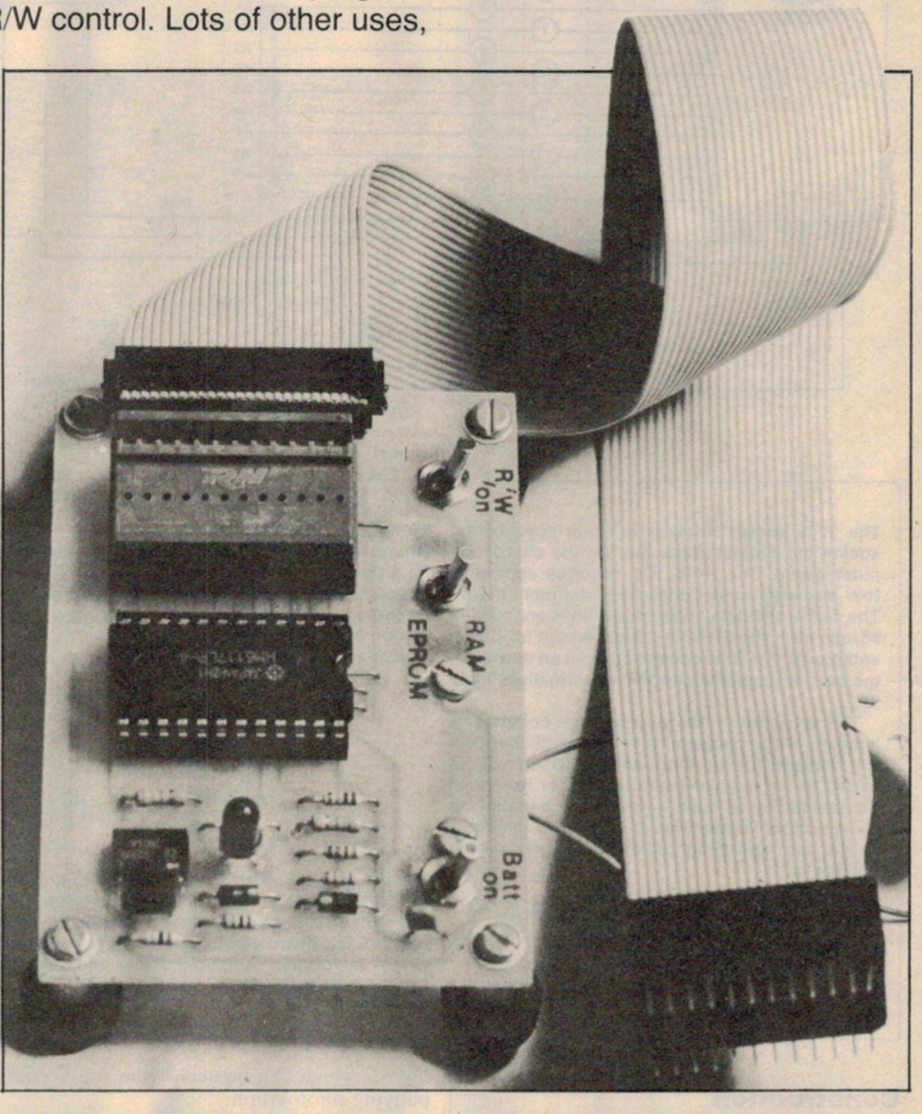
The program bug debugger

Ever burnt a program into an EPROM and found it wouldn't run? Without a logic analyser it's nigh on impossible to debug. This project costs far less than a logic analyser and gets you out of the jam. The Bug Debugger provides 'mirror image' RAM (battery-backed if you like), permits the addition of program breakpoints and interception of R/W control. Lots of other uses, besides.

Graeme Teesdale



MANY MICROCOMPUTER programmers would have, at some time or another, experienced the situation where, having written a program in CP/M or assembler, burnt an EPROM from the hex listing, placed it in the board and executed a test run — to find nothing has happened!

