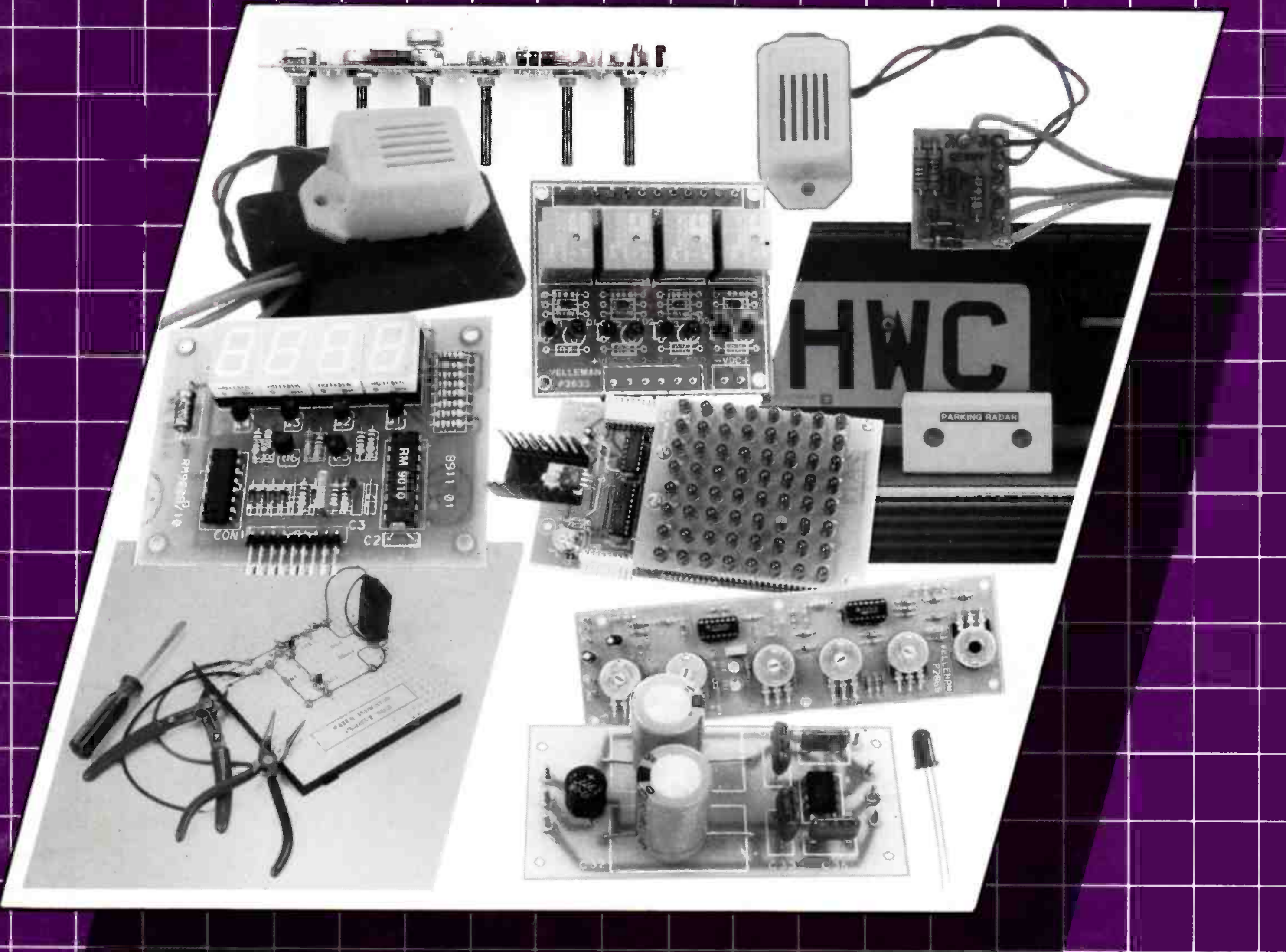
PROJECTS BOOK 50/62

Maplin Maplin Maplin



CAR LIGHTS-ON REMINDER
±15V TRACKING REGULATOR
PROGRAMMABLE TIMER
WATER INDICATOR
MOVING MESSAGE DISPLAY
CAR PARKING RADAR
MONITOR & EFFECTS MODULE
ADD-ON RELAY CARD

CONTENTS

MAPLIN PROJECTS BOOK FIFTY AND SIXTY TWO

EDITORIAL

This Project Book replaces Issues 50 and 62 of *Electronics*, which are now out of print, and contains a compilation of projects from both magazines. Other issues of *Electronics* will also be replaced by Project Books once they are out of print. For current prices of kits please consult the latest Maplin Catalogue or the free price change leaflet, order as CA99H.

R.T. Smith

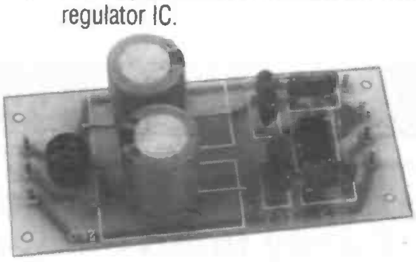
Editor Robert Ball.
Compiled by Mike Holmes.
Contributing Authors Joe Fuller, Tony Bricknell, Robert Penfold, John Koushappas, Martin Pipe, John Mosely, Mike Holmes, Dennis Butcher.
Technical Authors Robin Hall, Dean Hodgkins B.Eng(Hons), Mike Holmes.
Technical Illustrators Ross Nisbet, Lesley Foster, Paul Evans, Nicola Hull.
Production Scheduler Steve Drake.
Print Co-ordinator John Craddock.
Publications Manager Roy Smith.
Development Manager Tony Bricknell.
Drawing Office Manager John Dudley.
Art Director Peter Blackmore.
Designer Jim Bowler.
Published by Maplin Electronics plc.
Mail Order P.O. Box 3, Rayleigh, Essex SS6 8LR.
Telephone Retail Sales: (0702) 554161.
Retail Enquiries: (0702) 552911.
Trade Sales: (0702) 554171.
Cashtel: (0702) 552941.
General: (0702) 554155.
Fax: (0702) 553935. **Telex:** 995695.

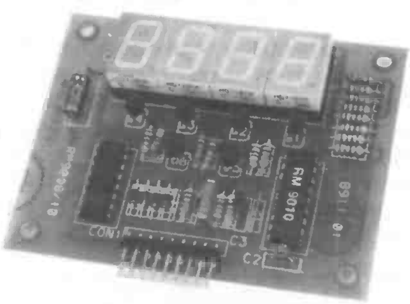
Visit a Maplin Shop for Personal Service

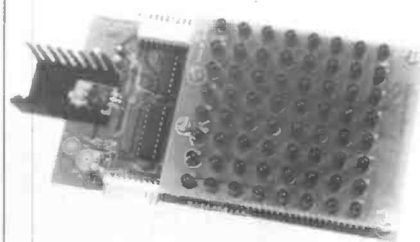
You'll find Maplin Service in:
Birmingham, Sutton New Road, Erdington.
Telephone: (021) 384 8411.
Brighton, 65 London Road. Telephone: (0273) 620930.
Bristol, 302 Gloucester Road, Horfield.
Telephone: (0272) 232014.
Cardiff, 29-31 City Road. Telephone: (0222) 464554.
Chatham, 2 Luton Road. Telephone: (0634) 818588.
Coventry, 12 Bishop Street. Telephone: (0203) 550504.
Dudley, Unit 7, Sterling Park. Telephone: (0384) 485051.
Edinburgh, 126 Dairy Road. Telephone: (031) 313 5551.
Glasgow, 264-266 Great Western Road.
Telephone: (041) 353 3323.
Ilford, 302-304 Green Lane. Telephone: (081) 599 0100.
Leeds, Carpet World Building, 3 Regent Street.
Telephone: (0532) 449200.
Leicester, Office World Building, Burton Street.
Telephone: (0533) 623288.
London, 146-148 Burnt Oak Broadway, Edgware.
Telephone: (081) 951 0969.
107-113 Stanstead Road, Forest Hill.
Telephone: (081) 291 9192.
120-122 King Street, Hammersmith.
Telephone: (081) 748 0926.
Manchester, 169 Cheetham Hill Road, Cheetham Hill.
Telephone: (061) 832 2550.
8 Oxford Road. Telephone: (061) 236 0281.
Middlesbrough, Unit 1, The Forbes Building,
309-321 Linthorpe Road. Telephone: (0642) 242900.
Milton Keynes, Snowdon Drive (next to Office World),
Winterhill. Telephone: (0908) 692720.
Newcastle-upon-Tyne, Unit 4, Allison Court (beside
The Metro Centre), Gateshead. Telephone: (091) 488 9555.
Northampton, 139 St James Road. Telephone: (0604) 756726.
Nottingham, 86-88 Lower Parliament Street.
Telephone: (0602) 410242.
Portsmouth, 98-100 Kingston Road. Telephone: (0705) 654411.
Preston, Unit 1, Corporation Street. Telephone: (0772) 258484.
Reading, 129-131 Oxford Road. Telephone: (0734) 566638.
Sheffield, 413 Langsett Road, Hillsborough.
Telephone: (0742) 855492.
Slough, 216-218 Farnham Road. Telephone: (0753) 551419.
Southampton, 46-48 Bevois Valley Road.
Telephone: (0703) 225831.
Southend-on-Sea, 282-284 London Road, Westcliff-on-Sea.

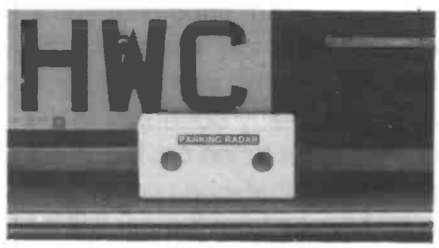
All our shops are open from 9.00am to 5.30pm Monday to Saturday (9.30am Friday). Shops that are open on Sundays are: Brighton if local conditions permit; Edinburgh 10.00am to 4.30pm; Glasgow 9.00am to 5.30pm; Manchester Oxford Road and Cheetham Hill, if local conditions permit; Newcastle 10.00am to 4.30pm. All shops are closed for Public Holidays. Shops do not close for lunch.

PROJECTS









FOR MAIL-ORDER ENQUIRIES, PLEASE *DO NOT* CALL OUR SHOPS AS THEY WILL BE UNABLE TO HELP YOU.
Send all mail to: P.O. Box 3, Rayleigh, Essex SS6 8LR.

CAR LIGHTS-ON WARNING INDICATOR

If your car is not fitted with some kind of 'lights-on warning', the chances are that you will (if you have not already done so!) leave your lights switched on. Murphy's law dictates that when you do so, your absence from the car will be of sufficient duration to ensure that the battery will be well and truly flat. Of course Murphy, not content to do things by halves, will ensure that it happens when you are late for some important occasion and that there is no one else around to give you a push or a jump start!

by Joe Fuller

FEATURES

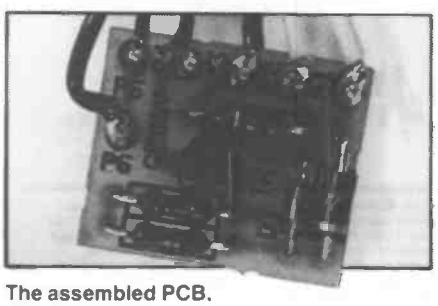
- ★ **Low-cost and small size**
- ★ **Easily fitted to most negative earth cars**
- ★ **Buzzer sounds when lights are left on**

Modern cars further aggrevate the situation as many of them, being fitted with electronic ignition or electronic engine management systems, just plain refuse to be push-started!

It is amazing that such mechanically advanced cars often *do not* have a lights-on warning indicator of some kind. To illustrate this, the prototype was installed in a 2·0 injection Ford Sierra Estate – despite being a 'Ghia', there was no lights-on warning device!

Various warning devices are available, however, some become a nuisance because they sound continuously when the lights are deliberately left on. For instance, whilst the driver is waiting in the car at night, with the engine switched off.

Some more sophisticated devices will not sound if the lights are switched on again *after* the ignition has been switched off, i.e. for parking lights. However, this fails to warn the driver if he inadvertently 'knocks' the light switch on when leaving the car – as is the case with many cars having the light switch 'stalk' on the driver's door side of the steering column.



The assembled PCB.

The Lights-On Warning indicator will emit a clearly audible buzzing sound when the car lights are left on, the ignition switch is turned-off and the driver's door opened. In this manner the buzzer will only sound when the driver is genuinely about to leave the car.

Now that you are thoroughly convinced that for the sake of a few pounds, you need not be caught out in the future, why not build this handy accessory (which the manufacturer should have included as standard) and fit it into your car? Enterprising readers may wish to offer this 'add on' to friends, relatives and neighbours for a suitable fee (don't forget to tell the tax man!). A personal tale of woe and the assurance that, "I've got one and it has stopped me from getting caught out again!" is sure to win a few favourable responses.

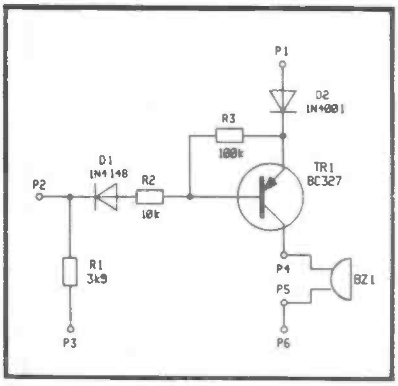


Figure 1. Circuit diagram.

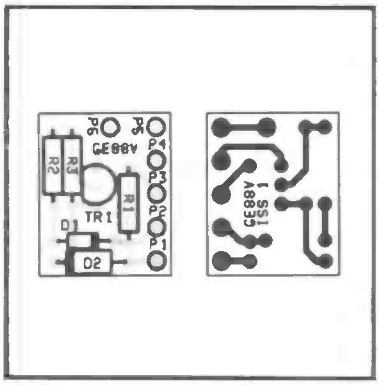


Figure 2. PCB legend and track.

The Lights-On unit attached to a convenient surface.

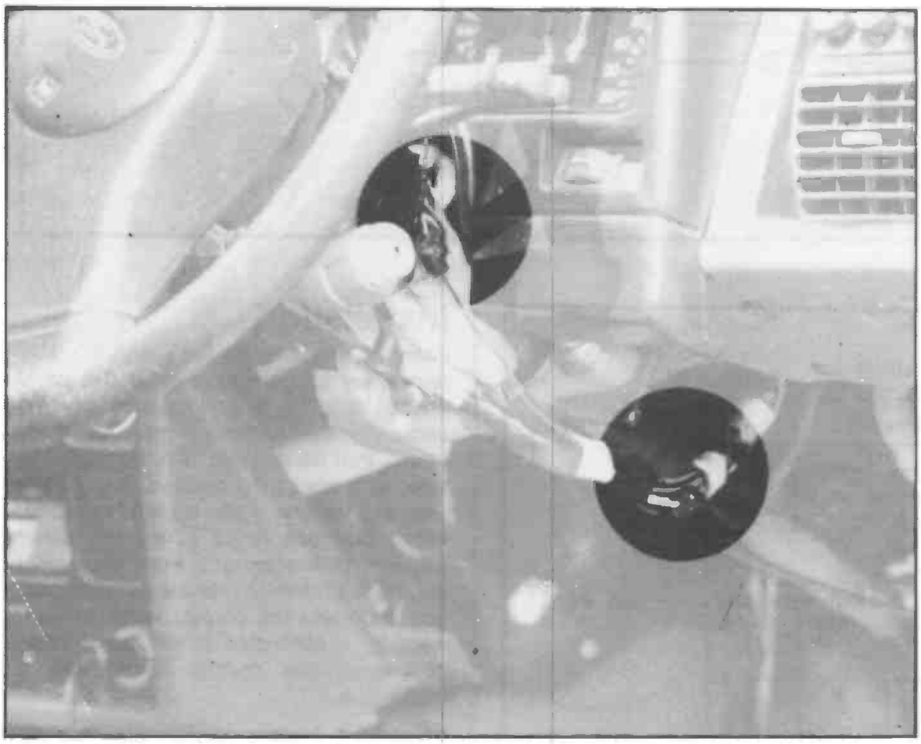


Photo 1. Connections to the sidelight circuit (left) and accessory circuit (right).

Circuit Description and Operation

The circuit of the Lights-On Warning Indicator is very simple, as can be seen from Figure 1. However, it is worthwhile to know how the circuit operates as this will help, should problems occur.

P1 of the unit is connected to the sidelight circuit of the car and provides power to the circuit only when the lights are switched on. The sidelight circuit is live when either sidelights or headlights are switched on.

P2 is connected to the accessory circuit and when the ignition switch is off, P2 is pulled low via R3 (P3 is connected to 0V). D1 is forward biased and turns on TR1 via R2. Note that the internal resistance of accessories (i.e, radio-cassette) may be sufficiently low to make the connection to P3 unnecessary; this can be determined by experimentation.

P6 is connected to the driver's door switch, thus when the door is opened, a complete path to 0V is provided by the door switch, allowing the buzzer to sound.

When the ignition switch is on, P2 is pulled high, reverse biasing D1. R3 ensures that TR1 is held in the off state. The positive supply to BZ1 is removed and thus prevents it from sounding, regardless of whether the driver's door is open or shut.

When the lights are off and the car doors are closed, the polarity of the supply to the unit is effectively reversed. D2 prevents damage to the circuit under this condition.

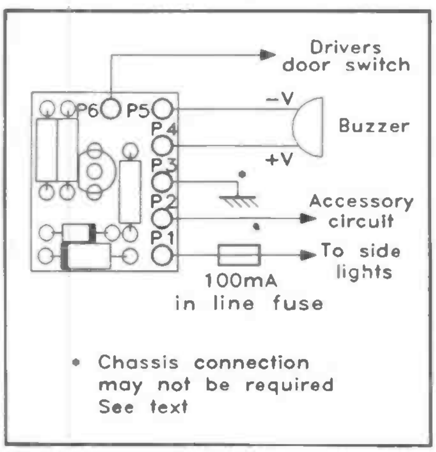


Figure 3 (above). PCB connections.
Photo 2 (left). Connections to the door switch circuit.

Construction

Assembly of the unit is simplicity itself, however, the complete beginner is referred to the Constructors' Guide supplied with the kit, which contains useful information on construction techniques.

Referring to Figure 2, it is advised that the PCB pins are fitted first, followed by the resistors and the diodes and finally the transistor. Make sure that the transistor is fitted fairly close to PCB otherwise the PCB will not fit into the case.

Next solder the buzzer's wires to the PCB pins, red (+V) to P4, black (−V) to P5. Attach the connecting wires to the PCB pins and label the free ends so that you can identify the wires after the PCB has been fitted into the case!

The PCB simply lays in the case, the wires protruding through the aperture provided. Screw the case together and affix the buzzer onto the lid of the case using one of the double-sided adhesive pads. The other pads can be placed onto the underside of the case ready for fitting into the car.

Although it is unlikely that there will be any problems with the unit, it is advisable to test it before fitting into the car. It is easier to take remedial action on the work bench than underneath the car dashboard! Using a 9 to 14V supply (i.e. PP3 battery, battery eliminator, etc.) connect P3 and P6 to 0V, then Connect P1 to +V, the buzzer should sound. Connect P2 to +V as well, this should silence the buzzer.

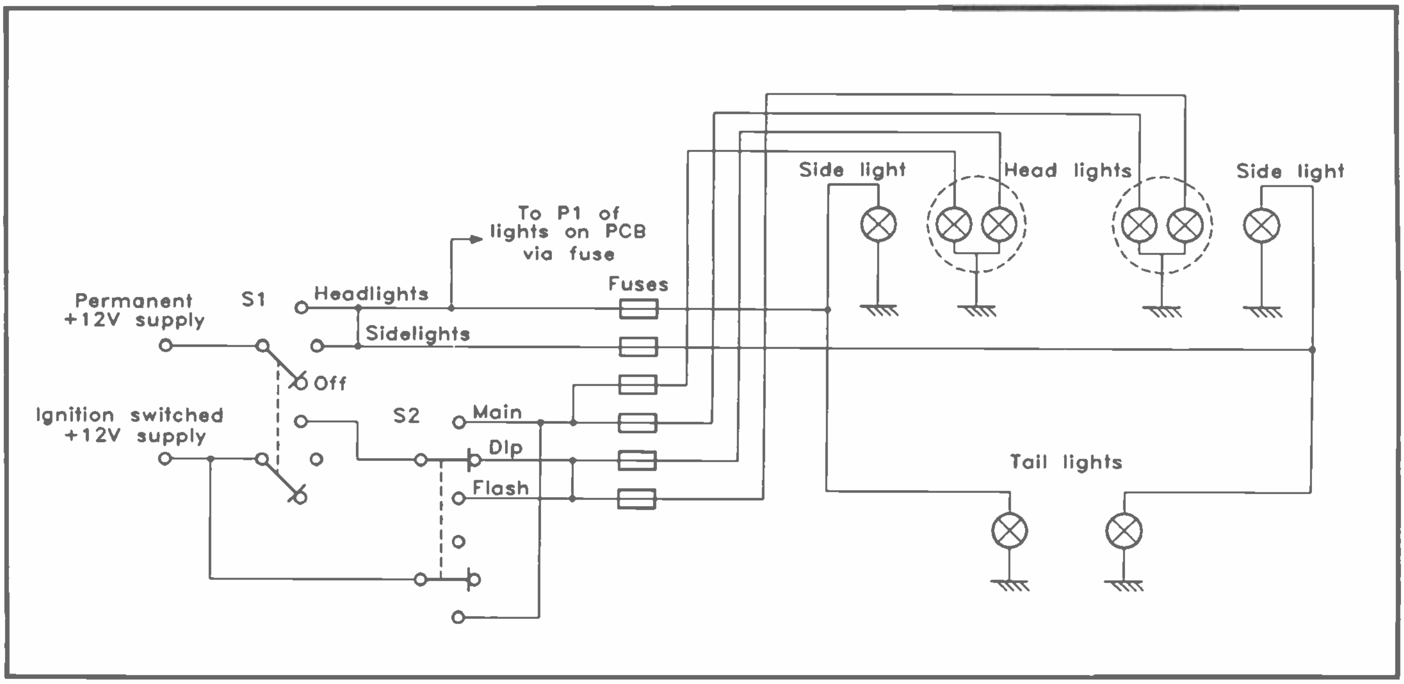


Figure 4. Typical lighting circuit and connections.

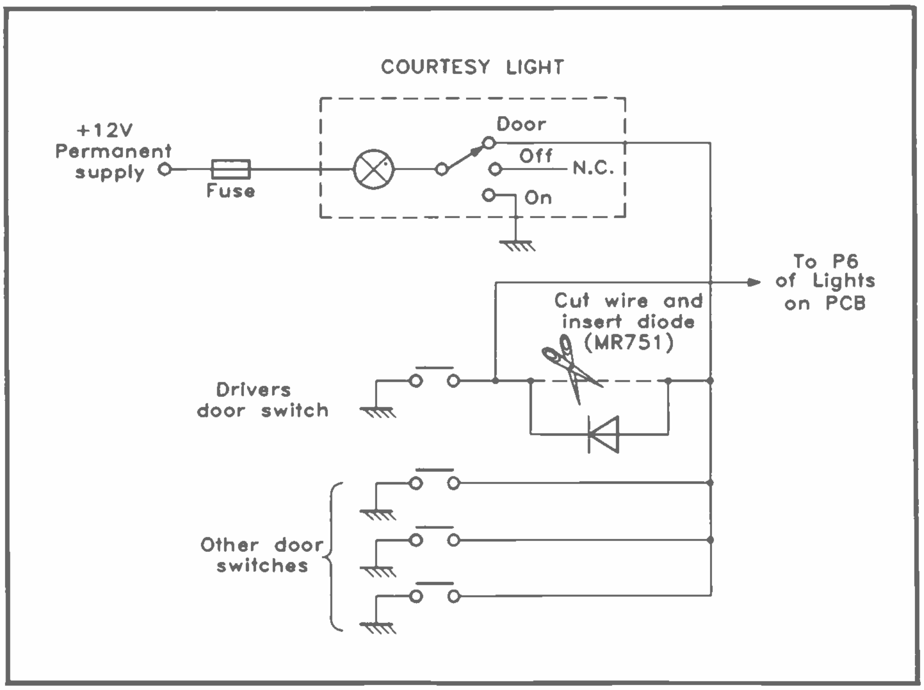


Figure 5. Typical courtesy light circuit and connections.