Shriek, expletive deleted, tearing of hair, pounding of bench, attacking of keyboard, recitation of magic incantations, utterances such as "bother!" and like manifestations of frustration rend the air on such occasions.

Without a logic analyser, it is difficult to debug the unit. However, if you could load the contents of the EPROM into a mirror image RAM where breakpoints can be inserted and R/W control removed, debugging is possible.

This project is designed to produce the hardware for the system which will plug into a standard 2716/2516 EPROM socket. Depending on the microprocessor used, you may need to write a short memory-block move routine. Many micros have such a command in their primitive VDU monitor software.

The project hardware can be used in three ways:

(1) As a temporary extension socket for existing EPROM.

(2) For additional RAM, using the extra *chip select:* In this mode the existing EPROM is placed into the debugger board so that the flying lead CS2 can be inserted into the original socket.

(3) As a mirror RAM debugger. In this mode the chip select switch is set to RAM position and the R/W switch off. The result is a volatile ROM (non-volatile if battery-backup is used).

With battery backup, the program can be stored in the RAM for many months as the current drain is very, very low (less than 5 µA; the prototype was measured at 3.2 µA).

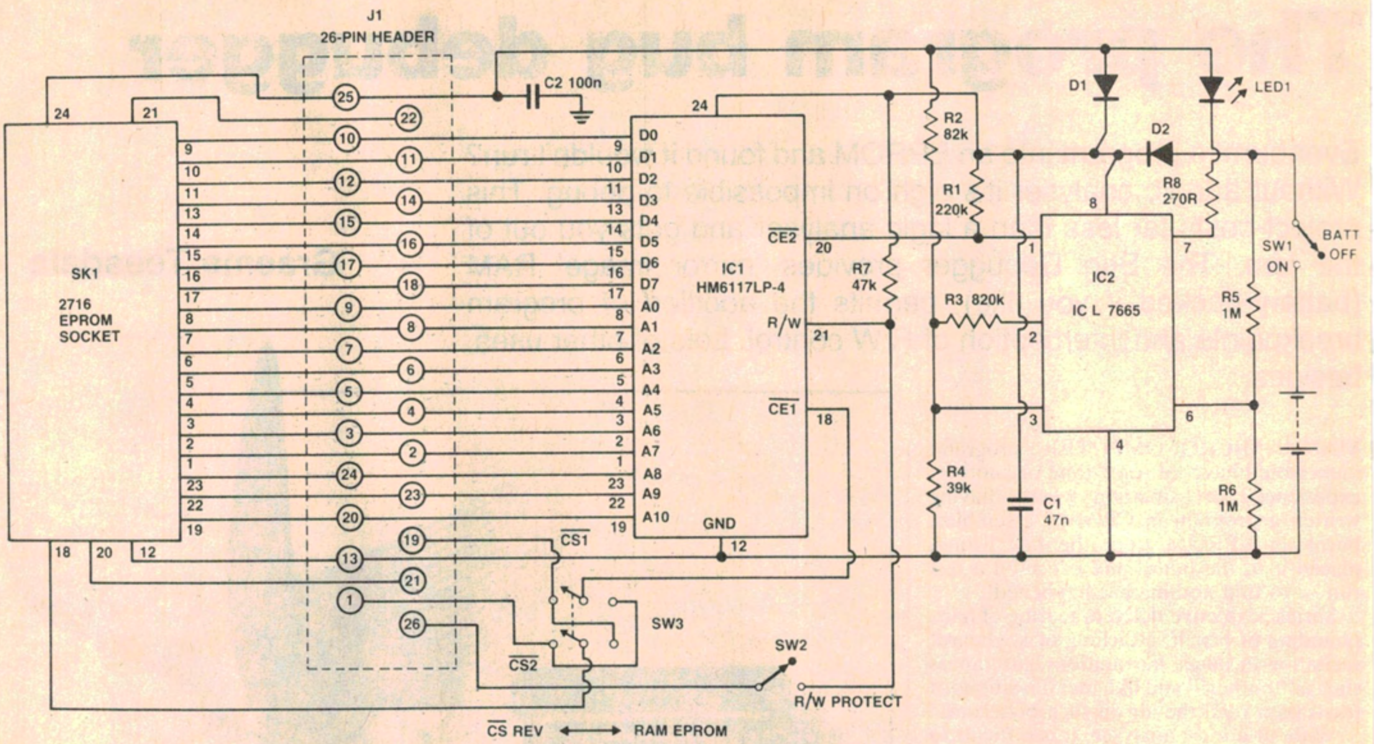
Design details

The circuit makes use of a Hitachi CMOS 2K x 8-bit RAM chip, type HM6117LP, and an Intersil micropower under/over-voltage detector, type ICL7665. The HM6117LP is a version of the HM6116LP, the previously used *output enable* line (OE), pin 20 is a second *chip select*, CS2, in the 6117, and acts as a *de-select* pin on power-down.

With the 6116, on power-down it is necessary to have pullup resistors on the address and data lines, determining input status, to obtain minimum standby current. Also, additional gating of WE and CE1 is required. The very low price differential between the 6117 and the 6116 made the former the obvious choice for this project.

The ICL7665 contains two individually programmable voltage detectors on a single chip, referenced to an internal 1.3 V source. The outputs from the two voltage comparators drive four open-drain FETs. Out 1 (pin 1) and Out 2 (pin 7) are N-channel, Hyst 1 (pin 2) and Hyst 2 (pin 5) are P-channel.