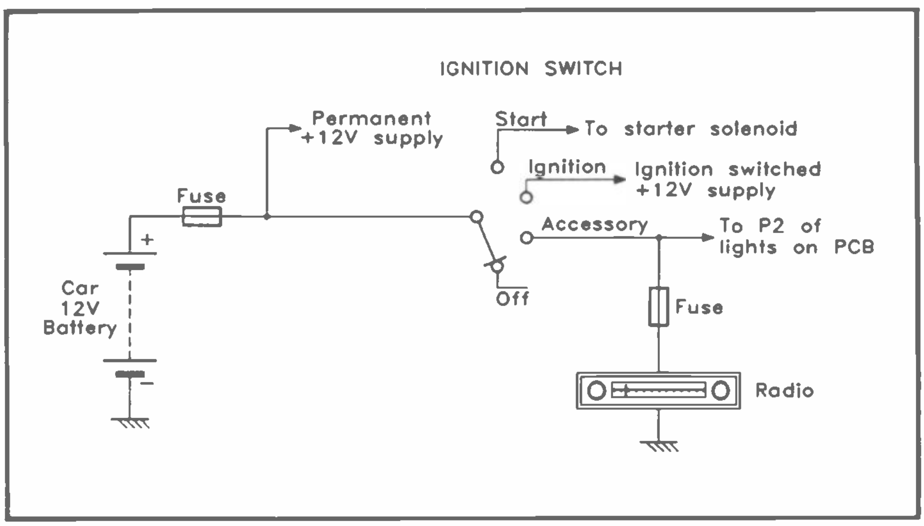


Figure 6. Typical ignition switch circuit and connections.

Installation

Refer to Figures 3, 4, 5 and 6. It is necessary to gain access to the car's wiring, which will undoubtedly involve removing the underside of the dashboard, trim panels, etc. It is advisable to refer to a workshop manual, e.g. of the 'Haynes' variety; if you do not have one, either buy one – as it will be useful anyway, or borrow one from your local library. A workshop manual will also help you ascertain the correct wires to connect to – otherwise it will be a case of tracing the correct wires with a multimeter.

Important Notes: disconnect the car battery before making connections to the wiring. Connections to existing wiring can be made using 'snap lock' connectors or terminal blocks of adequate current rating – remember the Lights-On unit draws very little current, but two 55W headlamp bulbs draw considerably more! Ensure that the new wiring will not become entangled with any controls, especially the brake pedal and steering column. To prevent short circuits, make sure that all connections are properly insulated, use adhesive electrical tape.

Connect P1, via a fuse and fuseholder, to a point in the wiring which becomes live when the sidelights are switched on (Figure 4 and Photo 1).

Connect P6, to the driver's door switch (Figure 5 and Photo 2). To prevent other doors from operating the buzzer, install an MR751 diode in series with the wire to the courtesy light.

Connect P2 to a point in the wiring which becomes live when the ignition switch is turned to 'accessory', i.e. +V supply to the radio (Figure 6 and Photo 1). Alternatively, if there is no 'accessory' position, connect P2 to a point in the wiring which becomes live when the ignition switch is turned to 'ignition'.

Connect P3 to the car's chassis (0V) or to a point in the wiring which is permanently connected to the car's chassis. Note that this connection may be unnecessary if the internal resistance of any accessory is sufficiently low. This may be ascertained by testing the unit with P3 left unconnected and all accessories switched *off*. If in doubt connect P3 as previously described.

Double-check connections, reconnect the car battery.

Testing

Switch lights on, leave ignition switched off and open the driver's door; the buzzer *should* sound.

With the driver's door shut, opening any other door should *not* cause the buzzer to sound.

With the ignition switched to 'accessory' or 'ignition', opening the driver's door should *not* cause the buzzer to sound.

With lights turned off, the buzzer should *not* sound with any combination of ignition switch positions or doors open or closed.

Assuming the unit is working correctly, refit underside of dashboard and trim panels.

Happy Motoring!

LIGHTS-ON WARNING PARTS LIST

RESISTORS: All 0·6W 1% Metal Film			
R1	3k9	1	(M3K9)
R2	10k	1	(M10K)
R3	100k	1	(M100K)
SEMICONDUCTORS			
D1	1N4148	1	(QL80B)
D2	1N4001	1	(QL73Q)
TR1	BC327	1	(QB66W)
	MR751	1	(YH96E)
MISCELLANEOUS			
BZ1	Buzzer 12V	1	(FL40T)
P1–6	Pins 2145	6 pins	(FL24B)★
	In-line Fuse Holder 1¼in.	1	(RX51F)
	100mA 1¼in. Fuse	1	(WR08J)
	PCB	1	(GE88V)
	Mini Box and Base	1	(JX56L)
	Quickstick Pads	3 pads	(HB22Y)★
	Instruction Leaflet	1	(XT11M)
	Constructors' Guide	1	(XH79L)

OPTIONAL (Not in kit)		
16/0·2 Wire	As Req.	(FA26D–FA36P)
Snap Lock Cable Connector	As Req.	(JR88V)
Terminal Block 5A	As Req.	(HF01B)

The Maplin 'Get-You-Working' Service is available for this project, see Constructors' Guide or current Maplin Catalogue for details.

The above items (excluding Optional) are available as a kit.

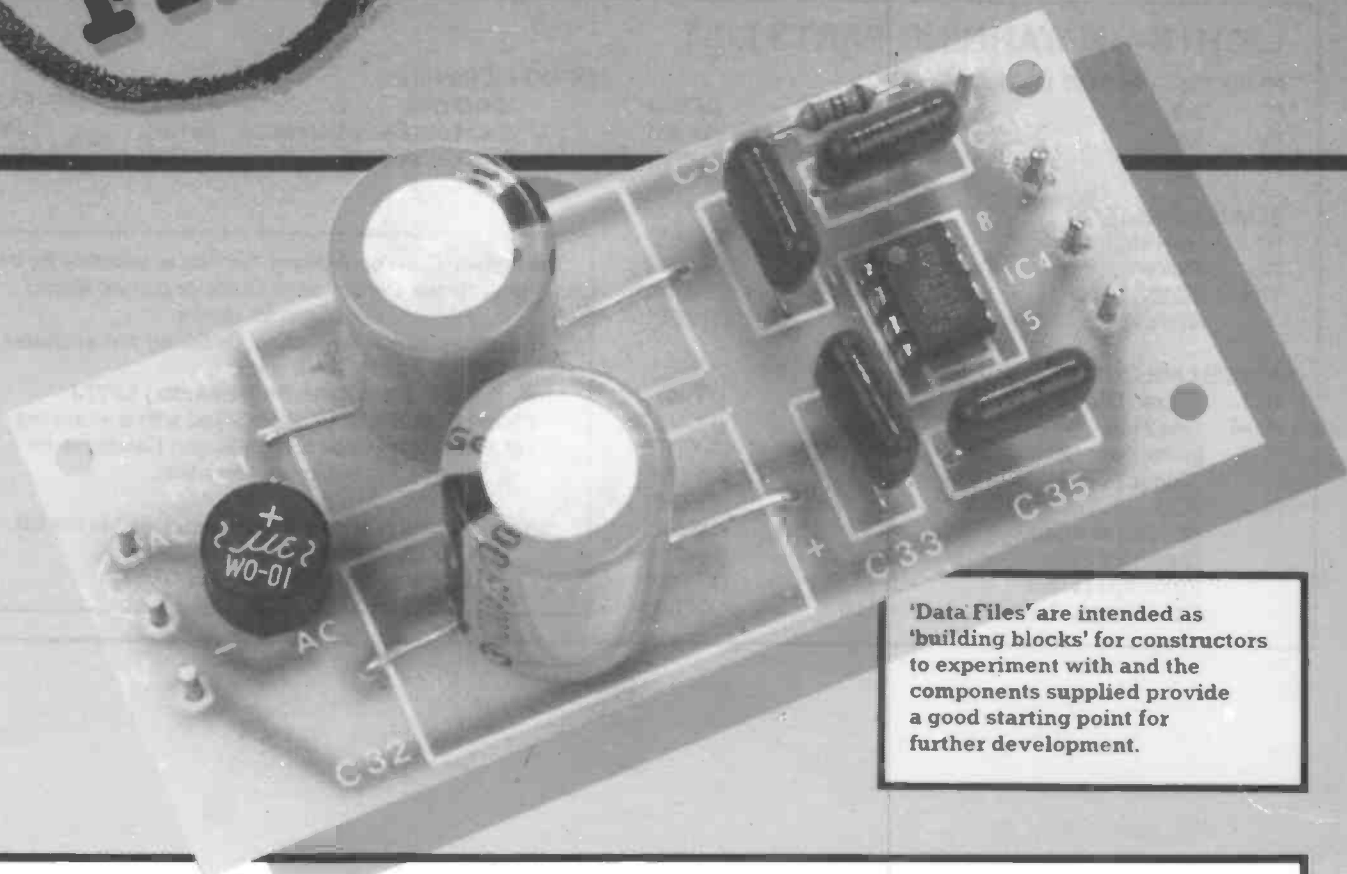
Order As (Lights-On Reminder) LP77J

Please Note: Order Codes marked with a ★ are not available singly, see current Maplin Catalogue for full ordering information.

The following new item (which is included in the kit) is also available separately.

Lights-On PCB **Order As GE88V**

DATA FILE RC4195 ±15V REGULATOR



'Data Files' are intended as 'building blocks' for constructors to experiment with and the components supplied provide a good starting point for further development.

FEATURES

★ ±15V operational amplifier power supply with low component count ★ Thermal shutdown ★ Short circuit protection ★ ±100mA output current ★ Can be used as a high voltage single output regulator

The RC4195 is a dual-polarity 'tracking' regulator designed to provide balanced positive and negative 15V output voltages. This device is designed for local 'on-card' regulation, eliminating distribution problems associated with single-point regulation. Only two external components are required for operation (two 10μF output decoupling capacitors).

Device Description

The IC can supply currents of up to 100mA per supply rail.

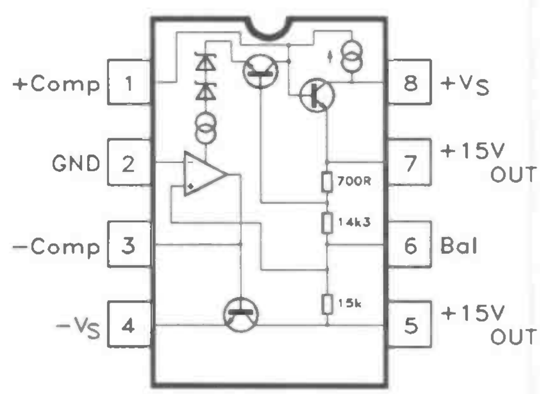


Figure 1. Functional block diagram.

To keep the device within its maximum power dissipation figure of 600mW, the maximum input voltage should be 18V. This is demonstrated by the following equation:

Power dissipation per rail = $(V_i - V_o) . I_L$, where:

V_i = Input voltage
V_o = Output voltage
I_L = Load current

This figure is multiplied by 2 (as there are two rails) to give the total IC power dissipation.

Substituting in the correct figures gives:

$((18-15) \times 0{\cdot}1) \times 2 = 0{\cdot}6W$

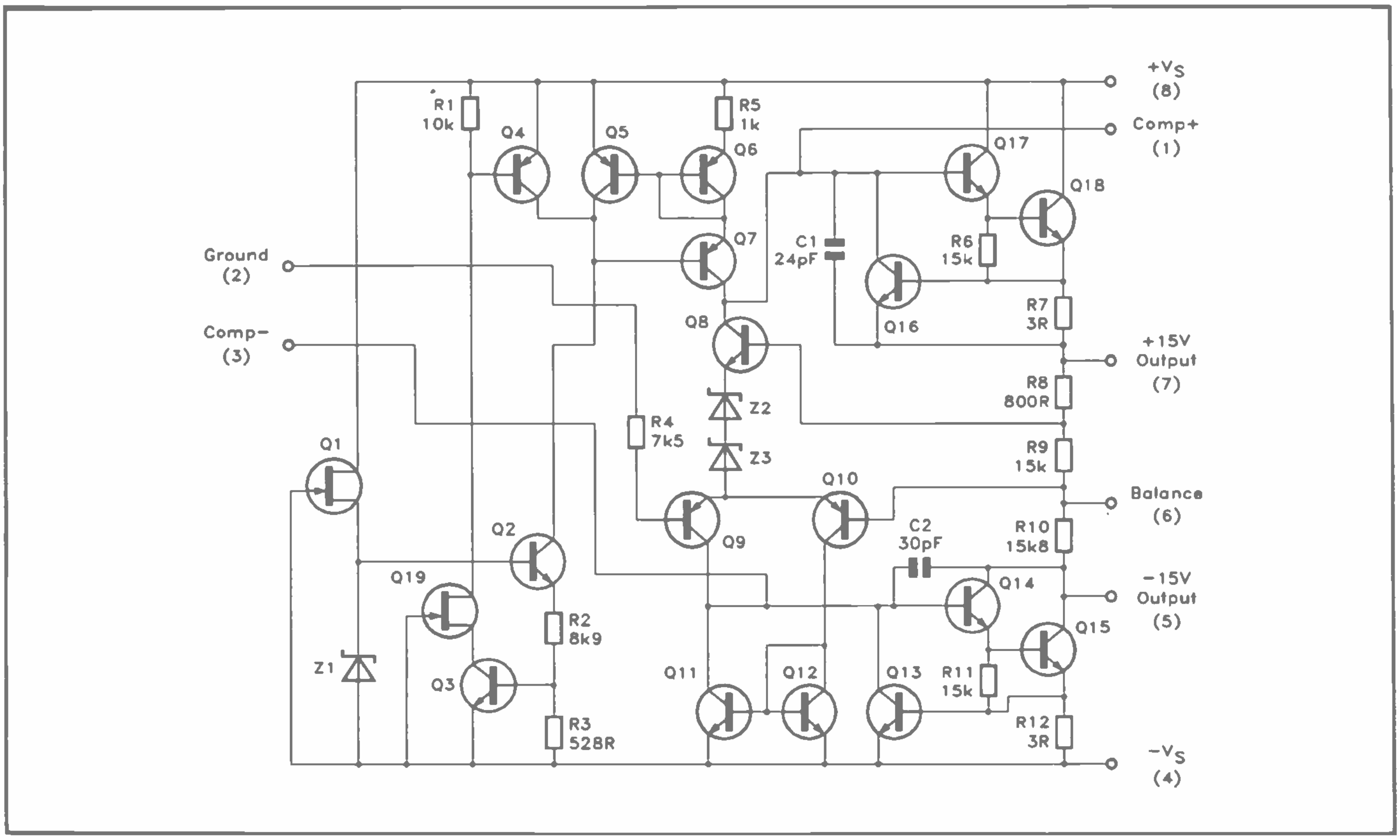


Figure 2. IC circuit diagram.

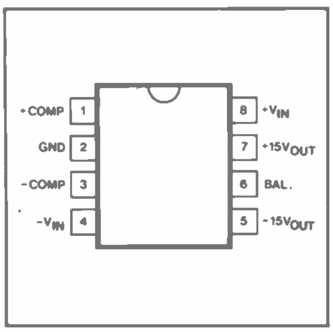


Figure 3. 4195 IC pin-out.

A functional block diagram is given in Figure 1. Figure 2 reveals the internal circuit diagram, while Figure 3 shows the IC pin-out. Tables 1 and 2 give the electrical and thermal characteristics respectively, while Table 3 gives the absolute maximum ratings. Graphs 1 to 6 show typical performance characteristics of the device. Figure 4 shows a typical application of the regulator, while Figure 5 shows the regulator configured to give a single high voltage output. Figure 6 shows how to use external pass transistors and current limiting circuitry to increase output current delivery. To balance the output voltages a potentiometer is fitted, with its resistive element connected across the output supply rails and its wiper connected to the 'balance' input, as shown in Figure 7.

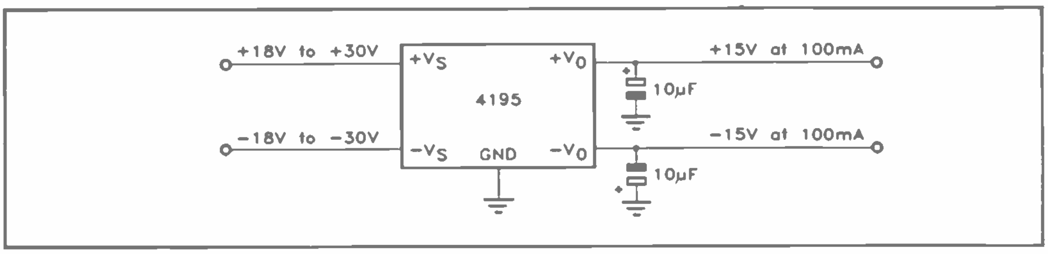


Figure 4. 4195 set up in balanced output configuration.

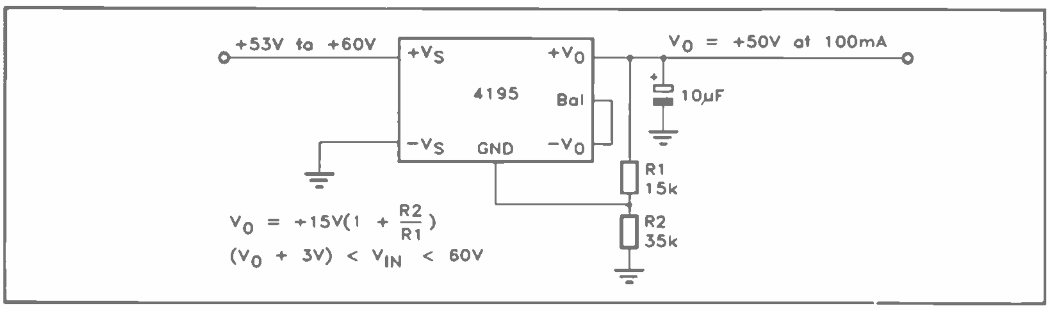


Figure 5. Positive single supply ($+15V < V_o < +50V$).

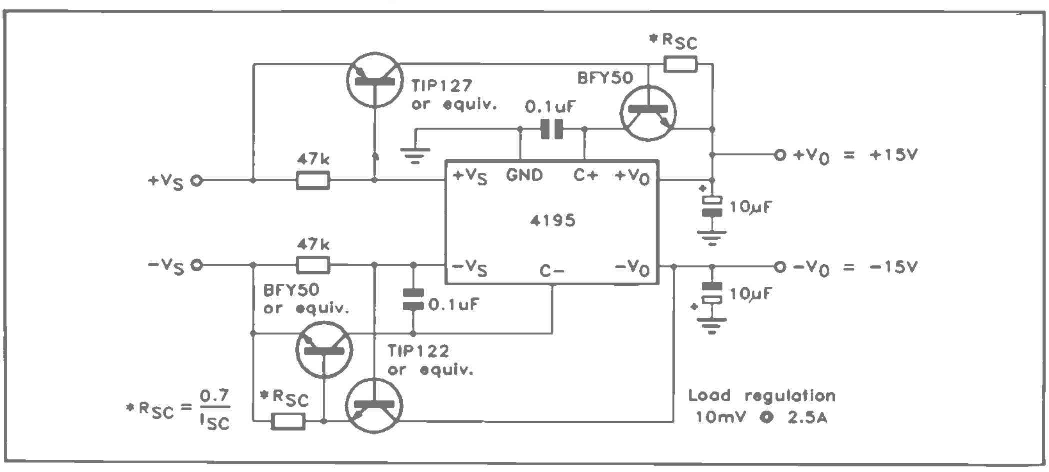


Figure 6. 4195 set up for high-current output.

($I_L = \pm 1mA$; $V_i = \pm 20V$; $C_L = 10\mu F$; $T_A = 0°C$ to $+70°C$).

Parameter	Conditions	Min.	Typ.	Max.
Line regulation:	$V_i = \pm 18V$ to $\pm 30V$		2mV	20mV
Load regulation:	I_L = 1mA to 100mA		5mV	30mV
Output voltage drift with temperature:			0·005%/°C	0·015%/°C
Supply current:	$V_i = \pm 30V$; I_L = 0mA		±1·5mA	±4·0mA
Supply voltage:		18V		30V
Output voltage:	T_j = +25°C	14·5V	15·0V	15·5V
Output voltage tracking:			±50mV	±300mV
Ripple rejection:	f = 120Hz; T_j = +25°C		75dB	
Input-Output voltage differential:	I_L = 50mA	3V		
Short circuit current:	T_j = +25°C		220mA	
Output voltage noise:	T_j = +25°C; f = 100Hz to 10kHz		60μV RMS	
Internal thermal shutdown:			175°C	

NOTE 1. The specifications given above apply for the given junction temperature, since pulse test conditions are used.

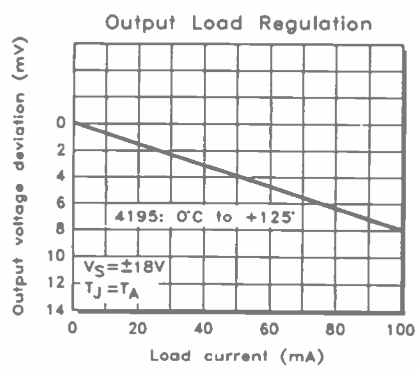
Table 1. Electrical characteristics.

Maximum junction temperature:	125°C
Maximum power dissipation (T_A <50 °C):	468mW
Thermal resistance θJ_A:	160°C/W
For T_A >50°C, derate at:	6·25mW per °C

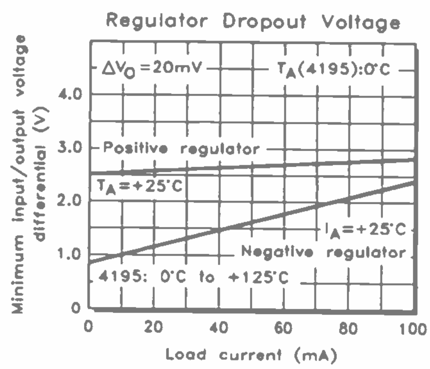
Table 2. Thermal characteristics.

Supply voltage (±Vi) to ground:	±30V
Load current:	100mA
Operating junction temperature range:	0°C to +125°C
Storage temperature range:	−65°C to +150°C
Lead soldering temperature (10 sec.):	+300°C

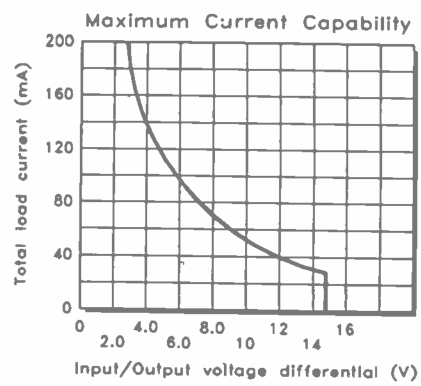
Table 3. Absolute maximum ratings.



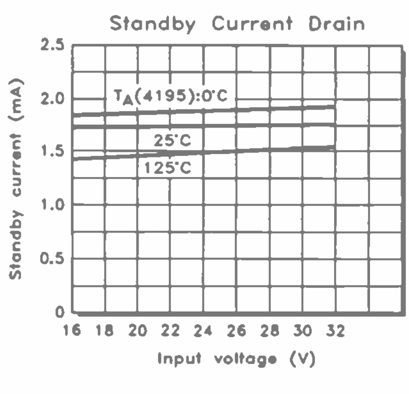
Graph 1. Output load regulation.



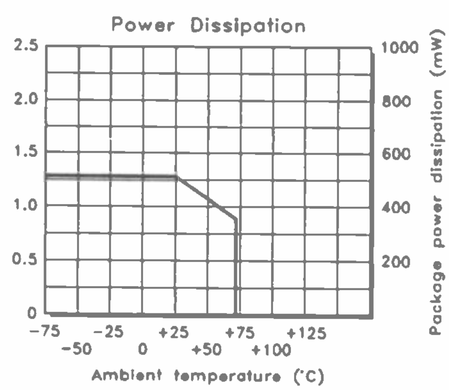
Graph 2. Regulator dropout voltage.



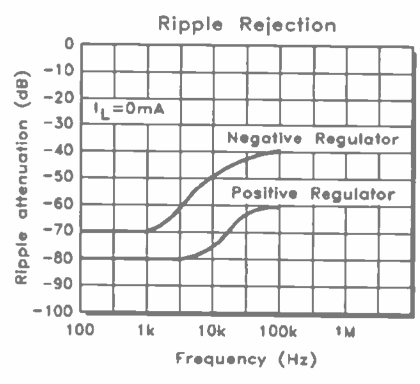
Graph 3. Maximum current capability.



Graph 4. Standby current drain.



Graph 5. Power dissipation.



Graph 6. Ripple rejection.

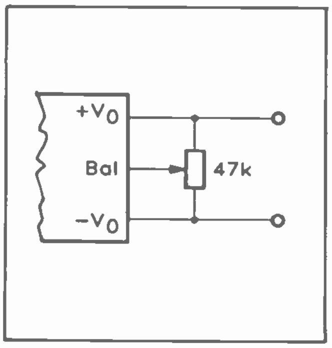


Figure 7. Balancing the output rails.

Kit Available

A complete kit of parts (including a high quality fibreglass PCB with a component legend to aid component positioning) is available, allowing a basic ±15V power supply to be constructed using the RC4195 IC. To aid heat dissipation, it is recommended that the regulator device is soldered directly to the PCB, so that the copper track will act as a heatsink. The circuit diagram for the kit is given in Figure 8, and the PCB legend is shown in Figure 9.