Each section is independent of the other but both use the same 1.3 V reference. The input impedances of the two inputs, Set 1 and Set 2, are each extremely high, with typically 10 femptoamps (0.01 nanoamps!) leakage current. Total current drawn by the 7665 is very low, typically 2.5 µA. The output MOS transistors are capable of sinking and sourcing LED currents, a fact I made use of here to indicate battery voltage. ▶



HOW IT WORKS — ETI-656

The 2716 socket SK(1) is a direct copy of the socket the project plugs into on the microcomputer board. The RAM, IC1, requires additional R/W and chip select lines from the main board. The rest of the data and address lines are bussed across from the EPROM socket except for pin 20 and pin 18. Pin 20 is controlled from an output on the over-voltage detector and is normally at logic '0'.

The set input on the over-voltage comparator is taken to the centre of the voltage divider (R2-R4) between the +5 V rail and 0 V. Hysteresis is added to the over-voltage comparator to reduce the possibility of false switching or oscillations as a result of fluctuations in the supply rail down to the top switching point. The upper switch voltage is:

$$V_H = \frac{R4 + R2}{R4} \times 1.3$$
$$= \frac{82k + 39k}{39k} \times 1.3$$
$$= 4.03\ V$$

The lower switching point is below the voltage where the Out 1 MOS transistor is off. Resistor R1, in this case, is used as a pullup resistor, pulling pin 20 of IC1 up to the supply rail. For data retention, pin 20 must be within 0.2 V of the voltage on the supply rail — pin 24 of IC1. The lower switching point is:

$$V_L = \frac{R4 + \frac{R2R3}{R2 + R3}}{R4} \times 1.3$$
$$= \frac{39k + \frac{82k \times 820k}{82k + 820k}}{39k} \times 1.3$$
$$= 3.78\ V$$

The undervoltage comparator is used to monitor the battery voltage. When it's below the setpoint, the Out 2 MOS transistor is on, switching the LED on. The LED *only* indicates battery condition when the project is plugged into the EPROM socket on a micro board and is deriving supply from there.

The under-voltage trip voltage is given by:

$$V_U = \frac{R6 + R5}{R6} \times 1.3$$
$$= \frac{10^6 + 10^6}{10^6} \times 1.3$$
$$= 2.6\ V$$

Diodes D1 and D2 are used as blocking diodes, isolating the +5 V and battery supplies from one another. The only precaution recommended by the manufacturers of the ICL7665 (Intersil) is decoupling very close to the supply pins.