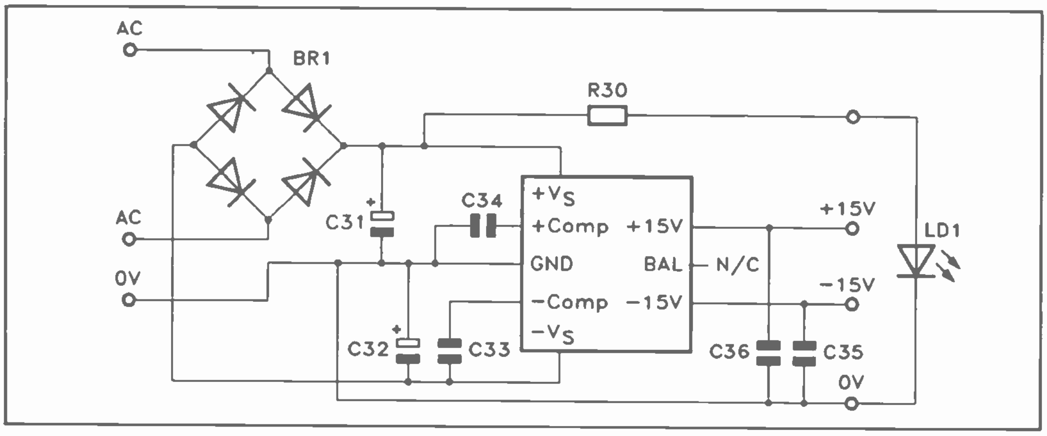
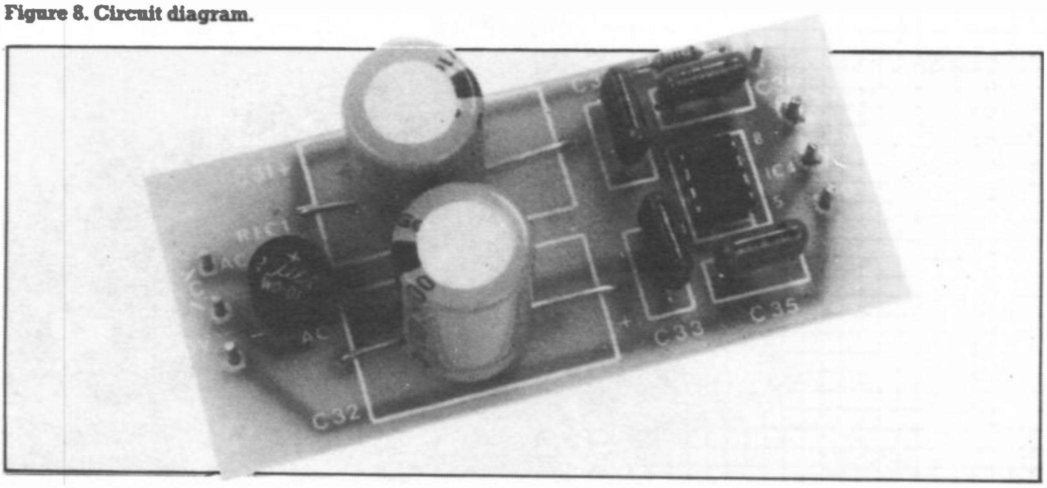


Figure 8. Circuit diagram.



The assembled ±15V regulator PCB.

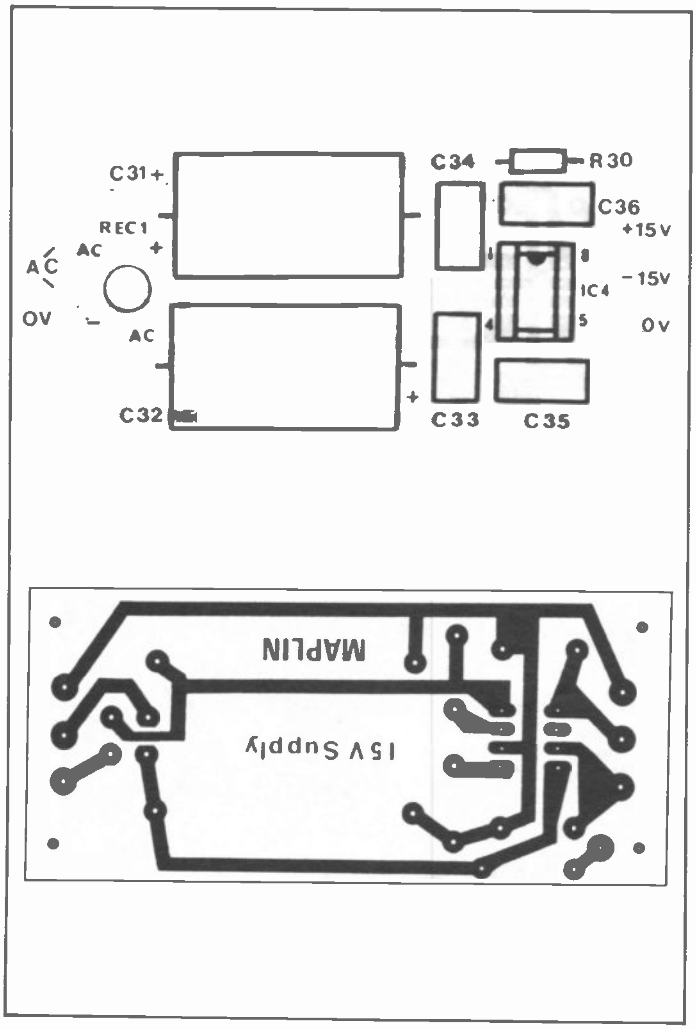


Figure 9. PCB legend and track.

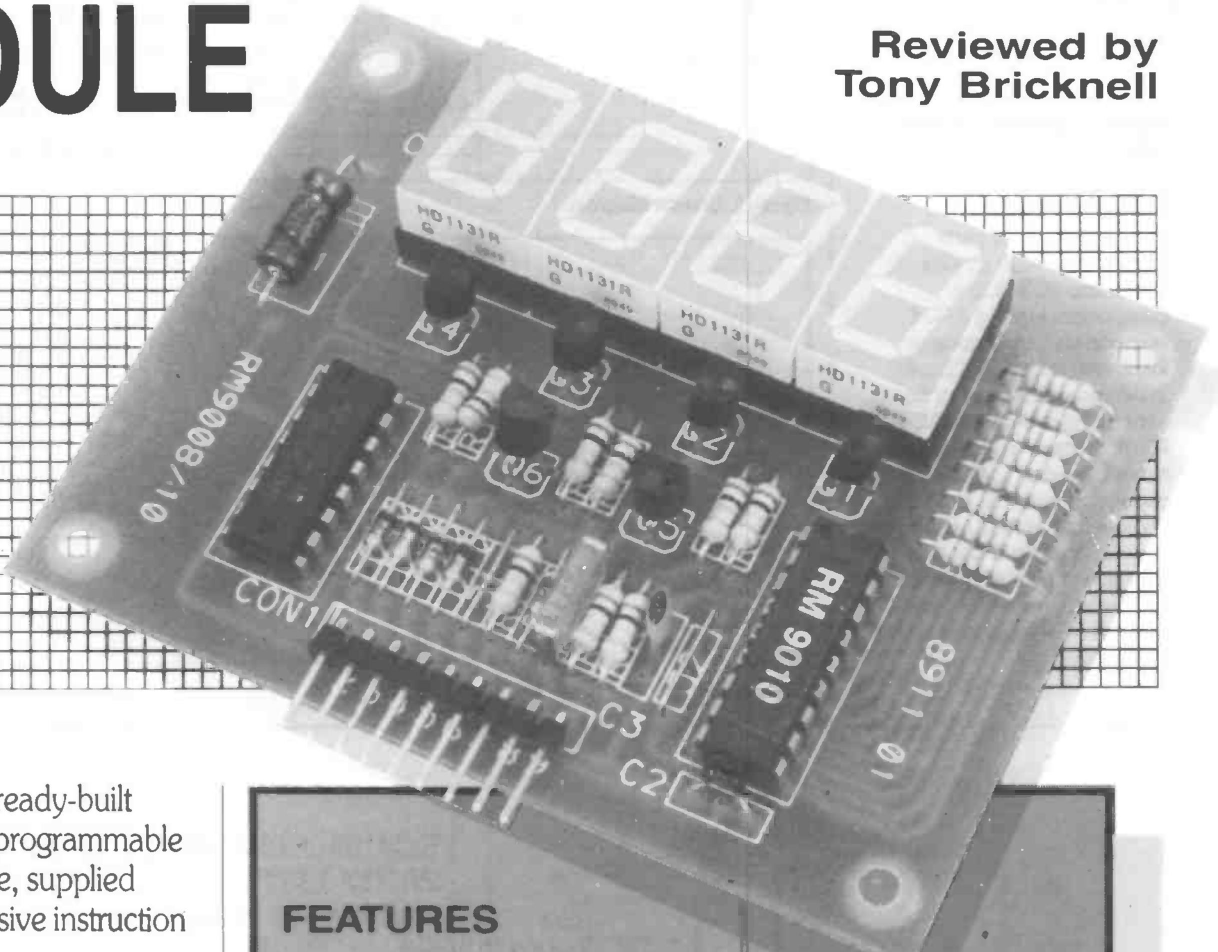
RC4195 ±15V REGULATOR PARTS LIST

RESISTOR: 0·6W 1% Metal Film			
R30	1k8	1	(M1K8)
CAPACITORS:			
C31,32	PC Elect 1000μF 35V	2	(FF18U)
C33,34,35,36	Polyester 0·1μF	4	(BX76H)
SEMICONDUCTORS:			
IC4	4195	1	(XX02C)
BR1	W01	1	(QL38R)
LD1	LED Red	1	(WL27E)
MISCELLANEOUS:			
	Pin 2141	7 Pins	(FL21X)★
	PCB	1	(XX04E)
	4195 D/F Leaflet	1	(XT33L)
	Constructors' Guide	1	(XH79L)

The Maplin 'Get-You-Working' service is not available for this project.
The above items are available as a kit.
Order As LP88V (4195 Data File)
Please Note: Order Code marked with a ★ is not available singly, see current Maplin Catalogue for full ordering information.

PROGRAMMABLE TIMER/CLOCK MODULE

Reviewed by
Tony Bricknell



The RM9010 is a ready-built single-board fully programmable timer/clock module, supplied with a comprehensive instruction leaflet.

Figure 1 shows the circuit diagram of the module. All connections to the module are present on a single 9-way connector, CON1. The wiring diagram of the RM9010 is shown in Figure 2. Four non-latching push-to-make switches are required to control the timer (SW1-SW4), and are connected to CON1-1 to CON1-4, with a common connection on CON1-5. An open-collector switched output is available on CON1-7 enabling the module to be used with a variety of different equipment. Note that this can only switch a maximum of 50V/100mA. To facilitate the switching of higher currents, the output may be used to switch a separate, off-board relay. When switching DC resistive or inductive loads up to 16A, it is recommended that the 'Remote Power Switch' kit is used (LP07H).

For switching 240V AC mains, it is recommended that the 'Mains Opto-Switch' kit (LP55K) is used. This employs a zero-crossing fired triac to virtually eliminate all switching noise that would otherwise occur. Note that TR1 should not be inserted, and instead a wire link should be placed between its collector and emitter pads on the PCB, as shown in Figure 2.

FEATURES

- ★ ½ inch 4-digit LED display
- ★ 2 programmable 'on' times
- ★ 2 programmable 'off' times
- ★ Manual on/off override and timer enable/disable function
- ★ Single +5V/100mA supply required
- ★ Clock input for mains frequency
- ★ 50V/100mA Open-collector switched output

APPLICATIONS

- ★ Security lighting control
- ★ Recording radio transmissions at preset times
- ★ Immersion heater control
- ★ Central heating control

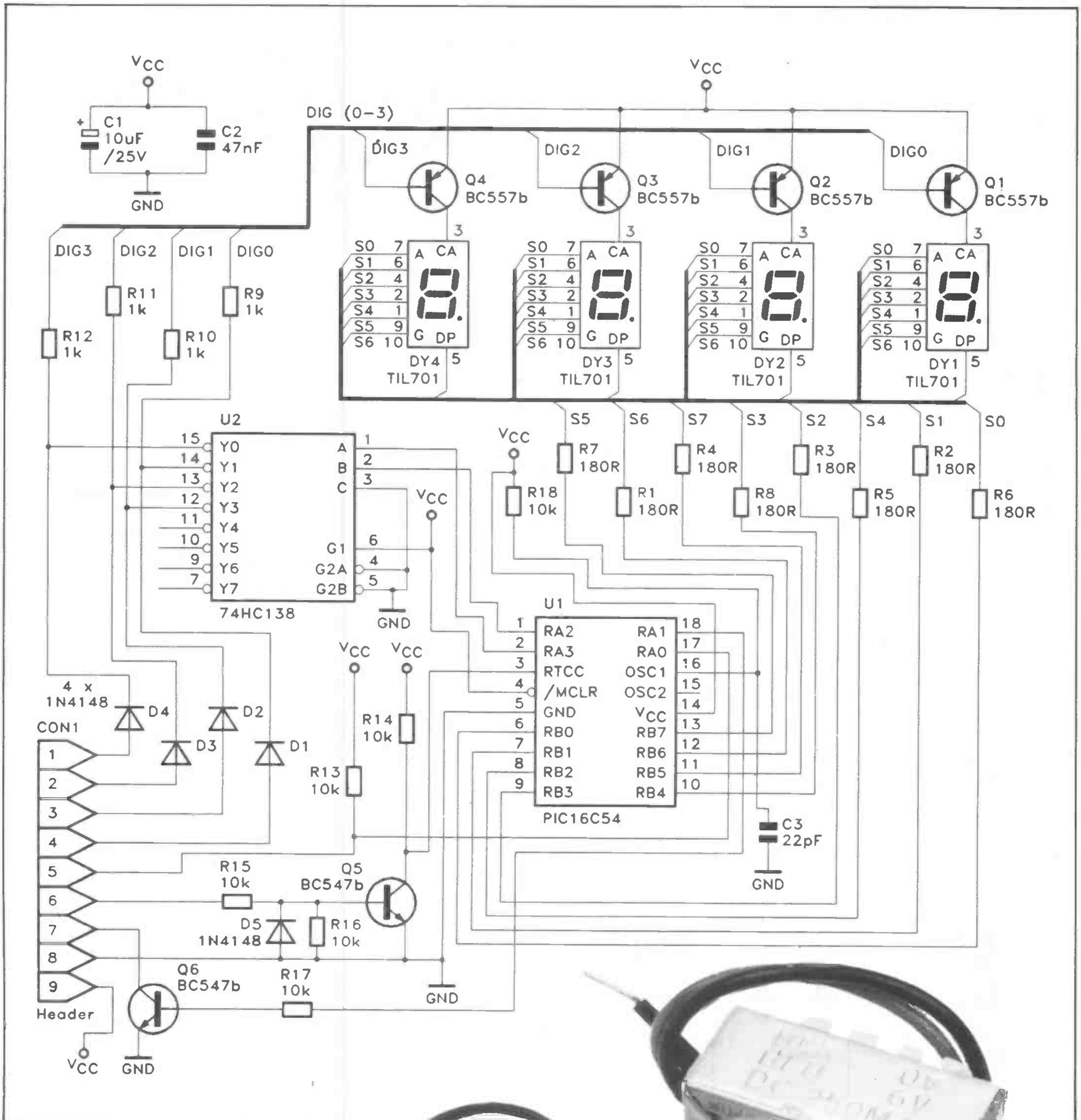


Figure 1. Circuit diagram.

Power Supply

The RM9010 requires a single supply voltage of +5V, and will draw an absolute maximum of 100mA. This is applied to CON1-9 (+5V) and CON1-8 (0V), and a 50Hz timebase applied to CON1-6. Figure 3 shows a suitable power supply which enables the module to be used with 240V 50Hz mains. This circuit can be built on stripboard; a suggested layout is shown in Figure 4. Alternatively, using the circuit shown in Figure 5, the module can operate from a DC voltage of between 7 and 15V. In this circuit, the 50Hz timebase is generated by IC1, with the benefit that the module can remain operative during a mains failure. A suggested stripboard layout for this circuit is shown in Figure 6.

Photos 1 and 2 show the two power supply circuits, each assembled on pieces of stripboard.

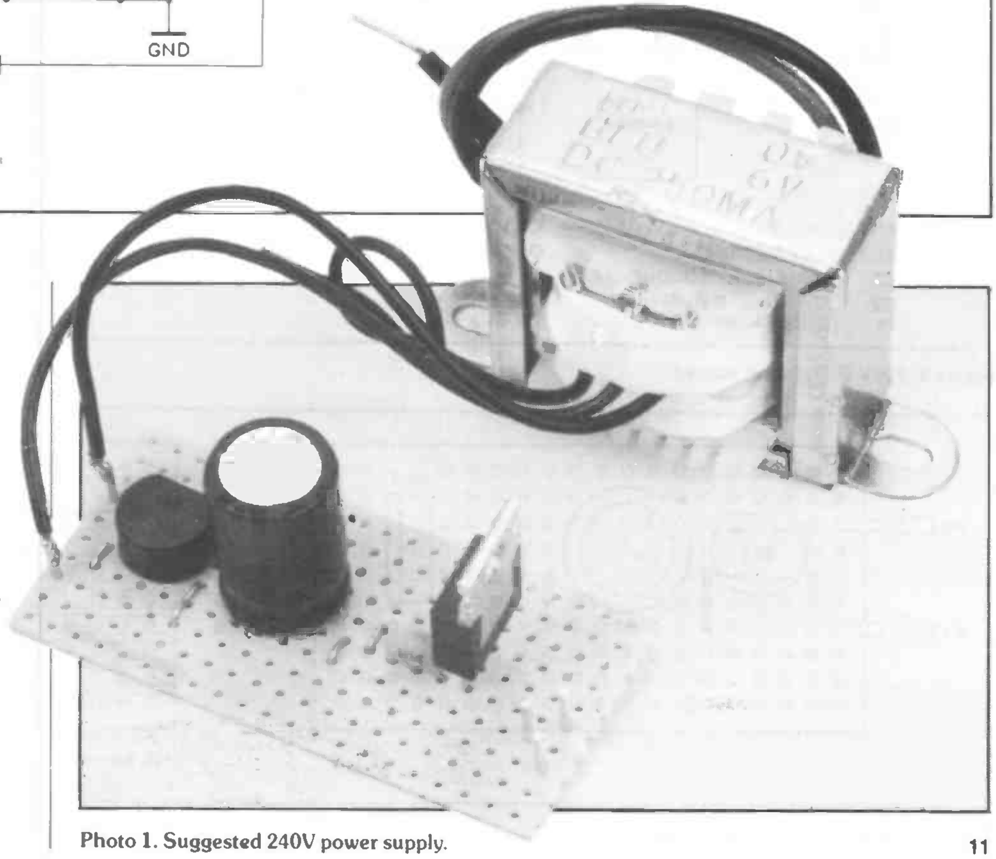


Photo 1. Suggested 240V power supply.

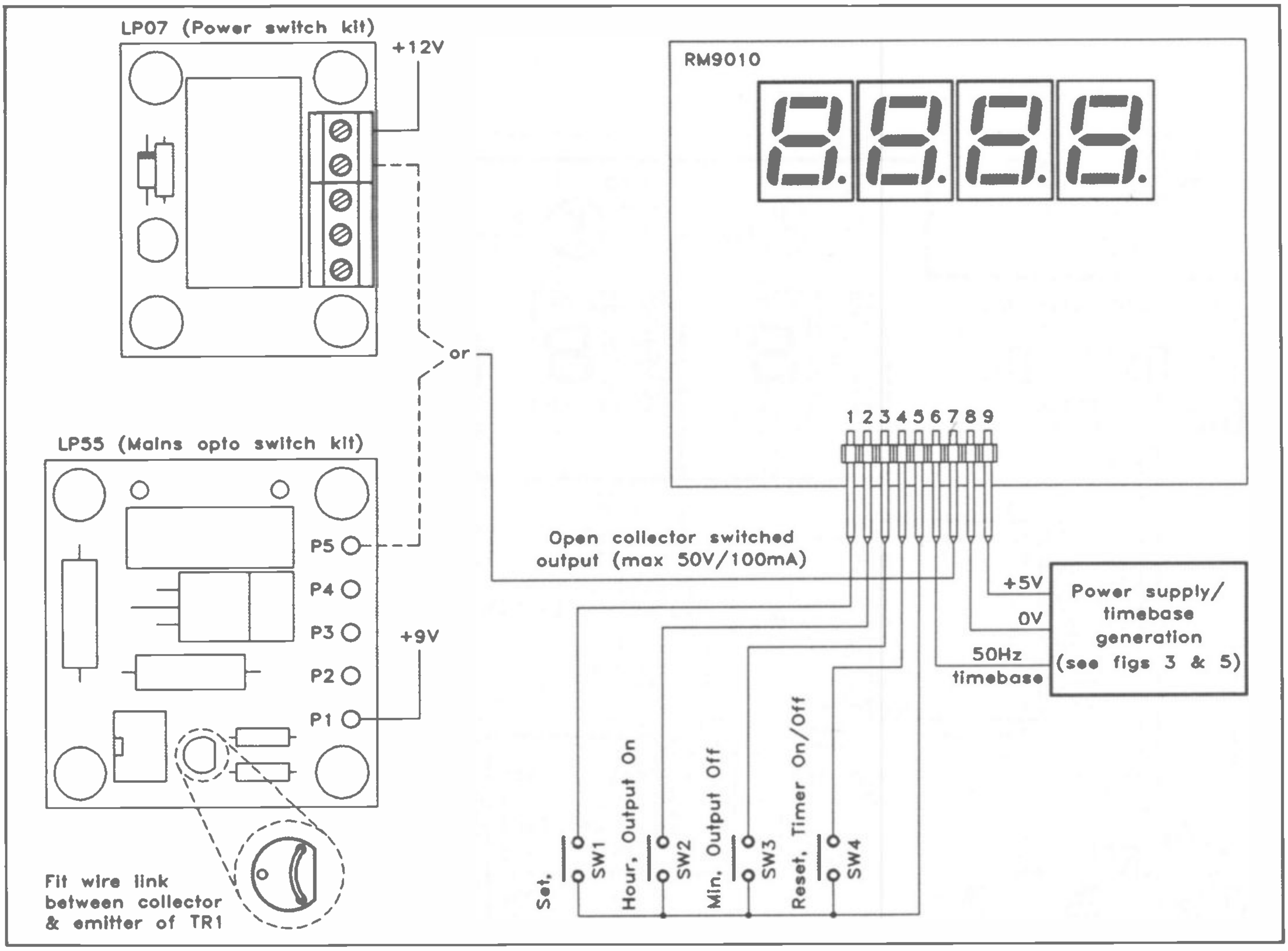


Figure 2. Wiring diagram.

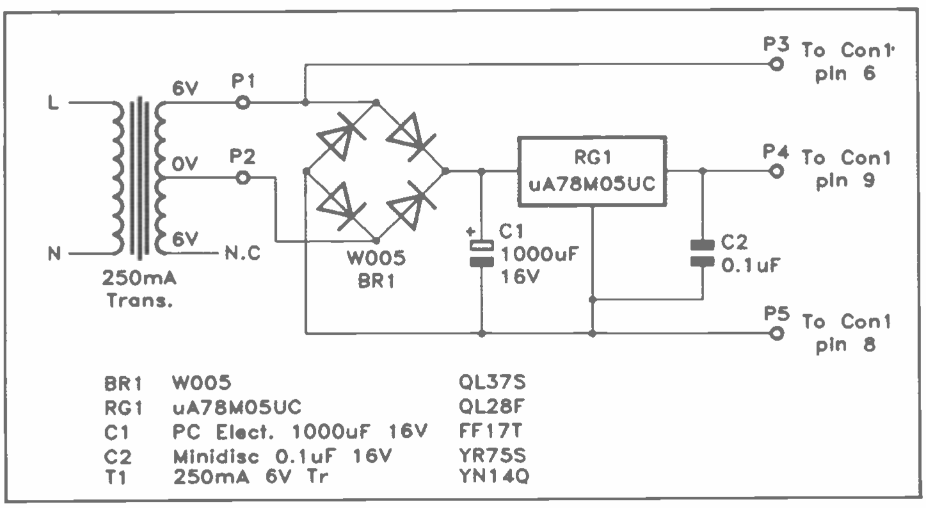


Figure 3. 240V 50Hz power supply.

Operating Instructions

On initial power-up, all internal registers are set to zero and the module is in 'Set' mode.

The flashing word 'hour' now invites the user to set the clock to the correct time, using the HOUR and MIN buttons. If an incorrect value is entered, pressing the RESET button returns the displayed time to 0.00.

Once the correct time has been entered, pressing the SET button several times allows the ON1, OFF1, ON2 and OFF2 times to be displayed. These can be edited in the same way as the clock time, by using the HOUR, MIN and RESET switches.

After displaying the OFF2 time, the module reverts to normal clock mode, indicated by the middle decimal point

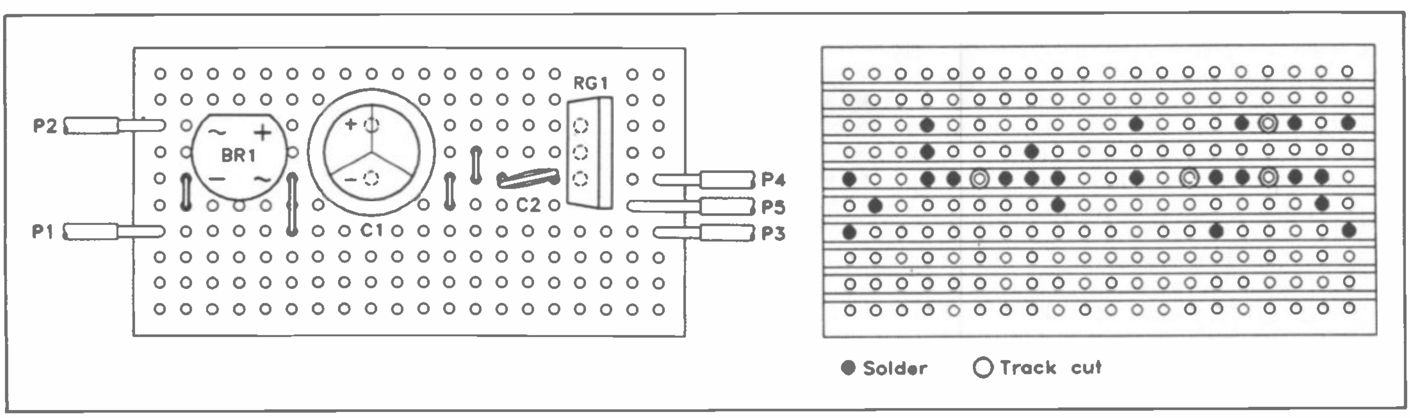


Figure 4. Suggested 240V stripboard layout.

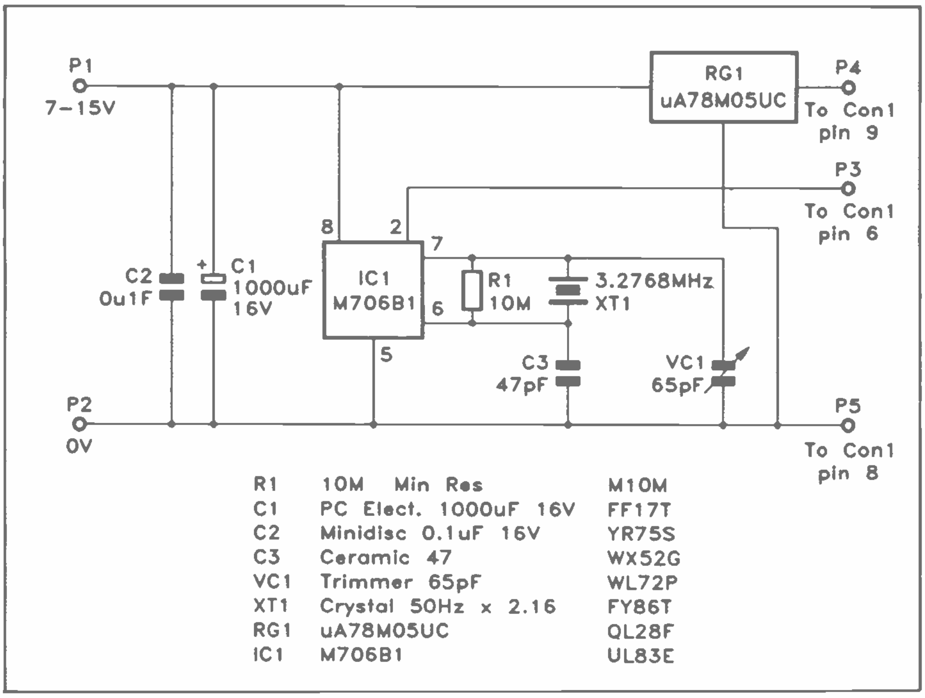


Figure 5. 7 to 15V DC power supply.

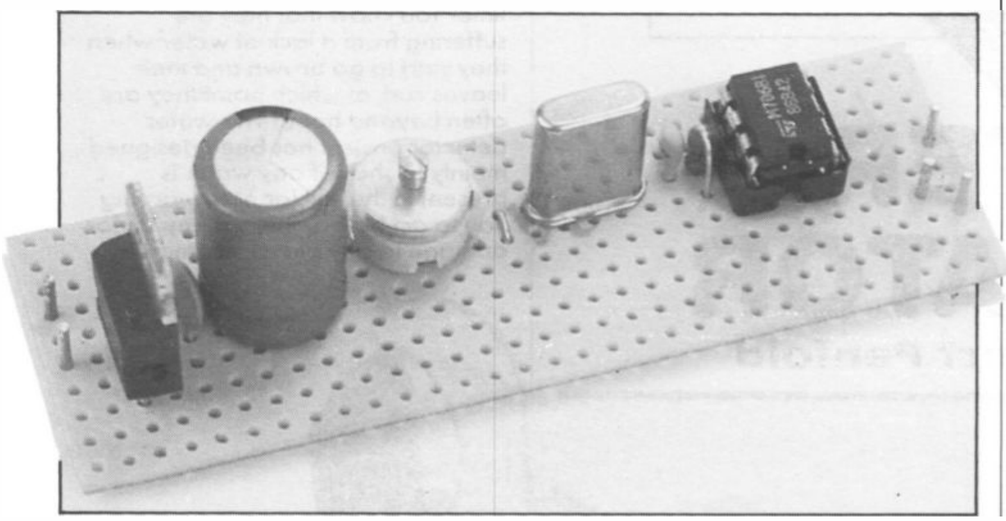


Photo 2. Suggested 7 to 15V power supply.

flashing. In the clock mode, the functions of the HOUR, MIN and RESET switches change to those shown below:

HOUR – Manually turn the output on.
MIN – Manually turn the output off.
RESET – Controls whether the output is switched according to the programmed values or not, by turning the timer function on and off. When the timer is disabled, the middle decimal point is continuously lit.

The status of the output is always shown by the right-hand decimal point – when illuminated, the output is on.

To re-set the clock, or the programmed ON and OFF times, it is necessary to enter the set mode by pressing the SET button. Successive depressions of the SET button will now take the module through CLOCK, ON1, OFF1, ON2 and OFF2 display/set modes.

Applications

This compact clock/timer module, with its two 24-hour on/off switch times, is ideal for controlling immersion heaters and electric blankets, and for turning lighting on and off when on holiday (to give the impression that the house is occupied, etc).

In shops, the module could be used to switch window display lighting at peak times and, in industrial process control applications, valves and heating elements could be switched at fixed time intervals.

Availability

The Programmable Timer/Clock module is obtainable from Maplin Electronics by mail-order or through their numerous regional stores. The order code is LP87U.

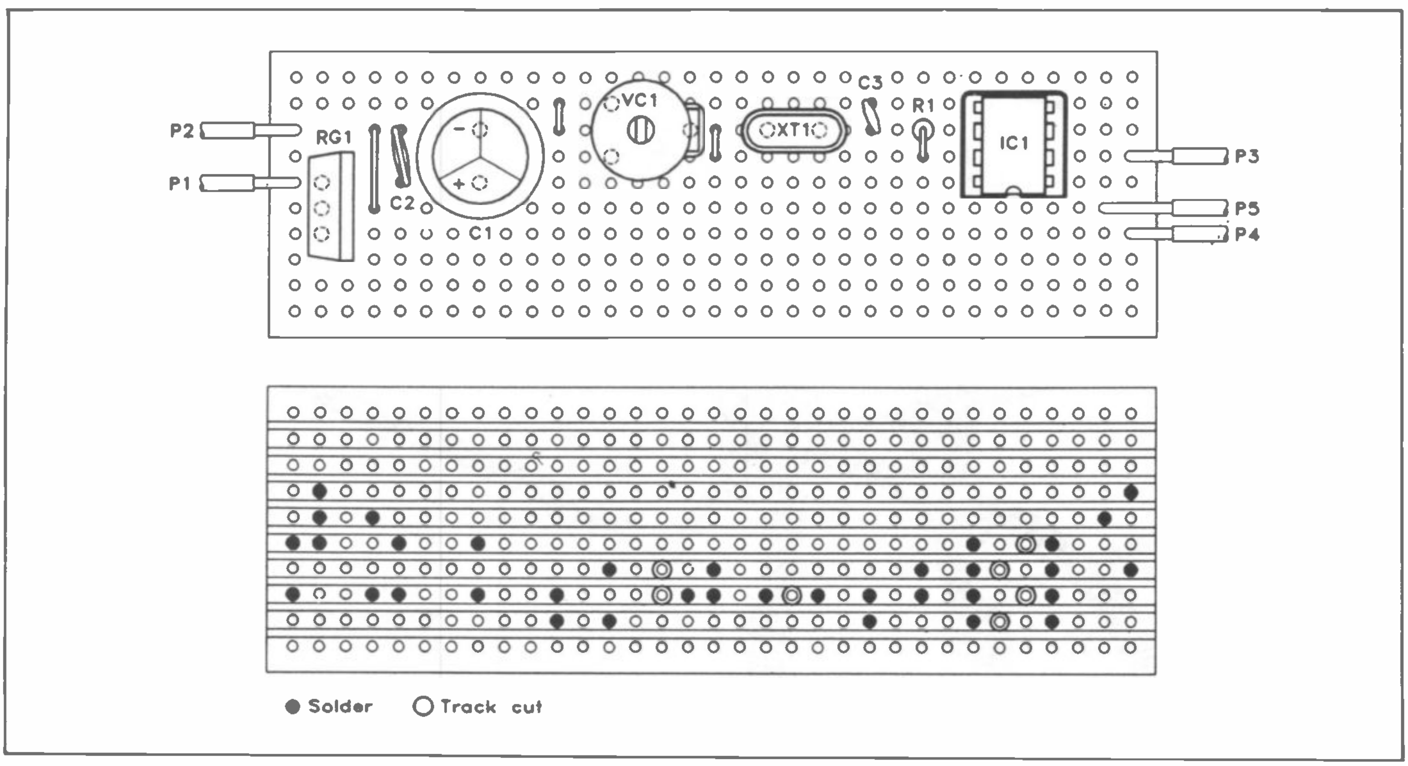


Figure 6. Suggested 7 to 15V stripboard layout.

All You Need to Build Your Own

FUN TRONICS

WATER INDICATOR

Text by Robert Penfold

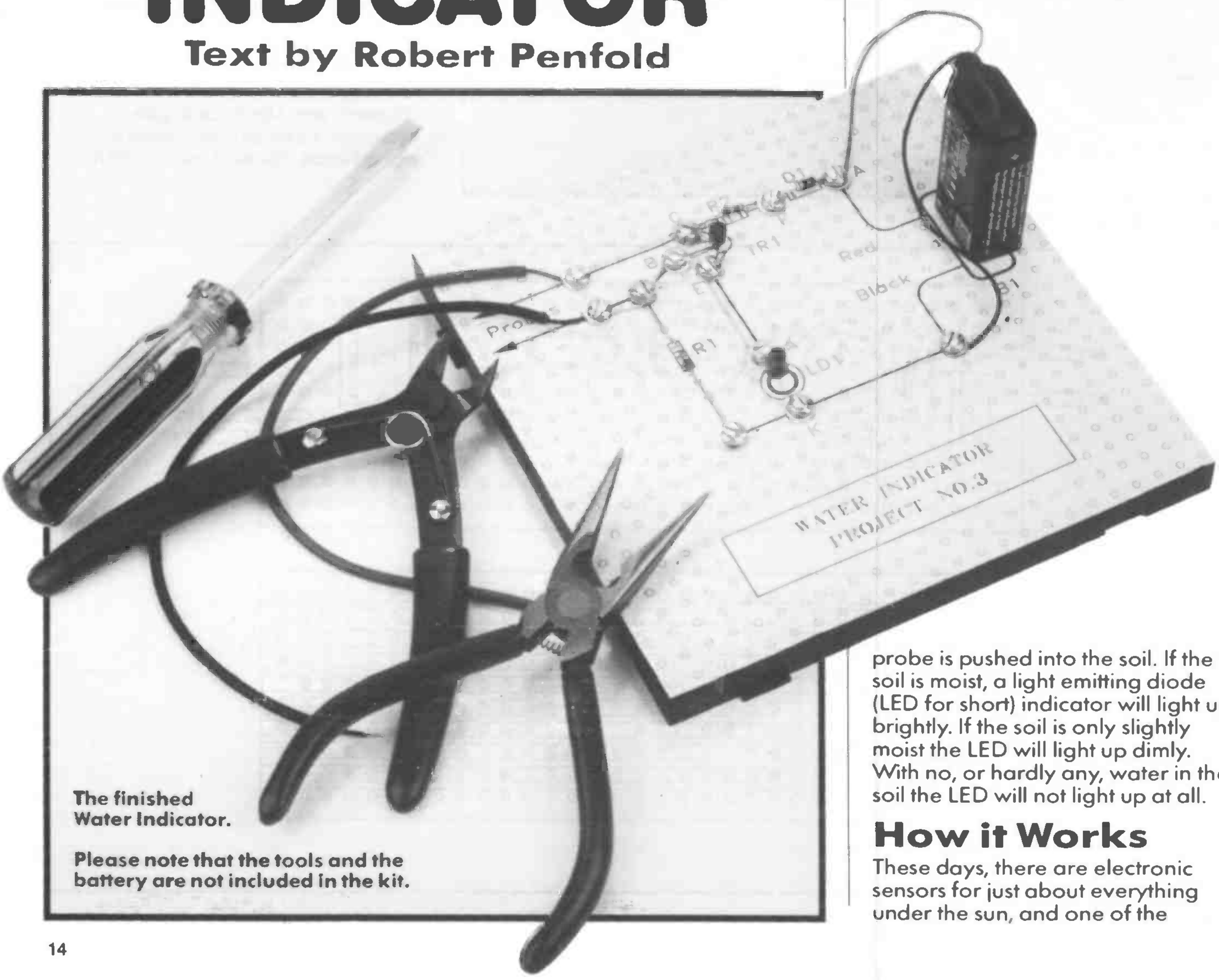
This is the third in a series of easy-to-build electronics projects for complete beginners, who require a simple and fun starter to electronic project building. These projects are ideal for the young person as no soldering is needed. All the projects are built on the same type and size of plastic 'peg-board'. The only tools needed to build this project, and the others in the series, are a pair of wire cutters/strippers and a small screwdriver; a pair of pliers would also be useful.

This Month's Project

As no species of plant have yet learned to speak, they can not tell you if they are thirsty. This has resulted in many pot plants becoming dead twigs before their time! You know that they are suffering from a lack of water when they start to go brown and their leaves curl, at which point they are often beyond hope! This water detector project has been designed mainly to show if any water is present in the soil (or not), warning you to act before there is any major damage to any plants. A simple probe is pushed into the soil. If the soil is moist, a light emitting diode (LED for short) indicator will light up brightly. If the soil is only slightly moist the LED will light up dimly. With no, or hardly any, water in the soil the LED will not light up at all.

How it Works

These days, there are electronic sensors for just about everything under the sun, and one of the



The finished Water Indicator.

Please note that the tools and the battery are not included in the kit.

easiest things to sense is water. Actually this is not quite true. *Pure* water does not conduct electricity very well, making its presence difficult to detect. However, the water normally found in the everyday world is not completely pure, and even tiny amounts of impurity are enough to make water able to pass electricity. This makes it very easy to detect. Such impurities are present in tap water, and even in rain water. The water found in soil certainly contains enough impurities to make it easy to detect.

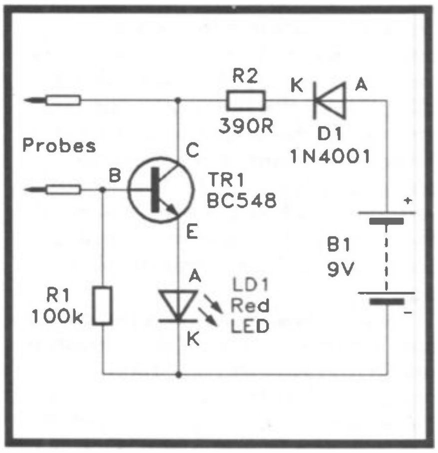


Figure 1. The Water Indicator circuit diagram.

Figure 1 shows the circuit diagram for the Water Indicator project. The probes consist simply of two pieces of insulated wire (with their ends stripped bare), which are pushed into the soil. In electronic terms, the 'resistance' of moist soil, to the flow of an electric current, is quite low. However, using a small probe does not permit a large enough current flow to drive an LED. Instead, a simple transistor amplifier must be used, allowing the tiny amount of electricity flowing through the soil to produce a larger flow through the LED.

TR1 is the transistor amplifier, and LD1 is the indicator LED connected at TR1's output. R2 limits the supply current to a level that is safe for TR1 and LD1. Some of the current from R2 will also flow into the probe. This will pass through the the soil (which simply acts as another resistor), and then back through the other probe, where it can be amplified by TR1. R1, another resistor, is present to 'tap off' some of the current from the probe, and return it to the battery. This is done to make the circuit less sensitive, otherwise LD1 would glow brightly even if there was a lot of resistance (in other words, hardly any water) present in the soil. D1 is a protection diode; if you should accidentally try to connect the battery the wrong way round, D1 will block the flow of electricity. This ensures that none of the components will be damaged.

Getting it Together

Firstly, read through this section and then *carefully* follow its instructions, one step at a time. Refer to the photographs of the finished project if this helps.

1. Cut out the component guide-sheet provided with the kit (which is a full-size copy of Figure 2), and glue it to the top of the plastic board. Paper glue or gum should be okay. Do not soak the paper with glue, a few small 'dabs' will do.

2. Fit the link-wires to the board using the self-tapping screws and washers provided. The link-wires are made from bare wire. Loop the wire, in a *clockwise* direction around each screw to which it must connect, taking the wire under the washers. Do not fully tighten a screw until all the leads that are under it are in place, and do not over-tighten the screws, otherwise the plastic board may be damaged.

3. Recognise and fit the components, in the order given below, using the same method as for the link wires. Cut the components' wires so that they are just long enough to loop around the screws; otherwise long leads left flapping around may touch each other (this is known as a 'short-circuit') and may stop your circuit from working.

a) The first components to be fitted are Resistors R1 and R2. These are small sausage-like components having a leadout wire at each end, and several coloured bands around their bodies. For R1 these colours are brown, black, yellow, and gold. The colours for R2 are orange, white, brown, and gold. These first three bands tell us the value of the resistor. R1 has a value of 100,000 ohms, which is often written as 100 kilohms, or 100kΩ for short. R2, however, has a value of 390 ohms (written as 390Ω or 390R for short.) The fourth band tells us how near to the given value the resistor is likely to be. This fourth band is known as the 'tolerance' band. Unlike diodes or transistors, it does not matter which way round resistors are connected.

b) Next fit the LED, LD1, which is a 'blob' of clear red plastic, with two wires coming out of one end. It is fitted in the position shown on the guide-sheet, and *must* be connected the right way round – or it will not light up. One side of LD1 is flattened (the lead on this side of the LED is known as the cathode (K), while the lead on the other, rounded, side is called the anode (A)), see Figure 3. Make sure that LD1 is fitted so that the 'flattened' side lines up with the drawing of the LED printed on the guide-sheet.

c) TR1 should be fitted next to the board. This has a small black

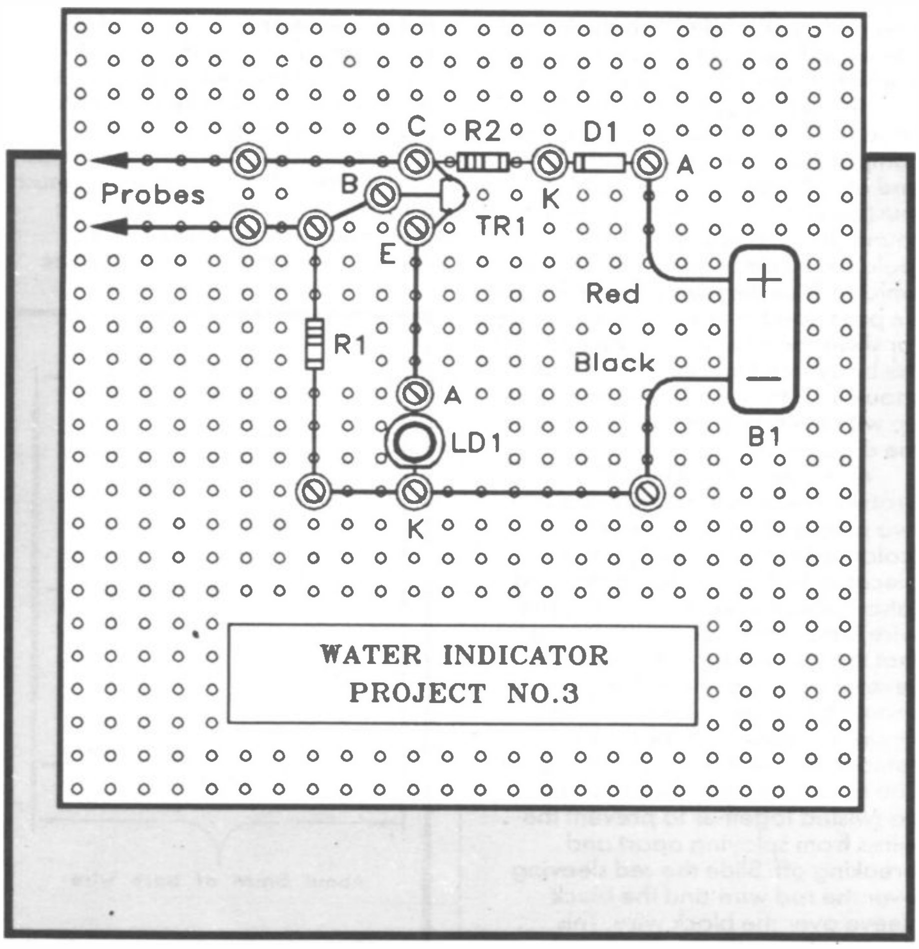


Figure 2. The layout of the components.

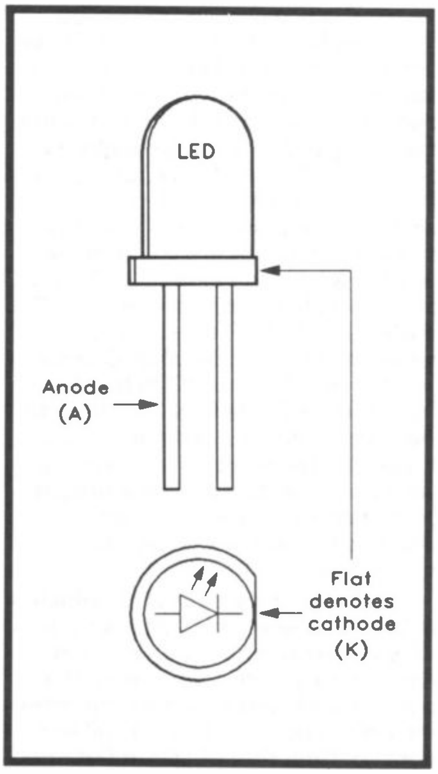


Figure 3. LED symbol and connections.

plastic body with three lead-out wires. It will be marked with the type number, which in this case is 'BC548'. Other markings may also be present; you will have to get used to picking out the important markings on chips and transistors (and ignoring the others!). You must ensure that TR1 is fitted to the board correctly. Figure 4 shows which lead is which, making this task easy.

d) The next component to be fitted is D1, which is a small tube-like component having a lead at each end of its black body. Like LD1, it must be connected the right way round (In other words, D1 is a 'polarised' component). Its 'polarity', which tells us the way in which it must be positioned, is indicated by a white (or silver) band close to one end of the body, see Figure 5. The diode should be fitted so that the band lines up with the band on the drawing of the diode on the guide-sheet.

e) We now need to make the probes. These are made up from two pieces of insulated wire (coloured red and black) and two pieces of hollow insulated sleeving (also coloured red and black). The wire is multi-stranded, which means that the metal core consists of several very fine wires. The probes should be made up as shown in Figure 6, use wire strippers to remove the insulation where shown. The bare ends of the leads should be twisted together to prevent the wires from splaying apart and breaking off. Slide the red sleeving over the red wire and the black sleeve over the black wire. This sleeving forms the body of each probe. Connect the two wires to the

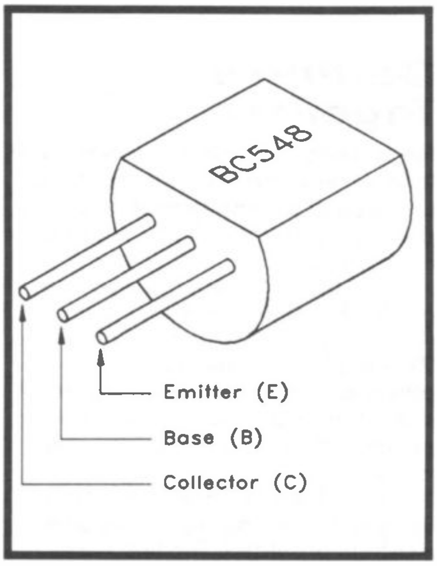


Figure 4. Transistor circuit symbols and lead identification.