Under certain circumstances, like very rapidly increasing power supply input, the four-layer PNPN type structure of the chip can latch-up, drawing excessive supply current. The problem is more likely to happen when the Debugger's battery is switched on before the Debugger is plugged into the microcomputer's power supply. Generally, growth and decay of the micro's +5 V supply would have a much longer time constant, presenting no problem.

Construction

Before commencing assembly, check the printed circuit board for shorts between tracks — particularly the data and address lines, no matter whether you've purchased a ready-made board or etched your own. A visual check alone may not show hairline shorts, so use an ohmmeter or continuity checker. Also check that all the holes have been drilled to the correct diameter.

I used a Scotchcal label to indicate the switch functions on the board. If you're making your own, do it now. Attach the label to the appropriate position on the board, which can be seen from the accompanying photograph.

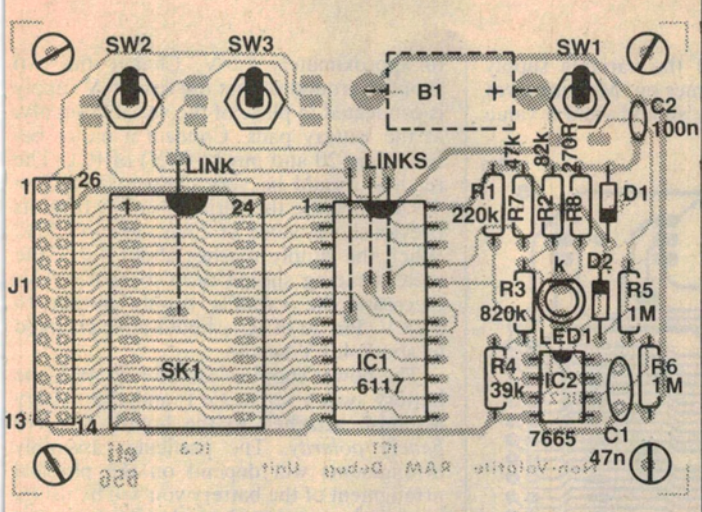
The first thing to do is solder the four links in place. Use 22g tinned copper wire.

Start component assembly by mounting J1, the 26-pin header plug strip, followed by the IC socket and the EPROM socket (SK1). I used a low insertion pressure socket here to avoid problems with pins that might get bent under when inserting an EPROM. After soldering these three components in place it is important to recheck for shorts between data and address lines as board tracks run between pins. If, or when, all is well, proceed by inserting and soldering resistors R1 to R8 in place, followed by capacitors C1 and C2, switches SW1, SW2 and SW3. The switches are wired up with short lengths of tinned copper wire, direct to the pads on the board as shown in the underside wiring diagram.

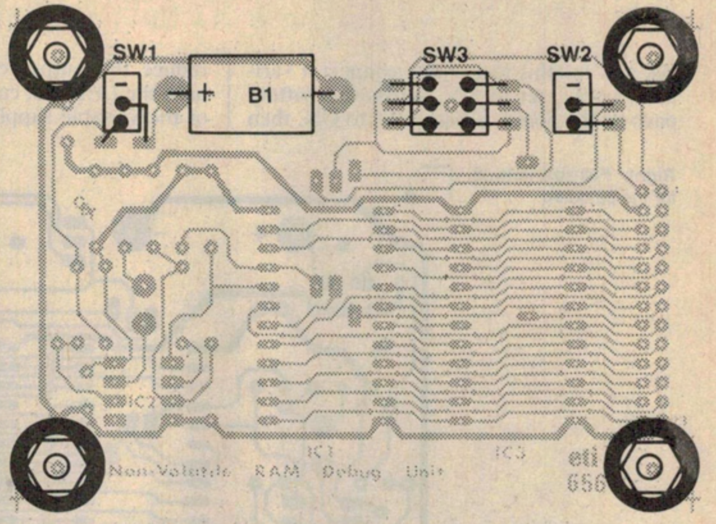
Next, insert the diodes, D1-D3, ensuring you get them all the right way round. Last of all, insert and solder IC2 in place, making sure that it, too, is correctly orientated.

If you're using battery backup, it is best to check the action of IC2 before proceeding to assemble it.