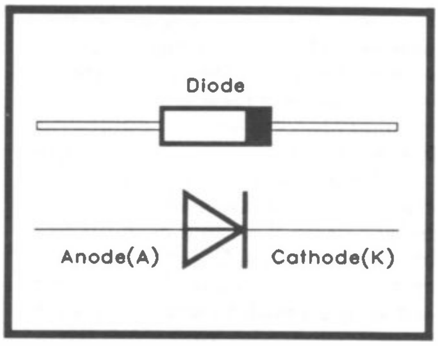


Figure 5. Diode symbol and connections (D1).

screws on the board marked 'Probes' – the red wire should go to the screw nearest to R2 (and the collector ('C') of the transistor). The free ends of the two wires (with the 8mm insulation stripped off) form the probes themselves.

f) Lastly fit the battery connector and battery. B1, the connector must be attached to the board with its coloured leads the correct way round. The battery connector has

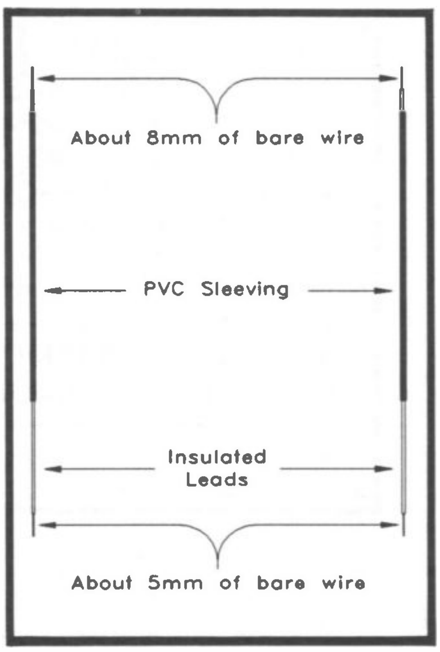


Figure 6. Preparing the probes.

two press-stud clips on a piece of plastic and two wires coming from it, coloured red and black. The red and black leads should be connected as shown on the layout sheet. The 9V PP3 type battery should be connected to the battery connector; it will only fit properly one way round.

Testing and Use

When the battery is connected, the LED should not light, although it will if the bare ends of the two probes are connected together. If the unit does not work in either case, disconnect the battery at once. The most likely cause of problems is TR1, LD1, D1, or the battery being connected the wrong way round. Check the connections to these components carefully. Also make sure that the fine strands of wire of the battery clip leads make good contact with their terminals on the board.

If all is well, the unit is ready for use. In order to obtain dependable results, the two wire probes should *always* be held the same distance (between 10 and 20mm) apart. It is a good idea to mount the probes on a wooden handle that will hold them a suitable distance apart. You might also like to use some fairly stout single-strand wire for the probes. If you wish to make the unit *more* sensitive, make the probes longer by removing more insulation from the wires. To make the unit *less* sensitive, trim the wires back slightly.

Other Uses

Lie Detector

This unit is really just a detector of high resistances. This can be the resistance of your body. If you try holding the bare ends of the two probes, LD1 might light up dimly. You will probably have to hold them quite hard to make anything happen, though. If you have no luck, try removing R1, which will substantially increase the sensitivity of the circuit. Simple lie detectors work by measuring skin resistance. In other words, if the subject (victim?) tells the truth the brightness of the LED will remain unchanged. If they tell a lie they sweat more, and the LED lights more brightly. Perhaps this is something not to be taken too seriously – because this method of lie detection has never proved very reliable in scientific tests!

Availability

The Funtronics Water Indicator is available from Maplin Electronics, through our chain of regional stores, or by mail order, order code LP90X.

WATER INDICATOR PARTS LIST

RESISTORS

R1	100k ¼W 5% Carbon Film
R2	390Ω ¼W 5% Carbon Film

SEMICONDUCTORS

TR1	BC548
LD1	LED Red
D1	1N4001

MISCELLANEOUS

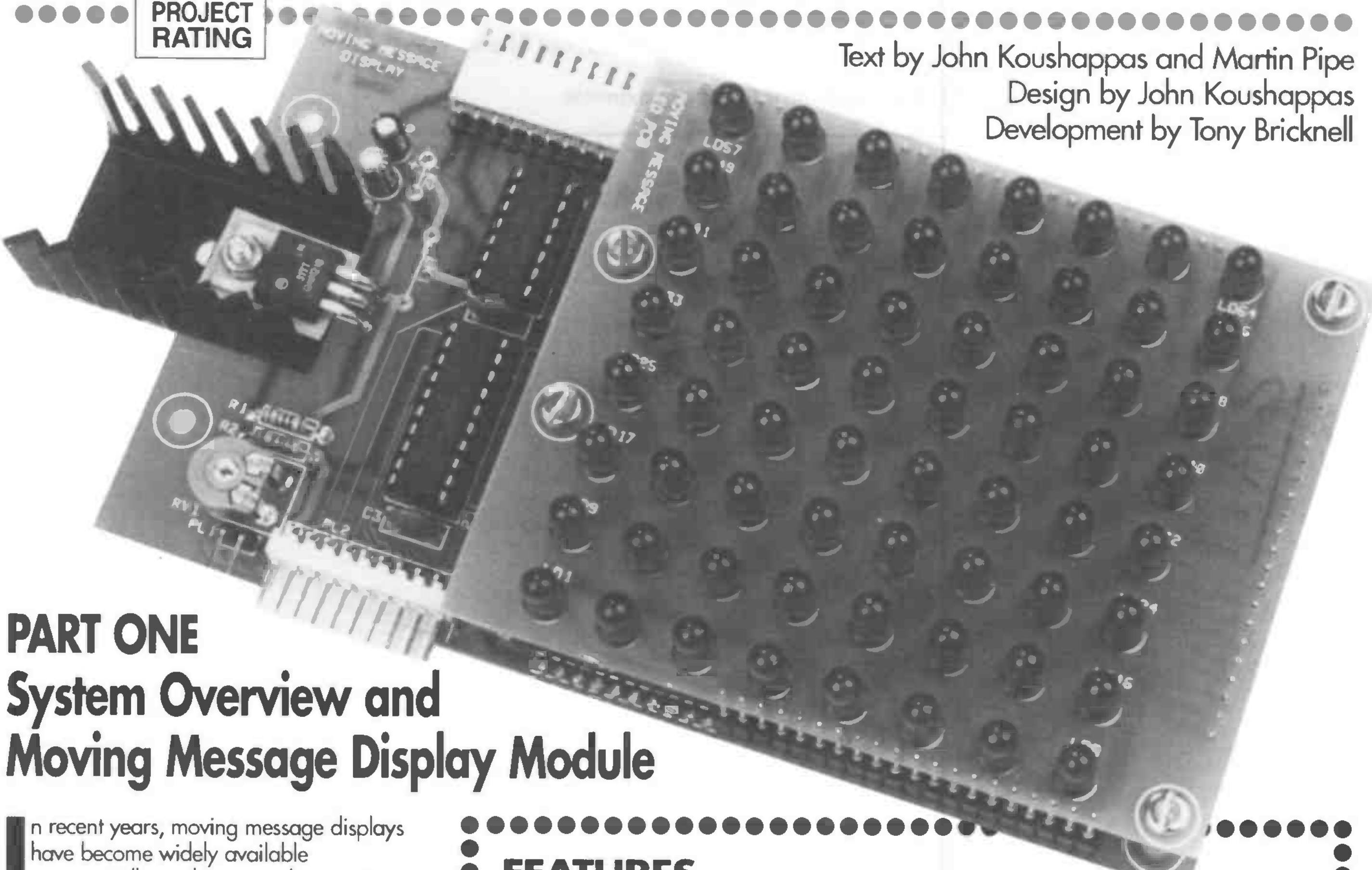
Self-Tapping Screw No.6 × ⅜in.	12
Washer 4BA	12
PP3 Clip	1
Tinned Copper Wire 0·71mm	30cm
Black Wire 7/0·2mm	30cm
Red Wire 7/0·2mm	30cm
Black 2mm Sleeving	30cm
Red 2mm Sleeving	30cm
Base Board	1
Instruction Leaflet and Layout Sheet	1

*Please Note: length is approximate

MOVING MESSAGE DISPLAY SYSTEM

3 PROJECT RATING

Text by John Koushappas and Martin Pipe
Design by John Koushappas
Development by Tony Bricknell



PART ONE
System Overview and Moving Message Display Module

In recent years, moving message displays have become widely available commercially, and can now be seen in many shop windows, post offices, railway stations, airports and even on television. The basis of these systems is often a fixed-length light emitting diode (LED) display controlled by a microcomputer system, the messages being entered on a miniature keyboard and stored in non-volatile memory.

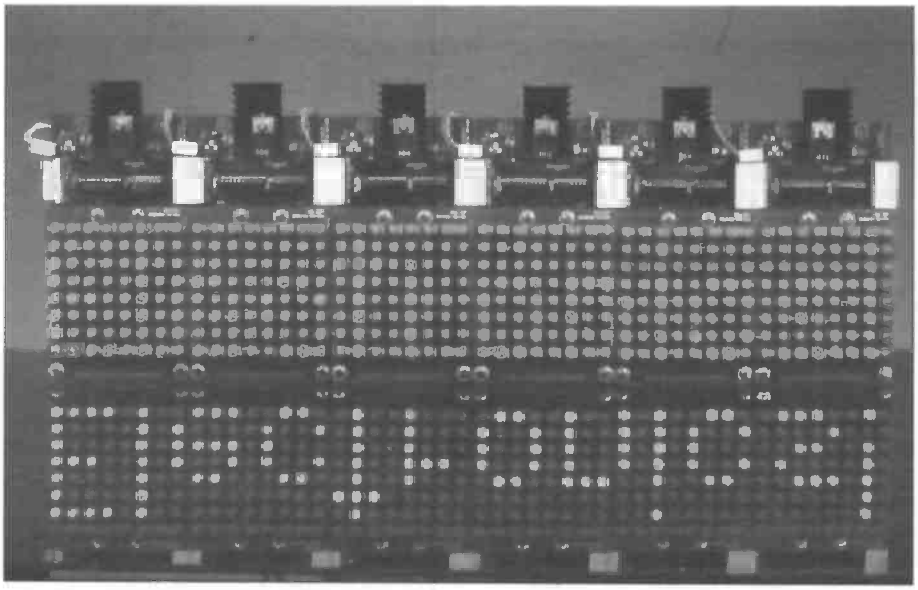
There are three basic addressing techniques that can be used to output messages to the LED displays. These are column scanning addressing, row scanning addressing and direct access addressing. Commercial message displays often employ row scanning addressing, whereby the relevant LEDs of only one row are on at any particular time, the data for the row being stored in a long shift register. As the display is scanned from top to bottom (or from bottom to top), there is a time lag between the rows and this results in the characteristic 'slant' in the letters of the message as it

FEATURES

★ Designed for use with any computer equipped with three 8-bit I/O ports – e.g., an IBM PC or compatible equipped with the Maplin 24-line PI/O card ★ Easily programmable from BASIC ★ Expandable to 32 boards by 'daisy-chaining' modules together ★ Large viewing area makes display highly readable in all lighting conditions ★ Programmable scrolling in all directions ★ Facilities for fade up/down ★ Programmable 'fizzle' effects ★ Direct pixel addressing for Speed (Animations, etc.) ★ Easy to Build

APPLICATIONS

★ Shop Displays ★ Announcements in Public Areas ★ Attention Grabbing! ★ Special Effects



A bank of display modules.

travels from right to left (or vice versa) across the LED display. Column scanning works on the same principles, but manipulates one column at a time instead.

This moving message display system uses a direct access architecture, and is fully controlled by a host computer. As a result, a wide variety of display techniques can be employed, all programmed in BASIC by the user. The system can, with appropriate programming, perform any of the functions seen on commercial displays.

A unique feature of this moving message display system is that it has been designed to be expandable. The unit module of this system is the Moving Message Display Module. The system is expandable from one module up to a maximum of 32 display modules, giving a 2·6m long high-resolution display, which is comparable with the largest commercial displays. All the necessary electronics to drive the LED display are also contained on the same board. As a result, the modules are easy to build.

0	0	1	0	0	0
0	1	0	1	0	0
1	0	0	0	1	0
1	0	0	0	1	0
1	1	1	1	1	0
1	0	0	0	1	0
1	0	0	0	1	0

Figure 1a. Binary representation of the character 'A' in matrix form.

The Series

This four-part series will describe the design and construction of an expandable moving message display system. In this, the first part, we look at the display board and the expandable nature of the system.

In the second part, we will look at how the single display module can be interfaced with a computer. Also discussed will be the controller board, which is used to supervise each pair of display modules in the completed Moving Message Display, and the display configuration options available.

In the third part, the expandable PSU – which powers the complete display system – will be covered. In the fourth and final part, we will discuss how the overall system is put together, along with interfacing, testing and programming.

Computer Requirements

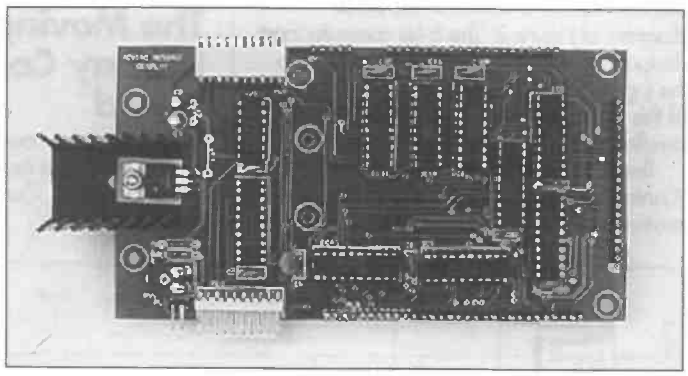
As the system is computer controlled, you obviously must have a computer to connect the hardware to! The computer can be any type provided it meets certain hardware and software specifications. The computer must have two 8-bit plus one 2-bit, parallel latching output ports. For example, the Intel 8155 and 8255 PPI (Programmable Peripheral Interface) are suitable. The 8255 PPI, out of interest, is used in the Maplin 24-line PI/O Card. Note that an IBM PC's printer port is not suitable.

From the software point of view, this project requires a BASIC interpreter (or compiler). For the expandable display, the BASIC interpreter must be capable of directly calling machine code routines. This will involve some advanced programming. This project will focus on the GWBASIC Interpreter running on an IBM PC compatible computer, but other types of BASIC can be used.

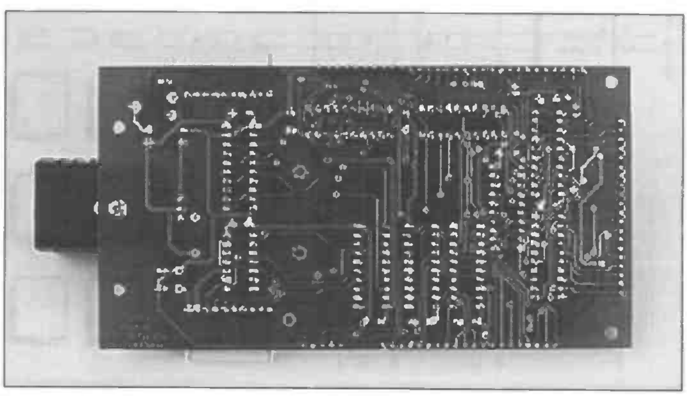
For interconnection of the display to the computer, knowledge of how to correctly connect to the parallel ports will be required. Examples will be given for connection to the Maplin 24-line PI/O card.

Bit Value	COLUMN 1	2	3	4	5	6
1			1			
2		1		1		
4	1				1	
8	1				1	
16	1	1	1	1	1	
32	1				1	
64	1				1	
Total	124	18	17	18	124	0

Figure 1b. Calculating bit values for the character matrix.



Component-side view of the control logic PCB.



Track-side view of the control logic PCB.

Moving Messages

Messages are made up by a computer program which converts the letters in the words of your messages into an array of '1's and '0's. This arrangement is called a *bit pattern*. For all the alphanumeric characters you want to use, you can design a corresponding bit pattern into the program. Figure 1(a) shows how the letter 'A' is represented.

When arranged to fit into the grid of Figure 1(b), it is possible, for any character, to calculate the totals for each of the columns 1 to 6. The numbers across the top of the grid show which column you are calculating for and the numbers down the left show the decimal value for each bit in the character. Adding up the decimal numbers corresponding to a '1' in each column will give a total for each column representing that part of the character. The set of totals for all six columns then describe the whole character's bit pattern. Thus the bit pattern for 'A' in our example is 124,18,17,18,124,0.

The method the computer program uses to choose these numbers is called the look-up table method. A *look-up table* is a *predefined* table of values corresponding to each letter or character. This look-up table may be embedded in the computer program itself, or may be loaded by the program as a data file.

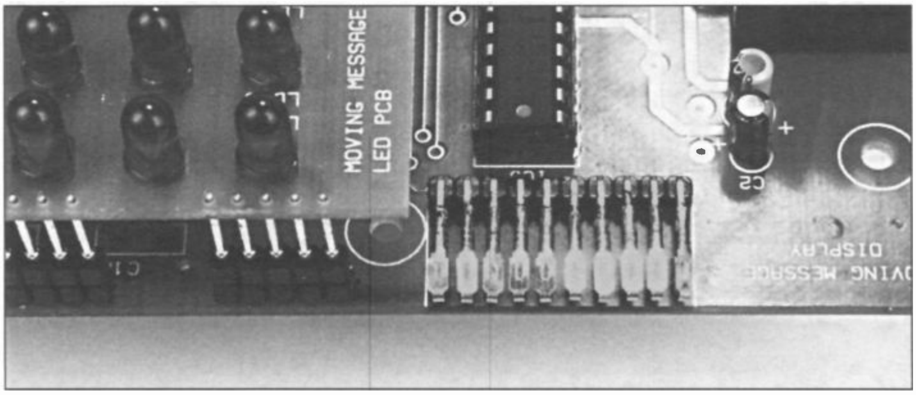
The joining together of several bit patterns corresponding to words and sentences may be executed by the program in one operation. Thus the program which converts the text you type into it into one long bit pattern is called a *Bit Pattern Compiler*.

The compiled bit pattern is then held in a continuous piece of memory which stores the whole message ready for the displaying program to use.

The Moving Message Display Module

The architecture of the Moving Message Display Module is shown in the block diagram of Figure 2. The 8-bit computer port designated to be the 'data port' connects to the inputs of the data bus buffer. The outputs of this buffer feed eight octal data latches simultaneously from the module's data bus.

The 8-bit port designated to be the 'Control Port' generates the eight column control lines, Co1 to Co8. When octal data



Close-up showing Minicon terminals fitted to the right angle pin strip.

latch number 1 is selected by the Co1 line, that latch can read the data on the data bus. When that column is deselected, the octal data latch number 1 stores or latches that data.

If a different piece of data is placed on the data bus by the computer via the port and the buffer, this does not affect data latch number 1, which is currently storing the previous data sent on the bus. Data latch number 2 can now be used to read the new data on the data bus. This is repeated for all the latches in the module.

With all eight octal data latches now holding data, the data can now be output whenever it is required. This is achieved by enabling the tri-state buffers from the Display Control buffer line. This is a special control line which is controlled from another of the computer's ports. When it is low, all the buffers are disabled and the display is turned off. When this control line is taken high, all the buffers are turned on and the LEDs are turned on corresponding to the data in each of the latches.

Physically, to save space the Moving Message Display Module is built up on two PCBs – one with the control logic, and another with the 64 LEDs on it.

The Moving Message Display Controller Board

When more than one moving message display module is required in your display system, the 8-bit 'Control Port' has to be decoded.

The purpose of the controller circuit in this system is to generate individual control output (Co) lines for each column of LEDs that are selected. A parallel access architecture was adopted so that any column of LEDs could be accessed with one computer program instruction. This parallel architecture is evident from the board's block diagram, shown in Figure 3.

There are two stages of decoding. The first decoder decodes the upper nibble (i.e. bits 4 to 7) of the control port into 16 individual lines. Each one of these lines can then select a second stage (4 to 16 line decoder), which decodes the lower nibble of the control port. Out of each second stage decoder emerge 16 control lines, each of which selects a column of LEDs on the moving message display module.

The architecture of the controller thus simply breaks down the 8-bit control port into a maximum of 256 individual control lines. Of course, if only two display modules are used, then only the upper nibble decoder chip and one lower nibble decoder chip are required. Therefore the controller system is sub-divided into modules proportional to the number of display modules required. There are two decoders on the first controller board in an expandable system, and only one decoder on each subsequent controller board. Each controller board thus supplies control lines for two display modules.

How the Two Modules Link Together

The data bus buffer and display control buffer outputs of the Moving Message Display Module are sent out of the module in the same parallel fashion as they entered. These outputs can therefore pass the same data to another identical module connected to the first module. This type of expansion is called *daisy-chaining*.

Similarly, the idea of daisy-chaining has been applied to the controller board. The result is a moving message display system which is completely expandable, from one display module up to a maximum of 32 display modules. The minimum system is a single module with no controller board. The minimum expandable system, or 'Base System', is two display modules and one controller board. Expansion continues at a rate of two display modules to one controller board up to a maximum of 32 display modules and 16 controller boards. The expandable structure of how the modules link together is shown in Figure 4.

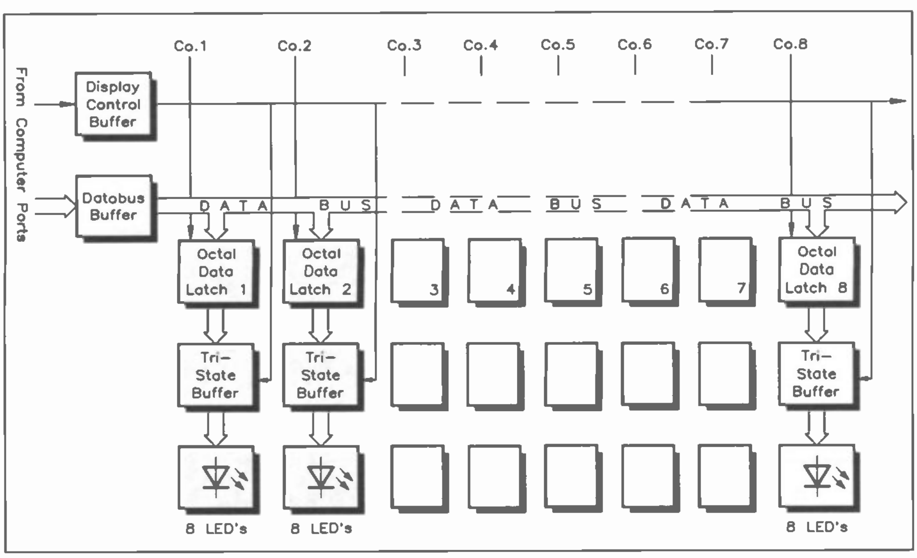


Figure 2. Display module block diagram.

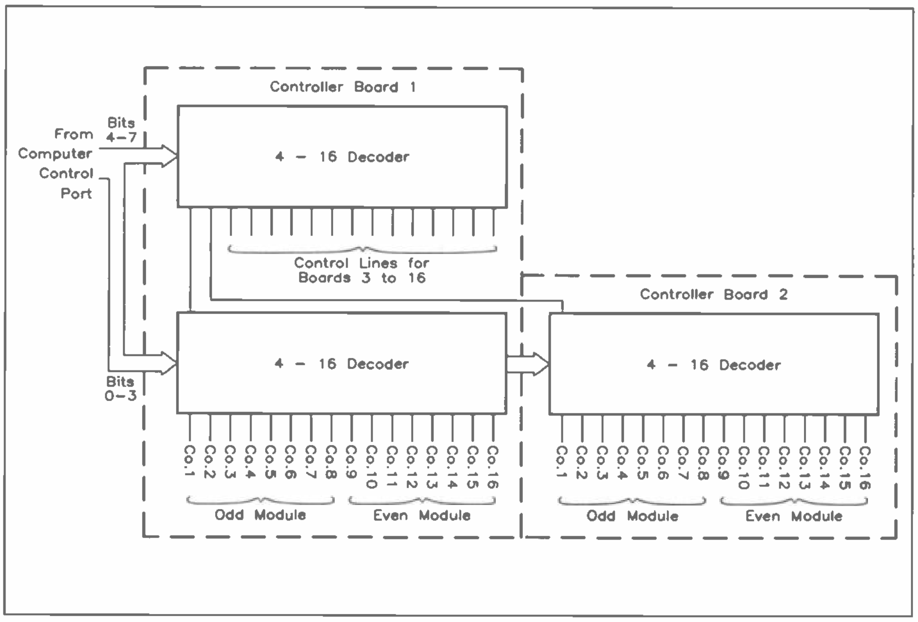


Figure 3. Controller board block diagram.

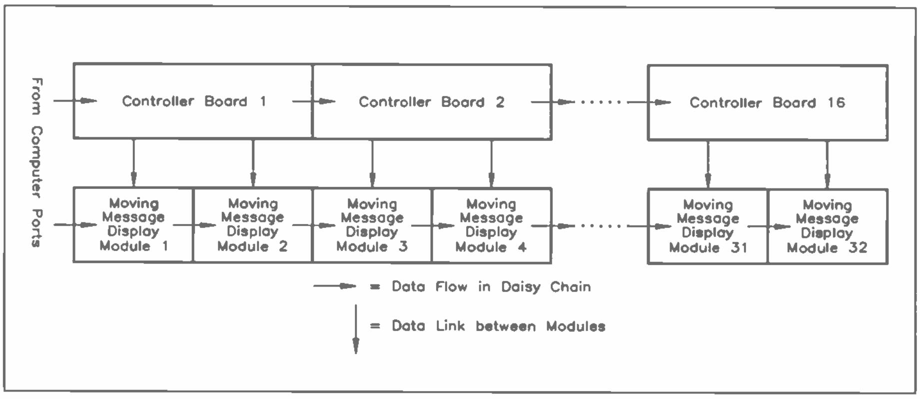


Figure 4. Moving Message Display System overall block diagram.