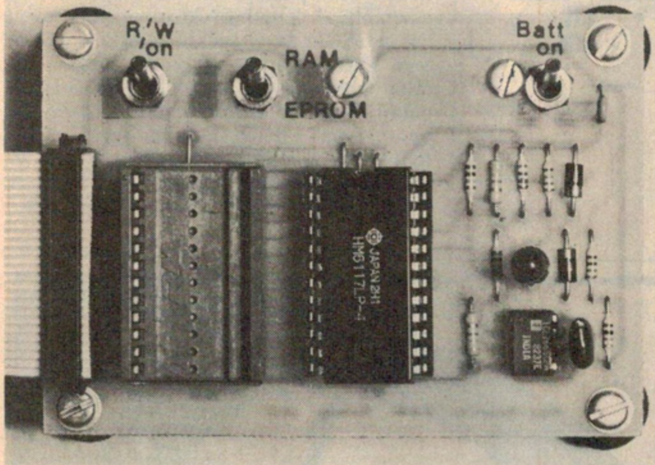
Connect a 5 V source between pins 24 and 12 of SK1 (EPROM socket). IC1 is not



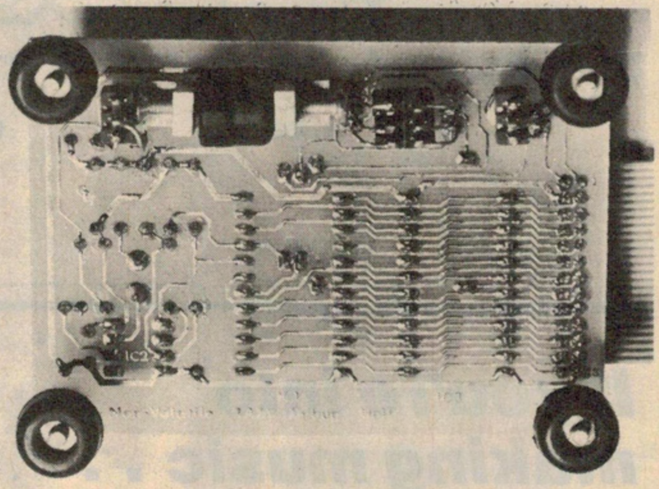
Topside overlay. Showing component placement. Put the links in first of all.



Bottomside overlay. Showing switch wiring and battery placement.



Topside view. Compare with the overlay above.



Bottom side view. Compare with the overlay above.