Moving Message Display Module – Circuit Description

As can be seen from Figure 5, the circuit diagram, an 74LS244 octal non-inverting buffer, IC2, is included on each Moving Message Display Module. This serves two functions. It isolates the eight octal latches of the display from the driving output port which supplies the data, via PL2. It drives, or fans out to, the eight octal latches, thus maintaining TTL data integrity. It also drives a ninth device, namely the data bus buffer on the next moving message display module, in the complete system.

The OE (active high enable) display control buffer carries out the same functions as the data bus buffer in that it isolates the display from the output port of the host computer whilst driving the eight octal latches. It also performs another function in that it performs two inversions. Firstly, it generates an $\overline{OE}$ line from the active high one (present on the PL2 input) for IC4 to IC11. Buffering is also provided by the gate – remember that there will be nine devices driven from it. The $\overline{OE}$ line is then inverted again to give an output of OE to leave the module as it entered.

The heart of the circuit is the 74LS373, which is an octal transparent data latch. Eight of these devices are used (ICs 4 to 11) – one for each column of LEDs. Each 74LS373 has eight data inputs (D0 to D7) and eight outputs (Q0 to Q7), each of which has an LED connected to it.

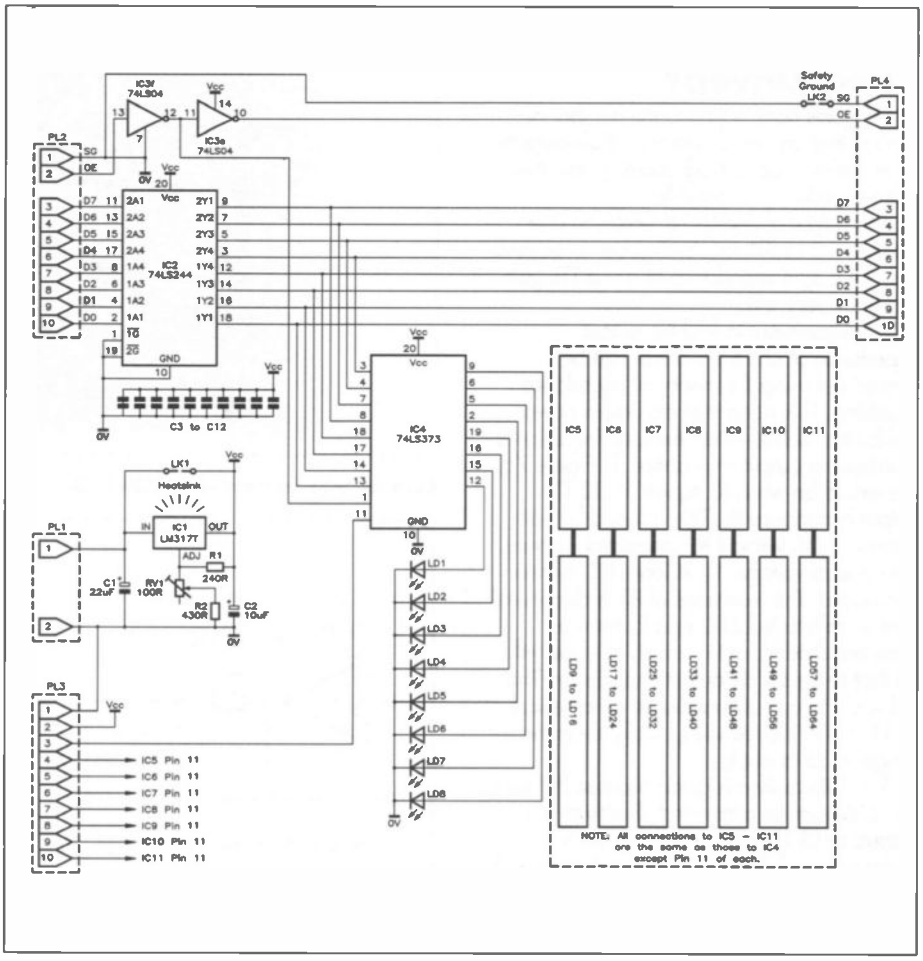


Figure 5. Display module circuit diagram.

Any data present on the inputs (D0 to D7) will pass directly to the outputs when the LE (latch enable) pin is high, as shown in the timing diagram of Figure 6. (A cross-over in the data input and data output timing diagram indicates a general transition for each of the eight bits denoted by the word 'data'). When LE is taken low, the data is latched on the outputs (Stage 'C' of Figure 6). Any subsequent change in the data while LE is low is not registered on the outputs because the latch is not in transparent mode (Stages 'A' and 'C' of Figure 6). The data thus latched can be output to the LEDs which form the display. The LEDs will show this data when the $\overline{OE}$ pin is held low.

The LEDs' anodes connect directly to the Q outputs and all the cathodes are commoned to ground. There are no current limiting resistors used in this design – the reasons for this will become evident later. As a result, quite a high current is made available to each LED, which can give the display a high level of brightness, even with standard LEDs. The LED can theoretically draw a current of up to 58mA, although in practice the current is around 16mA. However, this does not adversely harm the TTL driver; in practice the 74LS series of ICs with buffered outputs will cope with the heavy load placed on them by direct LED coupling indefinitely. The chip body may run warm but it will not be affected or damaged. This method of driving the LEDs is explained later.

There are eight of these circuits, based around IC4 to 11, for each column of LEDs. The first column, controlled by IC4, consists of LD1 to 8 inclusive; the second, controlled by IC5, consists of LD17 to 24 inclusive, and so on.

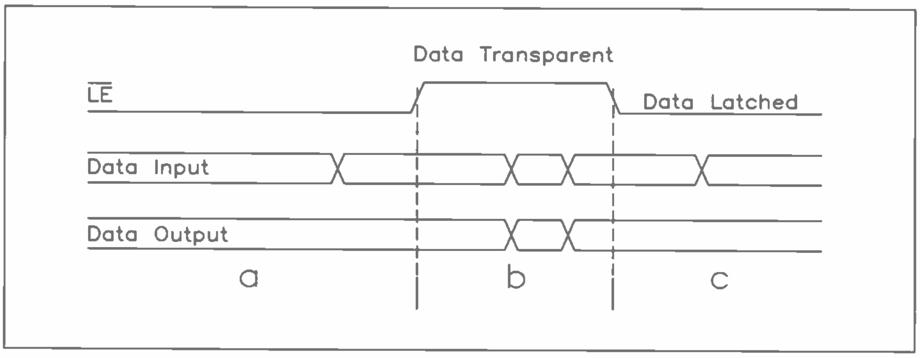


Figure 6. Timing diagram.

The column control lines enter the module via connector PL3. Each line connects to the 74LS373 Latch Enable (bar) input. For the single module, the source of the control lines is directly from the second parallel output port from the computer. The circuit detail shown boxed around IC4 and LD1 to 8 is repeated for ICs 5 to 11 and LEDs 9 to 64.

The moving message display module has its own voltage regulator on board, which is based around the standard variable voltage regulator, the LM317T. Employing a variable voltage regulator reduces the overall current consumption to approximately 1A, by reducing the TTL supply voltage. It also yields two useful consequences, in that it allows a simple yet powerful unregulated power supply to be the external power source for the display module (this will be covered in the final part of the series). In addition, the intensity of the LED display can be preset and fully controlled by hardware, as a result of having control of the output voltage of the regulator, as well as by software. Capacitors C3 to 12 (0·1µF) decouple the power supply for IC2 to 11.

Why are the LEDs Direct-Driven?

LS TTL gates have a limit to the number of devices that can be driven from their outputs since, unlike their CMOS counterparts, their input impedance is rather low.

The output of one LS TTL gate can drive typically 10 LS TTL inputs; in order to achieve this, the output impedance of an LS TTL gate has to be very low.

An experiment to find the output impedance of an LS TTL driver can be carried out using the theory of impedance matching. This states that maximum power will be transferred when the load impedance matches the output impedance. In Figure 7a, this would be when R_L equals the LS TTL output impedance R_o. This is found by first measuring V_o without R_L connected – giving the no load voltage, V_t. A load, R1, is then introduced. The resistance of R1 is decreased until V_o falls to $V_t/2$ (R_L and R_o form a potential divider); at this point, its value will be that of the LS TTL output impedance. The value of the output impedance for a typical LS TTL buffer device will typically be in the range of 35 to 40Ω.

This information is given because it shows that LEDs can be connected directly to the outputs of LS TTL gates without resistors being required. Figure 7b shows that an LED will draw approximately 58mA with full supply voltage applied to the TTL gate. A LS TTL output will thus regulate the current drawn by an LED connected directly to it.

It should be noted that the theory given above has been deliberately simplified to illustrate the point made. It is not a fully accurate representation of the actual output impedance; there are many other factors which have a bearing on the output impedance (e.g., R_o increases as V_s decreases, and power transfer is non-linear when R_L is not equal to R_o – as will be the case with an LED used in place of R_L).

The discussion of output impedance above is a general theory used in the field of amplifiers which can be used equally well in this case. However, the output impedance calculations/measurements discussed here do not relate directly to calculations of TTL fan in/out. For these, V_o must be maintained at, or above, the minimum voltage considered to represent logic 1 which is, typically, 2·4V. Buffers can drive devices down to a load of 133Ω.

Moving Message Display Module Construction

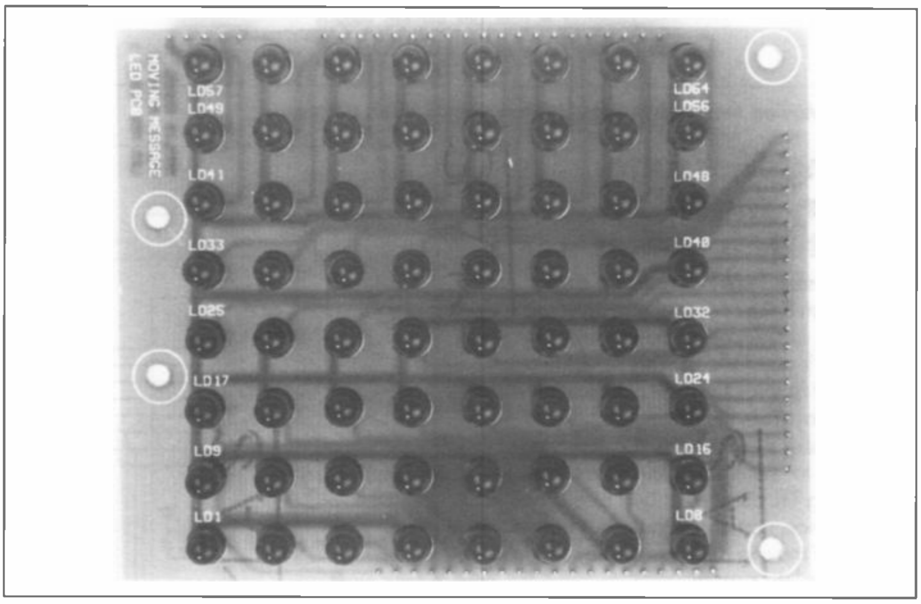
The Display Module consists of two separate boards; one that contains the LEDs, and another that contains the controlling logic. The PCB legends and track layouts of each PCB are shown in Figures 8a and 8b. Construction is fairly straightforward if the following information, along with that provided in the Constructors' Guide (supplied with the kit), is followed.

LED PCB

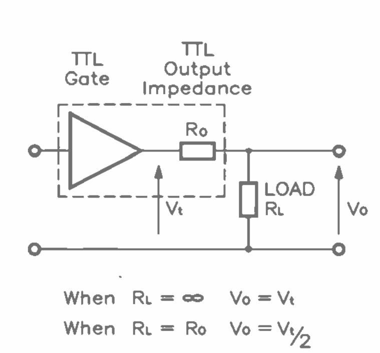
Fit each LED so that it is flush against the PCB, as shown in Figure 9. The flat side of the package should be lined up with the corresponding PCB legend. Each LED should be pressed against the PCB just prior to soldering it into position; this ensures flush mounting (and hence a tidy appearance) without having to reposition the device – unnecessarily reheating the joint may damage the LED and board.

Control Logic PCB

Begin construction with the resistors and the preset, followed by the capacitors and IC sockets. Attach the heatsink, regulator and insulator to the PCB using the M3 nut, shakeproof washer and 10mm bolt, as shown in Figure 10. Cut the supplied 12-way right angle pin strip into separate 2-way and



Component-side view of the LED PCB.



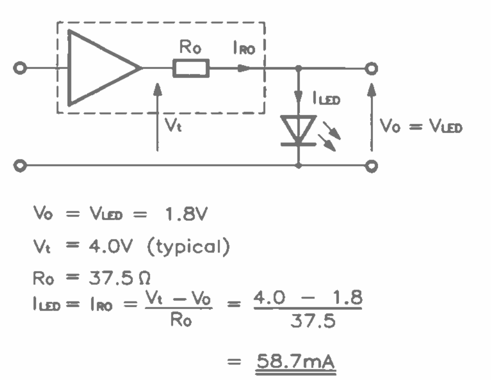


Figure 7a. TTL output impedance calculations. **Figure 7b. LED drive current calculations.**

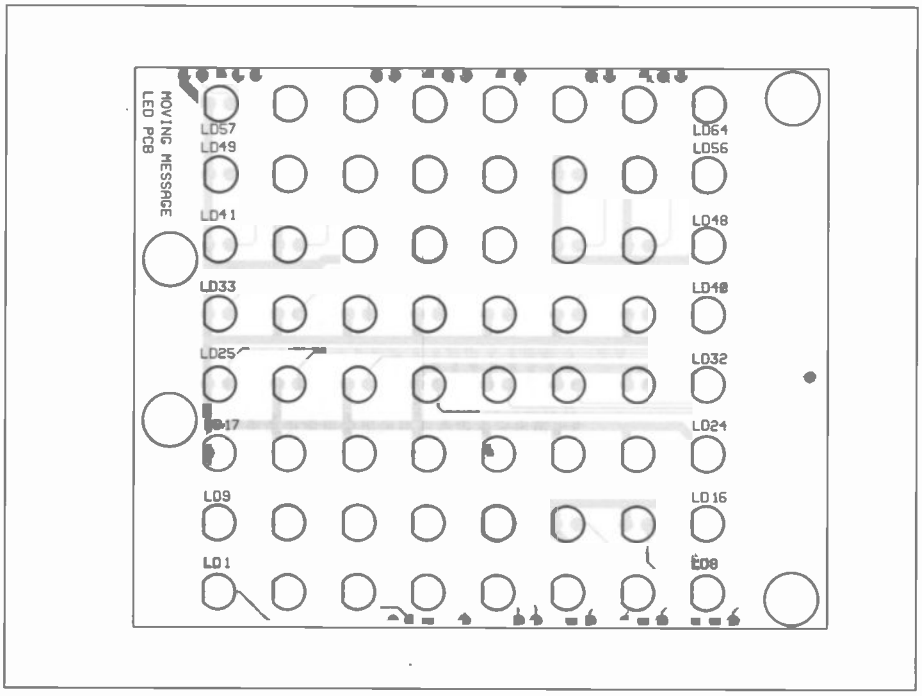


Figure 8a. LED PCB legend.

unwanted 31 sections! 20-way lengths should be cut off each of the 32-way strips, and mated with two of the 20-way socket strips (the thinner pins mate with the socket).

Two 12-way sections should be left; one of these should have a strip of four removed to leave a 8-way length. This, together with the remaining 12-way section, should be mated with the third 20-way socket strip.

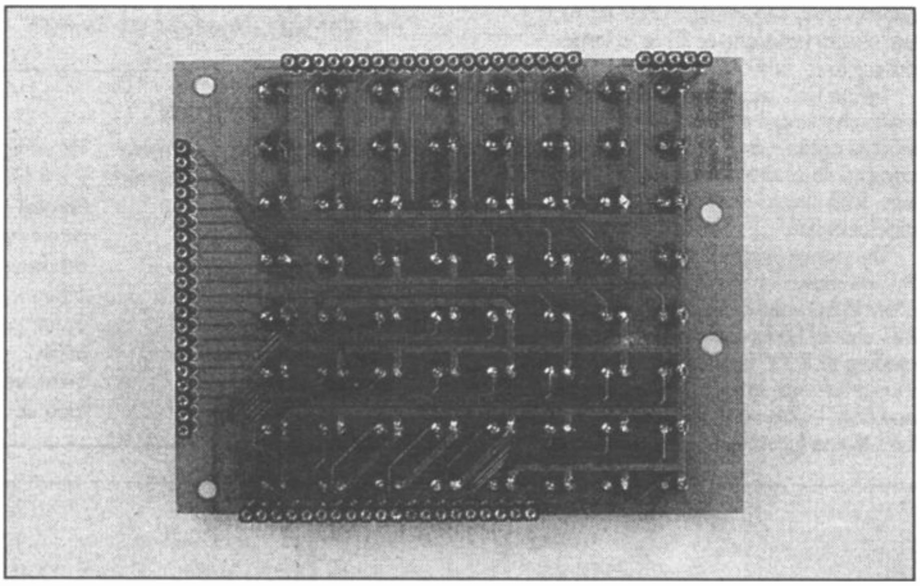
The remaining pin, together with the 4-way section prepared earlier, should be mated with the 5-way socket. Again, if you are purchasing the components separately, you will have to buy another 20-way socket strip, cutting off the unwanted 15 sections.

Assemble the PCBs and connectors as shown in Figure 10 (the pin end of the connectors are fitted to the main PCB); use the M3 spacers, screws, nuts and washers to temporarily hold the whole assembly together. Solder the pin connectors to the main PCB, and partly solder the socket connectors to the LED PCB. Note that you cannot fully solder these connectors as only the outer side of the joint is accessible – the important thing is to make sure that the sockets are at the correct

10-way strips (note that if you are buying the components separately, the smallest size available is 36-way (JW60Q)). Solder the Minicon terminals to the 10-way right-angle pin strip, as shown in Figure 11. Note that the end of the pin should line up with the second tag, so that the 10-way Minicon housing will fit into place properly.

Figure 12 shows the PCB positions of all connectors. Note from this diagram that PL3, the 10-way straight pin strip (square cross-section) is fitted from the *solder* side of the PCB.

All other connectors, including the 10-way right-angle pin strip with Minicon housing prepared earlier, are fitted on the component side of the board. Supplied in the kit are two 32-way, and one single-way, pin strips (these are characterised by the round cross-section of the pins). Single-way pin strips are not available separately; if you are buying components separately you will have to buy another 32-way, discarding the



Track-side view of the LED PCB.

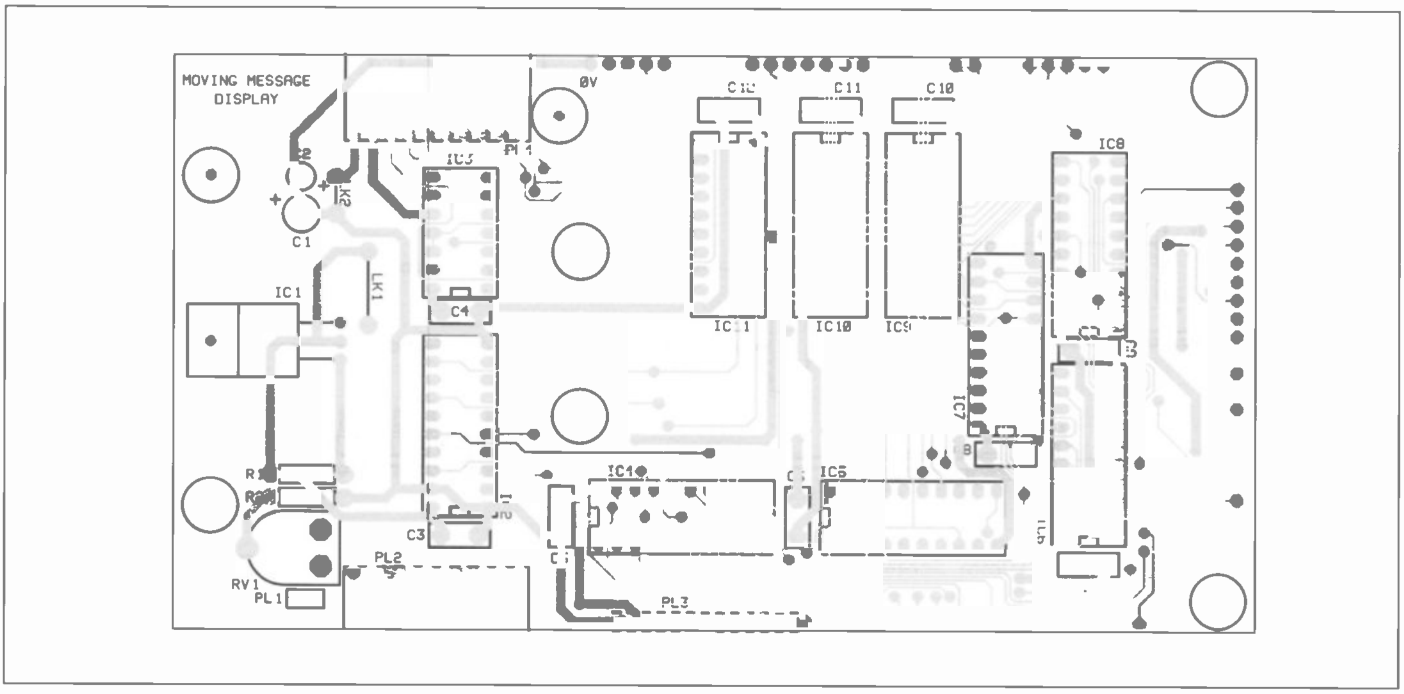


Figure 8b. Driver PCB legend. Note: DO NOT fit LK1 or LK2.

height when the time comes to solder them properly.

Next, separate the two PCBs and complete the soldering of the connectors on the LED PCB. Insert the ICs into the appropriate sockets. After a thorough inspection for misplaced components and soldering problems, the two PCBs can now be permanently reassembled together.

Circuit Link Options

There are two optional links, LK1 and LK2. LK1 is a Module Supply bypass, while LK2 is concerned with maintaining a signal ground between two moving message display boards in an expandable system. The links will be covered in subsequent articles; at this stage of the project, these links are not used and MUST NOT be fitted.

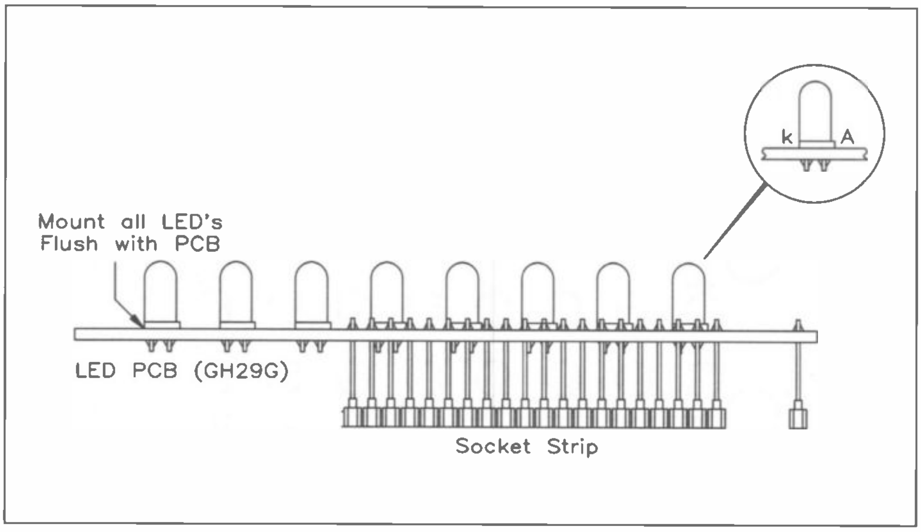


Figure 9. LED PCB assembly.

Moving Message Display Module Testing

To check the Moving Message Display Module, a 9V power supply capable of sourcing at least 1A is required. An appropriately set-up bench PSU is ideal; if you do not have one of these, a large battery (e.g., PP9) will suffice.

Before tests can begin, it is probably worth checking the assembly of the complete module again – any mistakes rectified at this stage could save money, time and frustration later. RV1 should now be turned fully anticlockwise.

The power supply should be applied to PL1, as shown in Figure 13. After connecting a DMM between the output and ground, RV1 should be turned clockwise until a reading of 3·9V is obtained. All of the LEDs should light up, since the output enable (OE) and data inputs normally float 'high'. When the OE line (present on pin 2 of PL2, as shown in Figure 13) is grounded (note that pin 1 of PL2 is a ground connection), the LEDs should go out.

If the LEDs do not all come on when power is applied, do not panic, as the inputs may be floating low.