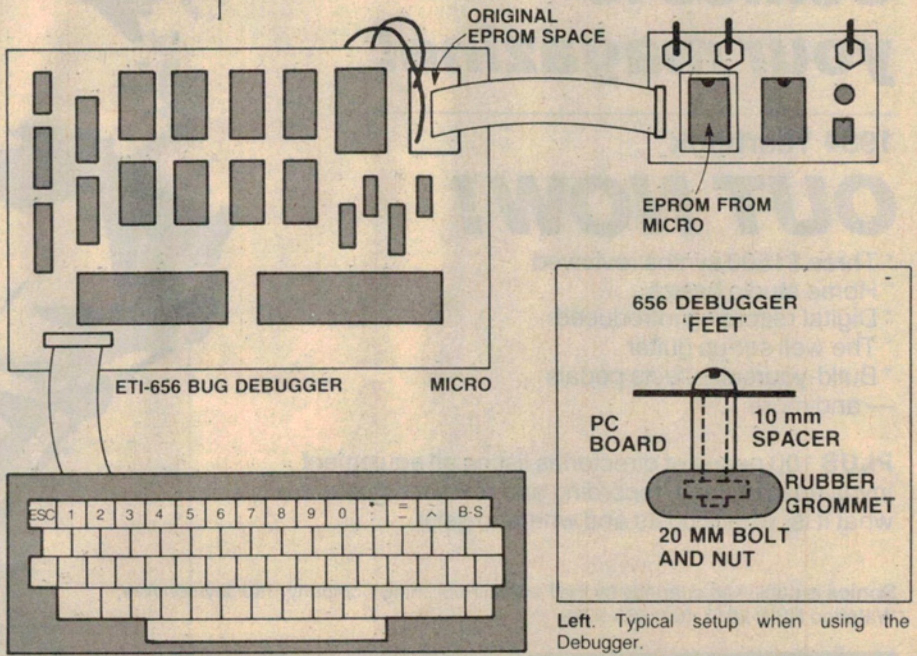
ETI-656 PARTS LIST

Resistors.................all ¼W, 5%
R1..........................220k
R2..........................82k
R3..........................820k
R4..........................39k
R5,R6....................1M
R7..........................47k
R8..........................270R

Capacitors
C1..........................47n greencap
C2..........................100n bluechip ceramic

Semiconductors
IC1........................HM6117LP-4
IC2........................ICL7665
D1,D2....................1N4001
D3..........................TIL220R or similar

Miscellaneous
ETI-656 pc board; Scotchcal label; 1 x 24-pin low insert pressure socket (SK1) — Robinson-Nugent type TSN246; 1 x 24-pin IC skt; 26-pin header; 24-pin DIP adaptor plug & cover (R-N type MPB-246, MPC-246-1); 2 x SPST min. toggle switches; 1 x DPDT min. toggle switch; min. 3 V NiCad; 26-way ribbon cable; nuts, bolts, standoffs, grommets.



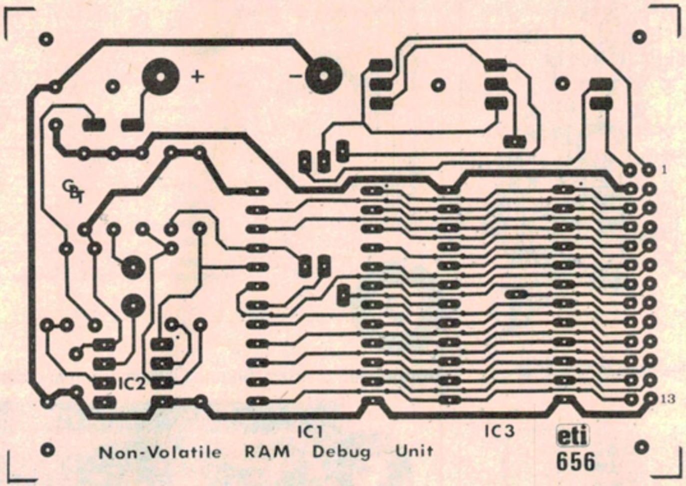
Left. Typical setup when using the Debugger.

inserted for this test. Now, connect a variable supply, set to 5 V, across the battery pads on the board. Switch SW1 to ON, then reduce the output of the variable supply until the LED just comes on. Measurement of the variable supply should yield a value of approximately 2.6 V. Change the two supplies around so that the fixed 5 V supply is connected in place of the variable supply, at the battery pads. Connect a meter between pin 20 and pin 12 (0 V) of IC1. The reading should be very close to 5 - 0.7 = 4.3 V. Increase the variable supply from its previously set value of 2.6 V, towards 5 V. When the supply reaches about 4.0 V the meter reading should drop to nearly 0 V. Decrease the variable supply output to 3.78 V. Pin 20 of IC1 should have changed to about 4.3 V again.

The power supplies can be removed, the battery switch set to OFF and the battery assembly attached to the board, *Observe battery polarity*. The particular assembly arrangments will depend on the physical arrangment of the battery you will be using, so exact construction details are not provided. IC1 can be inserted once the battery is in place. Now make up the cable assembly. A length of 26-way ribbon cable has a 26-pin IDC socket attached at one end. This plugs into J1 on the board. The other end goes to a solder DIP adaptor plug, as can be seen in the accompanying photograph. Flying leads for the CS2 and R/W lines are attached to pins 1 and 26, with small identifying labels stuck to them.

Go debug those bothersome bugs! ●

Right. Full-size artwork for the pc board.



Below. Full-size artwork for the Scotchcal label.