The other eight pins of PL2 are the eight data inputs (D0 to D7). When OE is high (i.e. all the LEDs are lit) and one (or more) of these data inputs are grounded, then the corresponding rows should be blanked out. If one or more fail to respond to this treatment, check your work again for solder bridges, misplaced components or dry joints.

Specification

Power Supply Requirement:	9V unregulated, 1A maximum
Computer Interface Required:	2 x 8-bit and 1 x 2-bit parallel latching output ports
Data Feed:	Parallel Load
Hardware Control:	Direct Addressing, Hardware Blanking
Display type:	64 5mm Red LEDs in an 8 x 8 matrix with 10mm spacing
PCB Dimensions:	5·8in. x 3·2in.
Viewing area:	3·2in. x 3·2in.
Test Programs (Part 2):	BASIC
Message Effects:	Scroll left, scroll right, scroll up, scroll down, fade up, fade down, flash, animations.

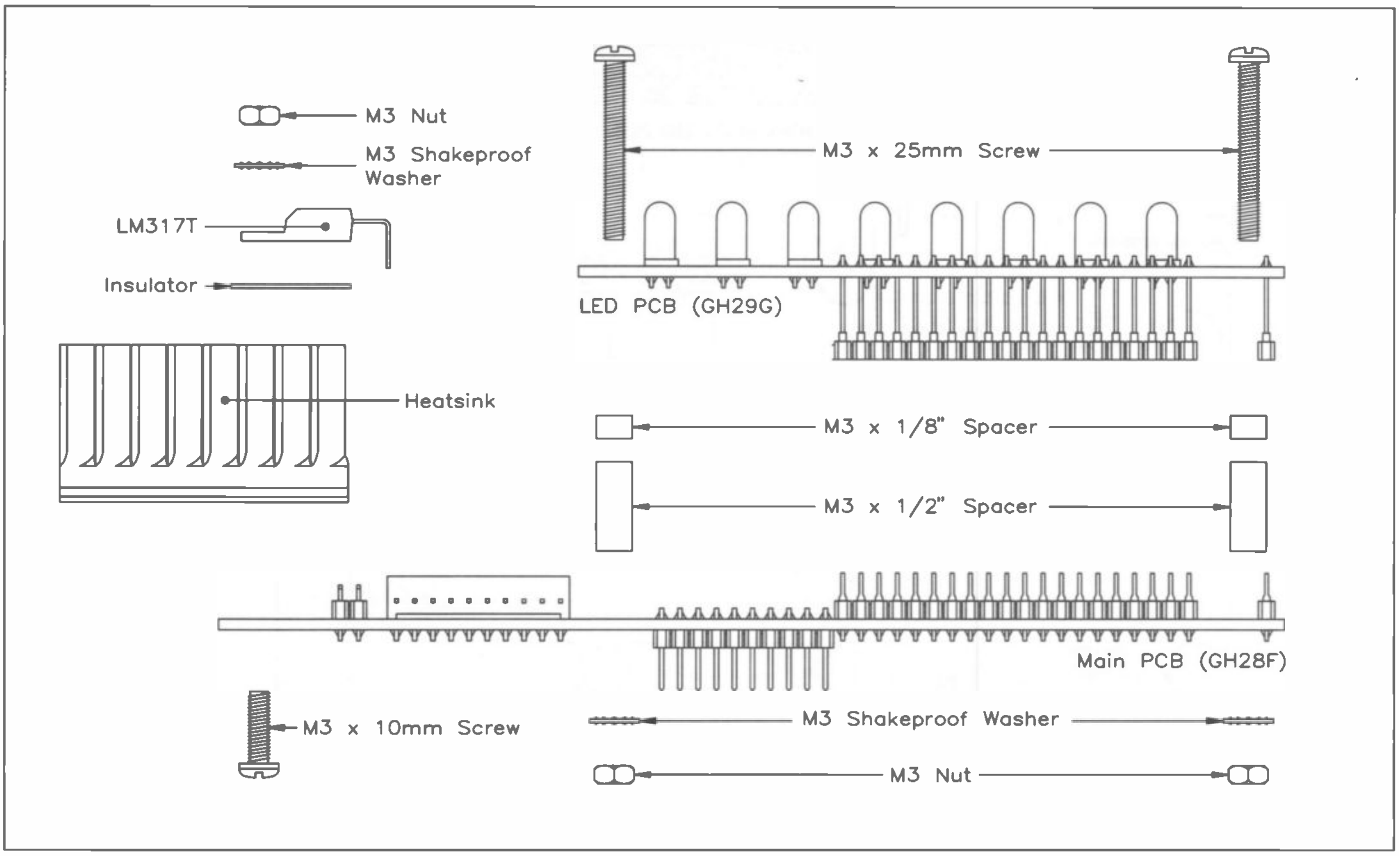


Figure 10. Fitting voltage regulator and final assembly.

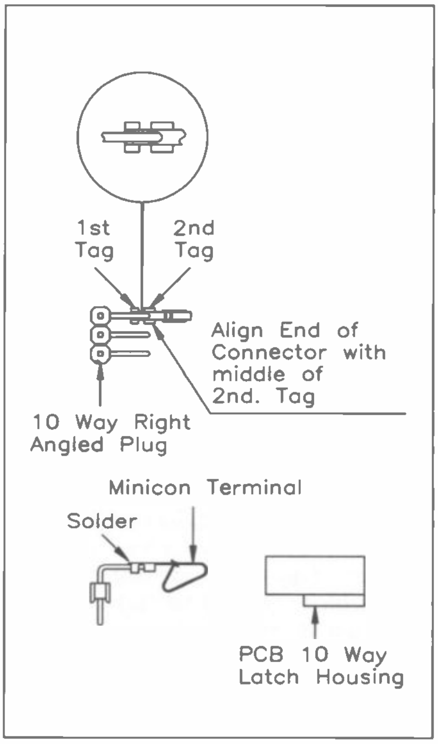


Figure 11. Minicon assembly.

Acknowledgments

Acknowledgments are due to the following, for the considerable help that they have contributed to this project: Mr. Ken Bone, Senior PCB Technician, for making seven excellent prototypes. Mr. Michael Robinson, for building up some of the prototypes. Middlesex University's Electronic and Mechanical Engineering laboratories and technicians, for providing excellent facilities and assistance. Professor George Goldspink, Head of School of Electronics, for giving his approval to the go-ahead of this project.

Next Month

In Part 2, we take a look at the interfacing, and software testing, of the Moving Message Display Module, and an in-depth look at the Controller Board.

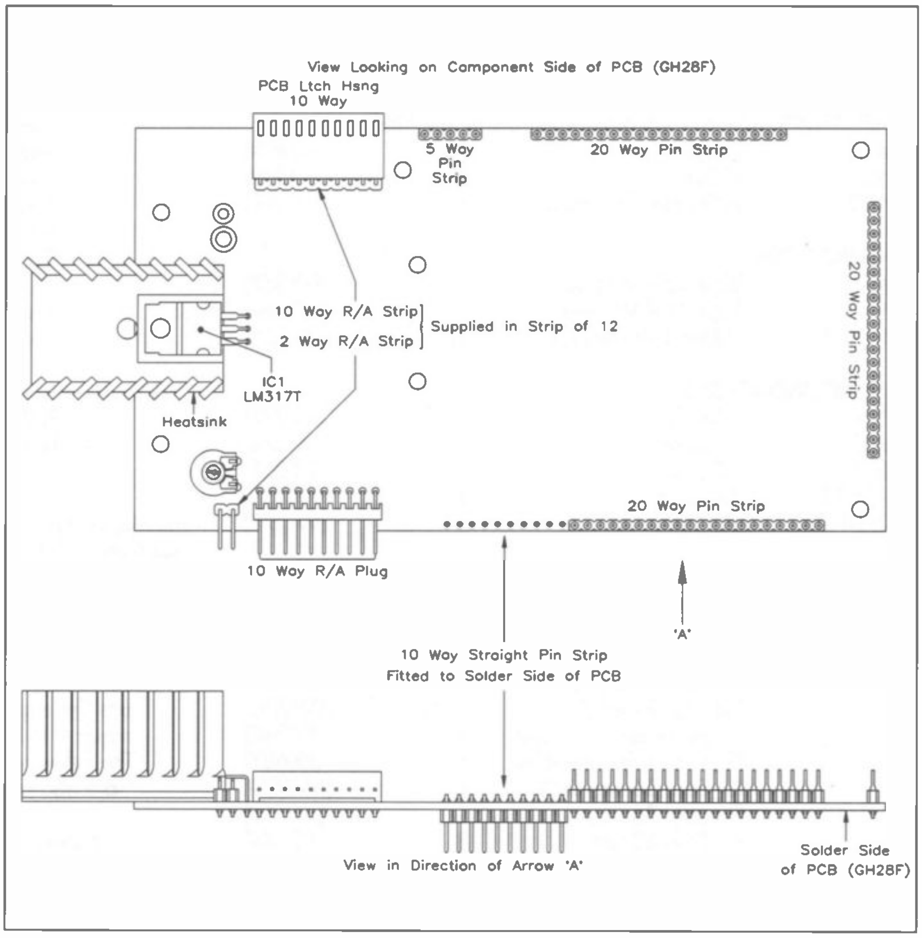


Figure 12. Fitting connectors to control logic PCB.

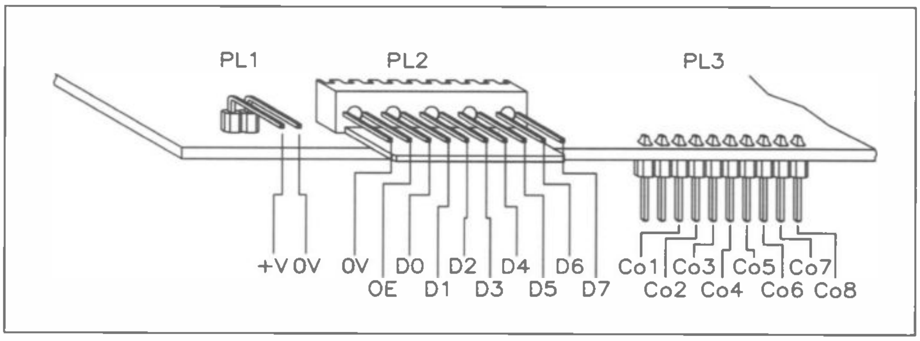


Figure 13. Connections for test purposes.

Glossary

Column scanning	Illuminate one column at a time continuously from left to right.
Direct access	Access any column at any time, in any order. Not necessary to continuously scan the display.
Row scanning	Illuminate a whole row at a time continuously from top to bottom.
Bit Pattern	Arrangement of '1's and '0's which defines a complete character or picture.
Bit Pattern Compiler	Program which converts text into bit patterns.
Daisy-chaining	Identical units joined together to make one large unit.
Data	One byte of a bit pattern.
Decoder (4 to 16 line)	Takes 4-bit binary input and outputs to only one line out of the 16 available output lines.
Fizzle	Message builds up by lights turning on in a random order.
Look-up table	Predefined table of bit patterns for each of the characters used. A look-up table is usually structured such that the data it contains can be easily looked up, for example, by a mathematical formula or relationship.
Non-volatile memory	Memory which does not lose its contents when the power supply to it is removed.
Nibble	Group of 4 bits (half a byte)
PPI	Parallel Peripheral Interface. A special IC which allows the computer to connect with an external piece of equipment or circuit.
8255 PPI	A programmable IC containing three 8-bit parallel input/output ports; when set up as output ports, they are latching; when set up as input ports, they are non-latching. Used in the Maplin 24-line PIO (Programmable Input/Output card) for the IBM series of PCs and their clones.
8155 RIOT	RAM Input/Output Timer. Similar to above but contains two 8-bit, and one 6-bit parallel input/output ports. It also contains some RAM and a timer. More common on older 8-bit Intel-based systems.

MOVING MESSAGE DISPLAY MODULE PARTS LIST

RESISTORS: All 0·6W 1% Metal Film (Unless specified)

R1	240Ω	1	(M240R)
R2	430Ω	1	(M430R)
RV2	100Ω Hor End Preset	1	(UF97F)

CAPACITORS

C1	22µF 16V Minelect	1	(YY36P)
C2	10µF 16V Minelect	1	(YY34M)
C3-12	100nF 16V Minidisc	10	(YR75S)

SEMICONDUCTORS

IC1	LM317T	1	(UF27E)
IC2	74LS244	1	(QQ56L)
IC3	74LS04	1	(YF04E)
IC4-11	74LS373	8	(YH15R)
LED1-64	LED Red 5mm	64	(WL27E)

MISCELLANEOUS

	DIL Socket 14-pin	1	(BL18U)
	DIL Socket 20-pin	9	(HQ77J)
	Powerfin Heatsink	1	(FG55K)
	Insulator TO220	1	(QY45Y)
	Skt. Hsg Terminal	1 Pkt	(YW25C)
	Socket Housing 10-way	1	(FY94C)
	Plug Assm 10-way R/A	1	(RK68Y)
	Pin Strip 36-way R/A	1	(JW60Q)
	Pin Strip 36-way	1	(JW59P)
	Pin Strip 32-way	3	(JR74R)
	Socket Strip 20-way	4	(KP51F)
	Spacer M3 x ½in.	1 Pkt	(FG34M)
	Spacer M3 x ⅛in.	1 Pkt	(FG32K)
	Steel Nut M3	1 Pkt	(JD61R)
	Shakeproof Washer M3	1 Pkt	(BF44X)
	Steel Screw M3 x 10mm	1 Pkt	(JY22Y)
	Screw Screw M3 x 25mm	1 Pkt	(JY26D)
	Control Logic PCB	1	(GH28F)
	LED PCB	1	(GH29G)
	Instruction Leaflet	1	(XU05F)
	Constructors' Guide	1	(XH79L)

OPTIONAL (Not in Kit)

	PP9 9V Battery	1	(FM05F)
	PP9 Battery Clips	1 Set	(HF27E)

The Maplin 'Get-You-Working' Service is available for this project, see Constructors' Guide or current Maplin Catalogue for details.

The above items (excluding Optional) are available as a kit, which offers a saving over buying the parts separately.

Order As LT21X (Moving Message Display Module)

Please Note: Where 'package' quantities are stated in the Parts List (e.g., packet, strip, reel, etc.), the exact quantity required to build the project will be supplied in the kit.

The following new items (which are included in the kit) are also available separately, but are not shown in the current Maplin Catalogue.

Move Mess C/Logic PCB **Order As GH28F.**

Move Mess LED PCB **Order As GH29G.**

CAR PARKING RADAR SYSTEM

Text by John Mosely

FEATURES

* Easy to install * Adjustable range
* Audible warning of obstacles

Assist your parking skills (or your partners!), by adding this short-range ultrasonic RADAR system to your vehicle. It will detect objects and 'measure' their distance by means of sensors mounted on the back of your car. The unit will allow you to accurately judge the amount of clear space behind your vehicle and detect possible unseen hazards within the radar range. The unit emits an audible 'bleep' when your car has reached a preset minimum distance from the obstacle.

Circuit Description

The circuit diagram, Figure 1, illustrates circuit operations for the transmitter and the way in which the receiver is used to produce a warning signal. Reflected ultrasonic pulses require a significant amount of signal processing before they can be used to activate the preset minimum distance alarm signal. It is the propagation delay of the 40kHz ultrasound waves passing through the atmosphere, when reflected by a large object, i.e., a car, that is used to calculate the distance between the radar unit and the object. The distance can be set in the range measuring 5cm to 1·5m. As soon as the preset minimum distance is reached, an acoustic alarm signal will be heard. In practice, this means you are about to hit the obstacle!

IC1 along with crystal X1, and C1, C2 and R1 produce a clock signal which is used to drive IC2, a 14-stage binary ripple counter, and gate N4 of IC4. The output from N4, a Schmitt trigger, drives gates N6 to N10 of IC5 which are used to provide a gated 40kHz pulse to transmitter transducer SEN1.

The reflected ultrasonic pulses are picked-up by the receiver transducer SEN2 and amplified by A2 and A3 of IC6, a low noise J-FET amplifier, and converted into a trigger pulse by A4 of IC6. These are fed to the time/delay comparison circuit formed by IC3 and N1, N2 an N3 of IC4. N5 of IC5 provides the drive to the buzzer, BUZ1.

Resistors R7 and R8 provide a half-supply reference voltage to A1 of IC6 which sets the switching threshold for A4 of IC6. Diode D5 provides protection in the unfortunate event of reverse battery connection.

Specification

Measuring range:	5cm to 1·5m
Transmission frequency:	40kHz
Power supply:	10 to 15V DC 16mA max.

PCB Assembly

The kit is in two main parts, the main PCB (marked P3502B) containing the majority of the ICs and other components, and the receiver PCB (marked P3502S) supporting both transmitter and receiver ultrasonic transducers. Included in the kit is an instruction booklet which lists the complete easy to follow, step-by-step, construction details; so construction should proceed smoothly. The Maplin project construction rating for this kit is set at a 2, indicating that it is easy to build, but not suitable for absolute beginners as setting-up and testing is required.

Bend and fit two wire links at the points marked J on the main PCB. Next mount the five resistors and the five diodes, observing the correct polarity of the diodes. Install the 5·2428MHz crystal by laying the crystal flat against the PCB and securing it by means of a wire strap before soldering the connections. The five IC sockets can now be soldered into the board, followed by the capacitors, again observing the polarity of both the IC holders and the capacitors. The trimmer RV1 and the two 4-pole screw connectors, J1 and J2, can now be soldered to the board followed by the buzzer. Be sure to insert the longest lead of the buzzer into the positive marked hole. Finally insert the five ICs into their respective sockets (all of them orientated towards the 'free' side of the PCB).

Assembling the transmitter/receiver PCB follows a similar method to the main PCB: Firstly the resistors, followed by the IC socket, the transistor T1, the capacitors and finally the six-pole screw connector J1.

The sensors can be soldered either directly into the PCB board or soldered to terminals to provide a vertical mounting. This will depend on how you intend to mount the unit in a box for use in your car. The terminals of the sensors which are connected to the housing must be soldered to the terminal marked '–'. Note SENS1, the transmitter is type MA40A5S or equivalent and SENS2, the receiver, is type MA40A5R or equivalent. Insert the IC, a TL074, into its socket observing the correct orientation.

This completes the assembly of the two PCBs. You should now check your work very carefully making sure there are no dry joints, and the components are in the correct position and orientation.

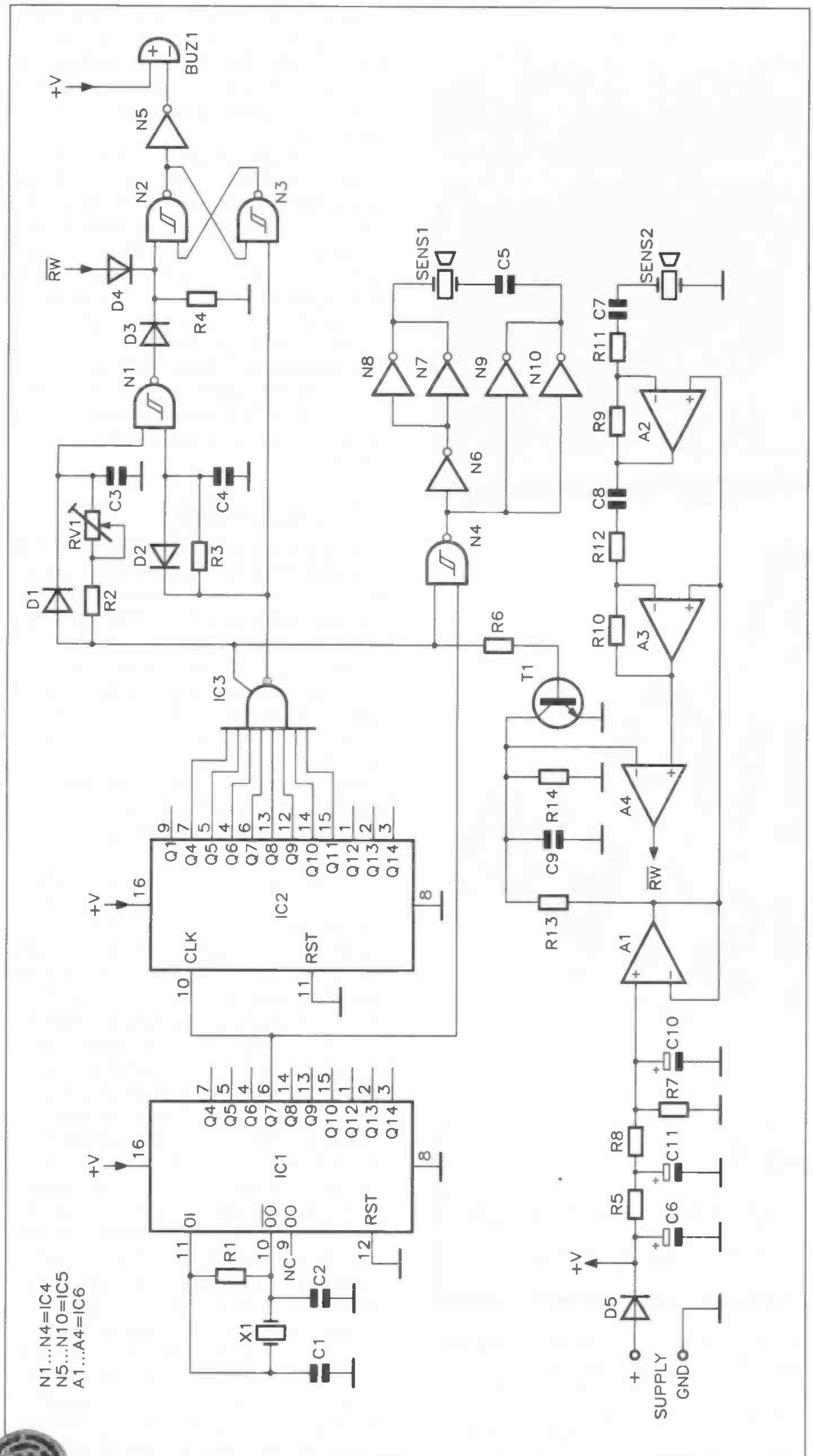
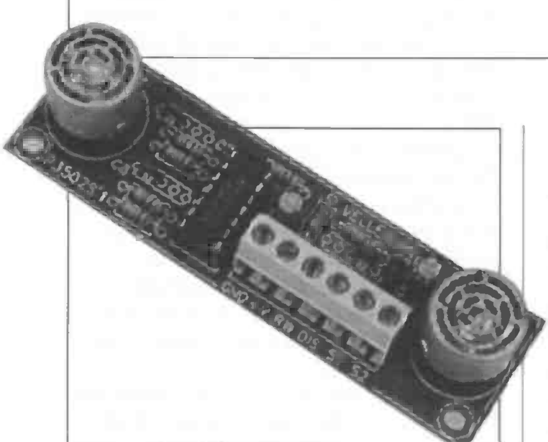


Figure 1. Circuit diagram.



The Receiver PCB with the sensors mounted vertically.

Testing

Connect the points GND, +V, RW, DIS, S1 and S2 on the main PCB to the corresponding points on the transmitter/receiver PCB. Make sure that the distance between the transmitter/receiver PCB and the main PCB is at least 50cm – adjust the trimmer RV1 to its mid-way position. Connect a 12V DC power supply (or a battery) between the points GND and +V. If you hold your hand, or a sheet of paper, in front of the sensors you should hear the sound of the buzzer when the distance from the object to the sensors decrease to about 70cm. If you do not, then check the PCBs and wiring for short circuits etc.

Installation

Mount the transmitter/receiver PCB in a suitable plastic box, which can be in either of two different ways (depending on where the unit is to be fixed on the car):

1) With the sensors in the horizontal position (Figure 2): Fit the four terminals for SENS1 and SENS2 and solder the sensors to the terminals so that they are horizontal with respect to the PCB. Drill two holes in the box as shown. Install the PCB behind the holes using plastic pillars, so that the sensors are aligned facing the holes without touching the box.

or

2) With the sensors in the vertical position (Figure 3): In this case the sensors are simply mounted on the PCB. Now drill the holes in the box as shown on the drawing. Install the PCB behind the holes using plastic pillars, so that the sensors are facing the holes without touching the box.

To help keep water out of the unit, cover the holes on the inside of the box with a piece of very fine wire gauze (see Figure 3). If the meshes of the wire gauze are to wide to prevent water penetration, the gauze should be doubled up two, or four times before fixing it to the holes.

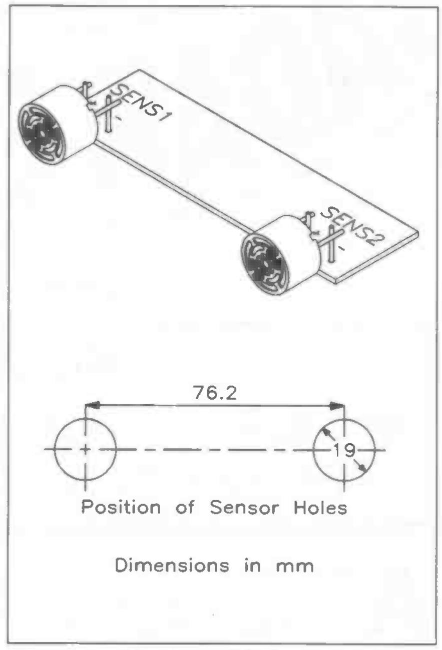


Figure 2. Horizontal method of mounting the sensors.

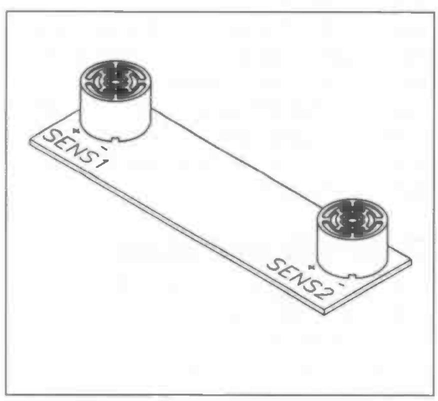
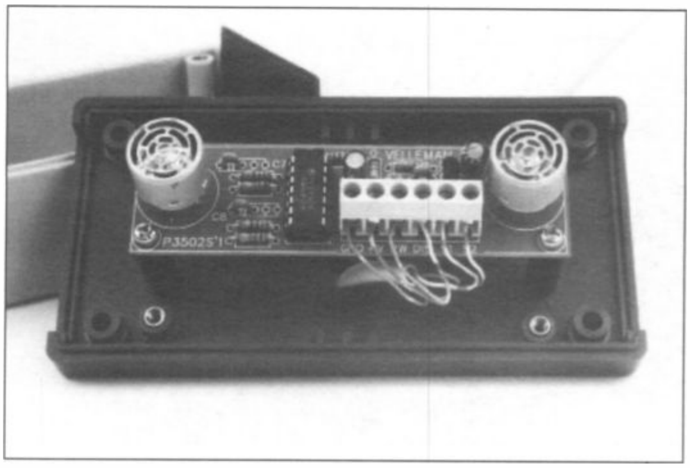
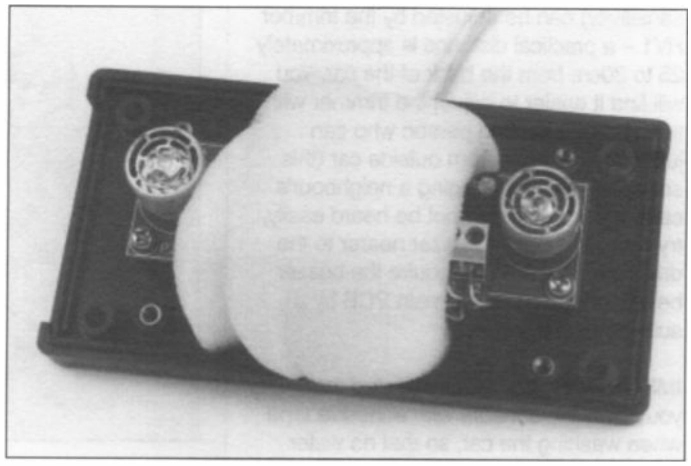
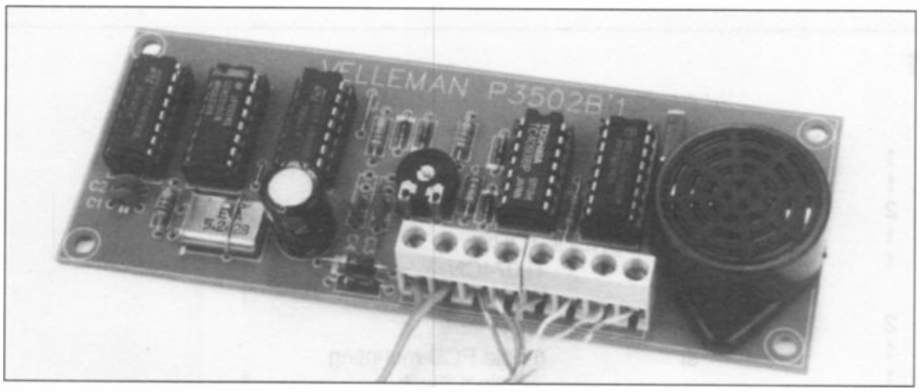


Figure 3. Vertical method of mounting the sensors.







Above left: The completed Receiver PCB mounted in its box.

Above right: The foam rubber supplied with the kit is used to fill the space between the sensors.

Left: The completed main PCB. If preferred, the buzzer may be positioned remotely from the PCB and connected by flying leads.

Fill the space between the two sensors with the piece of foam rubber which is included in the kit. Make a small hole in the bottom of the box in order to allow any water that may have entered the box to run out. When you are satisfied with the installation it is worth sealing the box joints with silicon rubber to help make the box as water tight as possible. Find a suitable place somewhere in the boot or back of the car, to install the main PCB (if possible as close as possible to the sensors, see below). You may prefer to mount the main PCB in a small plastic box for protection, so that the buzzer can be heard a hole, or series of holes, may have to be drilled in the box.

Find a suitable place, at the rear of the car, approximately in the middle, for installing the sensors either underneath or above the bumper. The sensors should be positioned on an horizontal plane. Before fixing the transmitter/receiver unit, connect a length of 6-core cable (or screened 5-core, with the shielding connected to the GND). This cable is then connected to the corresponding connectors on the main PCB.

Connections

Assuming negative earth, connect the GND terminal of the main PCB to the chassis of the car. The +V terminal of the main PCB should be connected to the positive lead on the reversing lamp, via a 100mA fuse.

In Use

The circuit is activated as soon as the gear stick is shifted into reverse (this is indicated by a 'bip' tone) and will now detect any obstacle within the range of the sensors. The detection distance (i.e. the

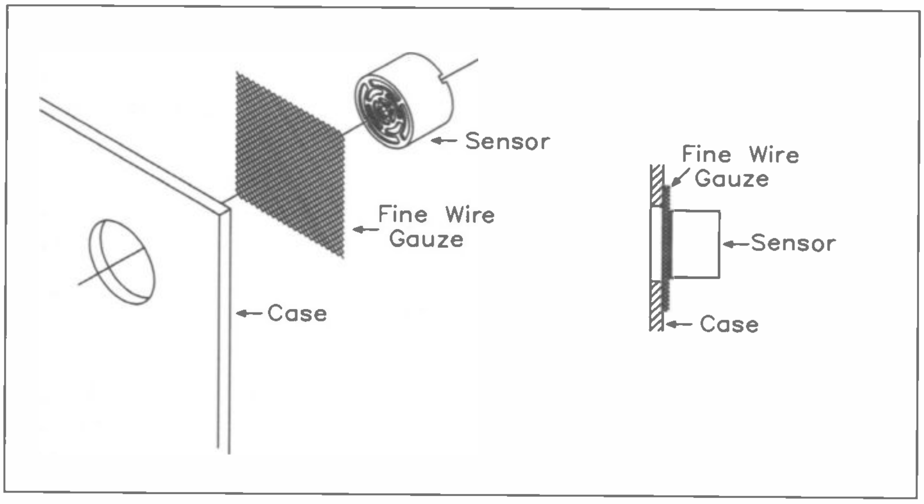


Figure 4. Fitting the fine wire gauze.

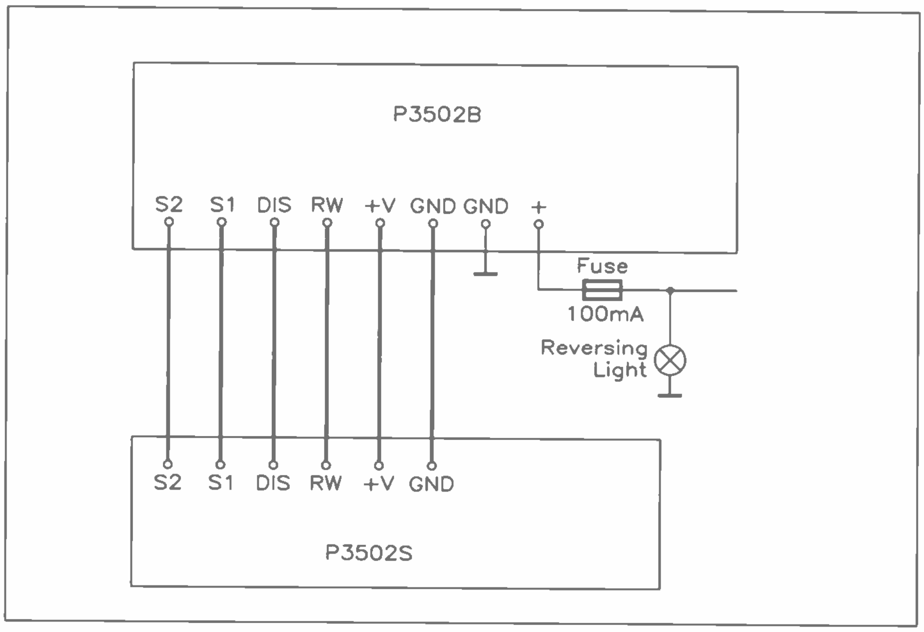


Figure 5. Wiring up the two boards.

sensitivity) can be adjusted by the trimmer RV1 – a practical distance is approximately 25 to 30cm from the back of the car. You will find it easier to adjust the trimmer with the help of a second person who can judge the distance from outside car (this should stop you damaging a neighbour's car!). If the buzzer cannot be heard easily, try repositioning the buzzer nearer to the driver's seat – this will require the buzzer being connected to the main PCB by a suitable long flying lead.

IMPORTANT: It is recommended that you cover the sensors with adhesive tape when washing the car, so that no water gets into the transmit or receiver sensors.

The completed Receiver Unit with protective gauze.

PARKING RADAR PARTS LIST

MAIN PCB

RESISTORS		
R1	10M	1
R2	22k	1
R3, R4	27k	2
R5	470Ω	1
RV1	470k	1
CAPACITORS		
C1, C2	12pF Ceramic	2
C3	22nF Ceramic	1
C4	10nF Ceramic	1
C5	100nF Ceramic	1
C6	470µF Radial Electrolytic	1
SEMICONDUCTORS		
IC1	4060BE	1
1C2	4020BC	1
IC3	4068BE	1
IC4	4093BP	1
IC5	4049BC	1
D1-4	1N4148 (or equivalent)	4
D5	1N4000 Series Diode	1
MISCELLANEOUS		
JI, J2	4-pole PCB-mounting Terminal Block	2
BUZ 1	Buzzer	1
X1	5·2428MHz Crystal	1
	PCB	1
	14-pin DIL Socket	2
	16-pin DIL Socket	3

TRANSMITTER/RECEIVER PCB

RESISTORS			
R7-10	15k	4	
R11,12	1k	2	
R13	10k	1	
CAPACITORS			
C7,8	10nF Ceramic	2	
C9	100nF Ceramic	1	
C10	10µF Radial Electrolytic	1	
C11	100µF Radial Electrolytic	1	
SEMICONDUCTORS			
IC6	TL074CN	1	
T1	BC547	1	
MISCELLANEOUS			
JI	6-pole PCB-mounting Terminal Block	1	
SENS1	MA40A5S (or equivalent)	1	
SENS2	MA40A5R (or equivalent)	1	
	PCB	1	
	14-pin DIL Socket	1	
OPTIONAL (Not in Kit)			
	Plastic Box D-003	1	(ZB01B)
	Plastic Box MB8	If req.	(KC90X)
	M3 Insulated Spacers	1pkt	(FS38R)
	6-Core Cable	As req.	(XS54J)
	Zip Wire	If req.	(XR39N)
	Wire Gauze	As req.	
	Silicon Rubber Sealant	As req.	
	In-line Fuse Holder	1	(RX51F)
	100mA $1\frac{1}{4}$in. Fuse	1	(WR08J)

The Maplin 'Get-You-Working' Service is available for this project, see Constructors' Guide or current Maplin Catalogue for details.
The above items (excluding Optional) are available in kit form only.
Order As VE08J (Parking Radar)
Please Note: Some parts, which are specific to this project (e.g., PCBs), are not available separately.

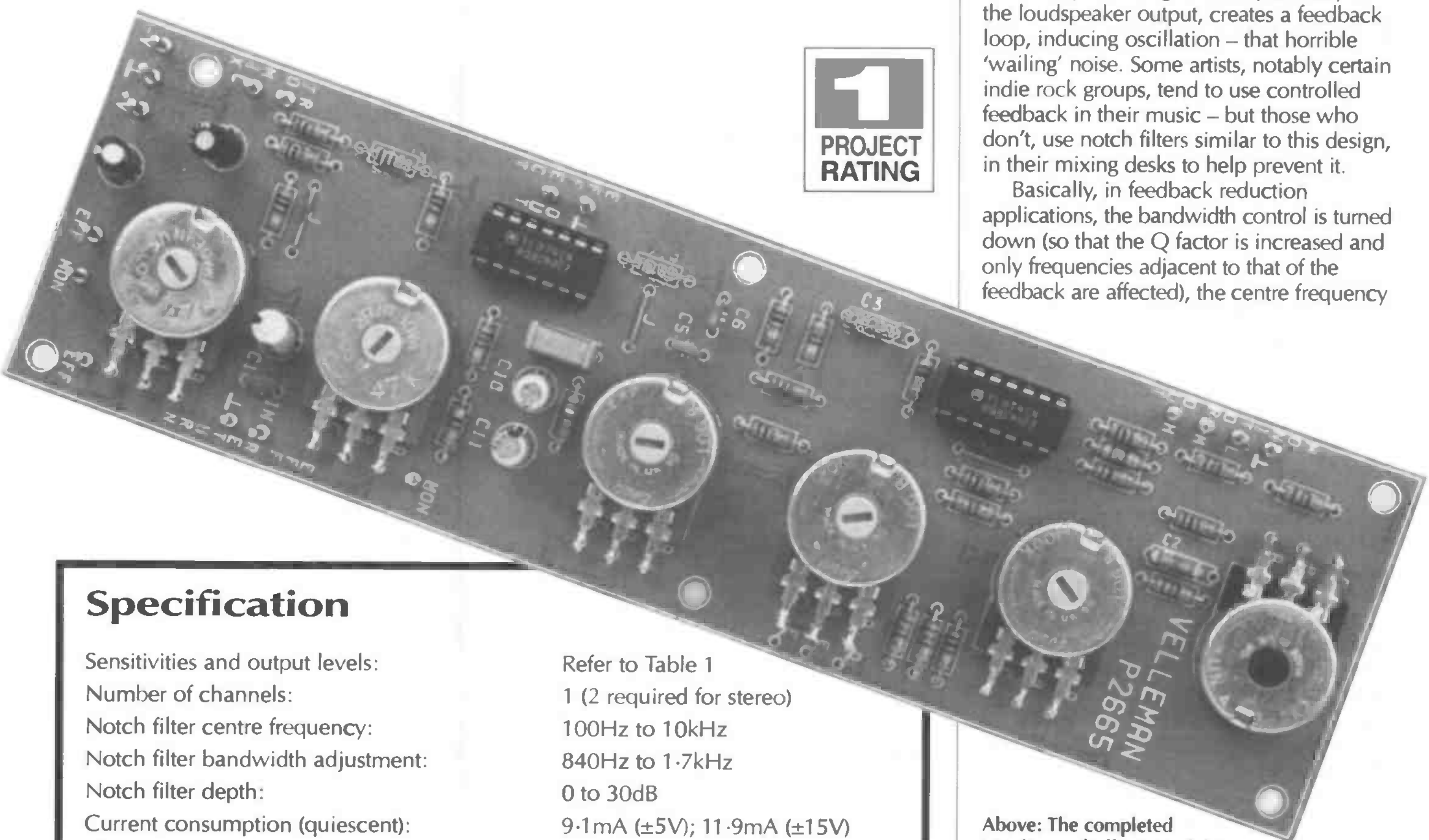
MONITOR EFFECTS AND MODULE

1
PROJECT RATING

This module incorporates all the buffering and amplification for 'effects send' and 'effects return' loops for one of the stereo master channels in a mixing desk, such as the 'Modular Mixing System'. Two are therefore required, one for each channel.

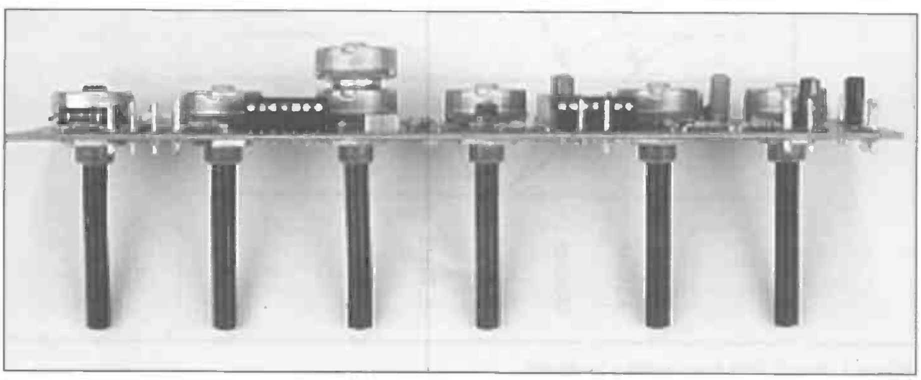
An interesting feature is the notch filter provided for the monitor channel. Apart from simple level attenuation (i.e. a 'volume control'), this circuit provides control over two parameters. Firstly, the Q-factor of the active filter – the 'width' of the notch – is adjustable between 0·4 and 2·5. This corresponds to an –3dB bandwidth variable between 840Hz and 1·7kHz. Secondly, the centre ('target') frequency can be varied across a significant part of the audio band – 100Hz to 10kHz. An adjustable notch filter is considerably useful in greatly reducing, or eliminating, acoustic feedback. Live performers often use 'foldback' speakers, which are arranged to face them so that they can monitor themselves playing. Under certain conditions this can create a terrible din, if the foldback level is advanced too far, since the microphone or guitar pick-up (the input), being in close proximity to the loudspeaker output, creates a feedback loop, inducing oscillation – that horrible 'wailing' noise. Some artists, notably certain indie rock groups, tend to use controlled feedback in their music – but those who don't, use notch filters similar to this design, in their mixing desks to help prevent it.

Basically, in feedback reduction applications, the bandwidth control is turned down (so that the Q factor is increased and only frequencies adjacent to that of the feedback are affected), the centre frequency



Specification

Sensitivities and output levels:	Refer to Table 1
Number of channels:	1 (2 required for stereo)
Notch filter centre frequency:	100Hz to 10kHz
Notch filter bandwidth adjustment:	840Hz to 1·7kHz
Notch filter depth:	0 to 30dB
Current consumption (quiescent):	9·1mA (±5V); 11·9mA (±15V)
Max. current consumption (both monitor and effects outputs into 1k load):	10·4mA (±5V); 13mA (±15V)



Above: The completed Monitor and effects Module.
Below: Side view of the Monitor and Effects Module, showing the arrangement of the potentiometers.

is adjusted to that of the feedback, and the attenuation (notch filter depth) adjusted to the optimum level. In some cases, a further 'fine tweak' of the controls may be required for best results.

Although primarily intended for use in mixers, such as the Modular Mixing System, but it could find uses elsewhere – such as guitar amplifiers and public address systems – where the problem of feedback rears its ugly head. Amateur radio enthusiasts may also find a tunable notch filter of great value when 'chasing DX' – a purpose-designed Maplin kit (LM16S) is available for this

Supply Voltage (±)	Monitor Levels (V_{rms})				Effects Send Levels (V_{rms})		Effects Return Levels (V_{rms})
	Input	Output*			Input	Output	Output**
		Low	Mid	High			
5·0V	2·0	0·6	1·3	2·0	2·0	2·2	1·2
15·0V	5·7	2·9	5·7	9·9	5·7	9·9	4·5

*** with gain of IC2(a) set to 1 ** with gain of IC2 (b) set to 1**
Note that all input measurements were made from a 1kΩ source; impedance of the load was also 1kΩ.

Table 1. Sensitivities and output levels.

application, however. The effects send and return amplifiers would also be useful in guitar amps. If the unit is to be used independently of a mixing desk, however, additional components will be required – see the 'Construction' section.

Note that this circuit requires a symmetrical supply which can be from ±5V min to ±15V max; the power supply voltages dictate the maximum signal levels that can be fed into the module (see Table 1).

Circuit Description

This section should be read with reference to the circuit diagram reproduced in Figure 1. IC2(a) and IC2(b) are input buffers; the gain of each is dependent upon the output impedance of the previous circuit and the values of R1 & R2 respectively (see 'Construction' section). Three of the op amps, IC1(a), IC1(b) and IC1(c), form the filter. RV2 sets the 'Q' of the filter circuit, which is variable from 0·4 to 2·5, and hence its bandwidth. The overall effect is shown in Figure 2. RV6(a) and (b) set the centre frequency, as shown in Figure 3. RV1 controls the amount of attenuation of the selected frequency. The filter network produces a signal in phase with the original; IC1(d) adds the buffered monitor signal, which is inverted, to the output from the filter. Phase cancellation then occurs, the frequencies affected and level of cut, depending on the response of the filter, are determined by the user. As a result, the circuit acts as a 'notch' filter. If an amplifier (with high input impedance) and speaker were to be attached to the wiper of RV1, a signal applied to the input of IC2(a) would sound very 'hollow' around a certain frequency, depending on how the controls have been set.

Note that IC1(d) inverts the signal again, so that it is in phase with respect to the input. There are three different levels available from the monitor output – high, medium and low – that are determined by divider network R11, R18, R19 and R22. IC2(c) is the effects driver amplifier; RV3 controls the level of signal sent to the effect units. IC2(d) is the fixed-gain effects return amplifier, RV4 providing attenuation as required. The effects output is sent to both channels, via R8 and R9 so as to avoid crosstalk at a later stage.

Construction

Construction is fairly straightforward, and full details are given in the leaflet supplied with the kit. If you are new to project building, refer to the Constructors' Guide (order separately as XH79L) for helpful practical advice on how to solder, component identification and the like.

When assembling the PCB, it is best to leave the potentiometers till last as some of the components are located between them – and once the potentiometers are in place, access to these areas of the board is limited. Otherwise, construction is mostly straightforward. Be sure to insert the electrolytic capacitors the correct way round; in addition, note that their leads will need pre-forming before they can fit neatly to the board prior to soldering. The orientation of IC1 and IC2 are also critical; note that these two components are fitted in IC sockets. The sockets, rather than the ICs themselves, should be soldered in place – the ICs should be inserted just before testing.

Each control potentiometer is fitted from the component side of the PCB, and a shakeproof washer is fitted under its securing nut as shown in Figure 4. Before tightening the nut, each potentiometer should be aligned so that its pins line up with the corresponding legend on the PCB; the supplied lengths of tinned copper wire are used for the interconnections as shown in Figure 5. These lengths of wire (or alternatively the component lead off-cuts) are also used for the three wire links on the board.

When fitting the PCB pins, note that they are also fitted from the component side. After completing assembly, it is prudent to check your work – finding any incorrectly-placed components could save considerable time and expense later on. Other gremlins to watch out for include solder bridges/whiskers and poor joints.

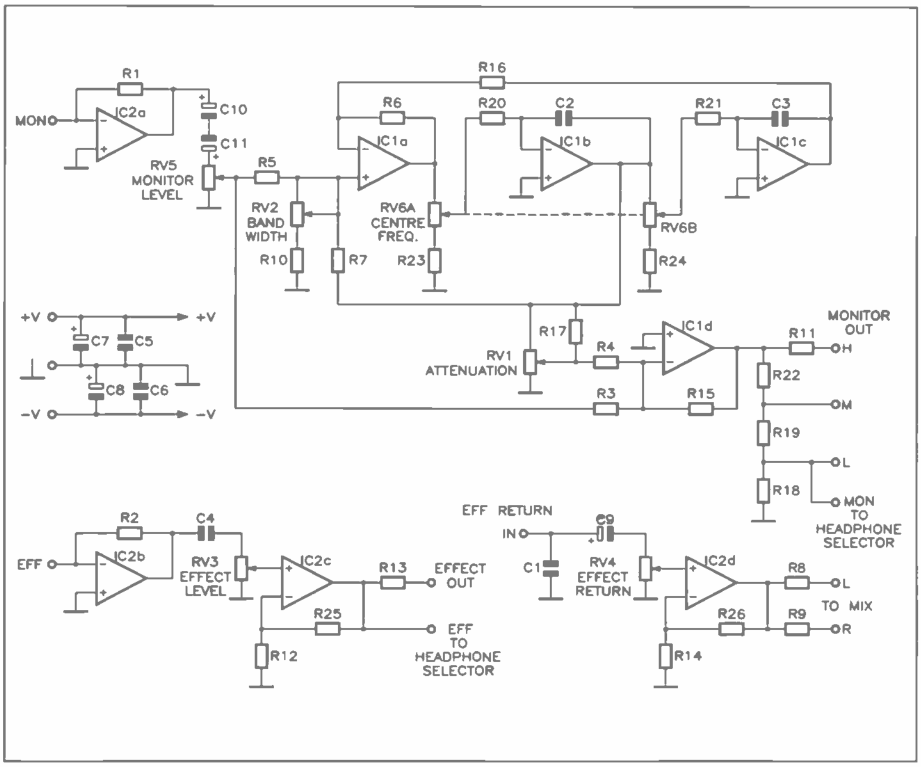


Figure 1. Circuit diagram of the Monitor and Effects Module.

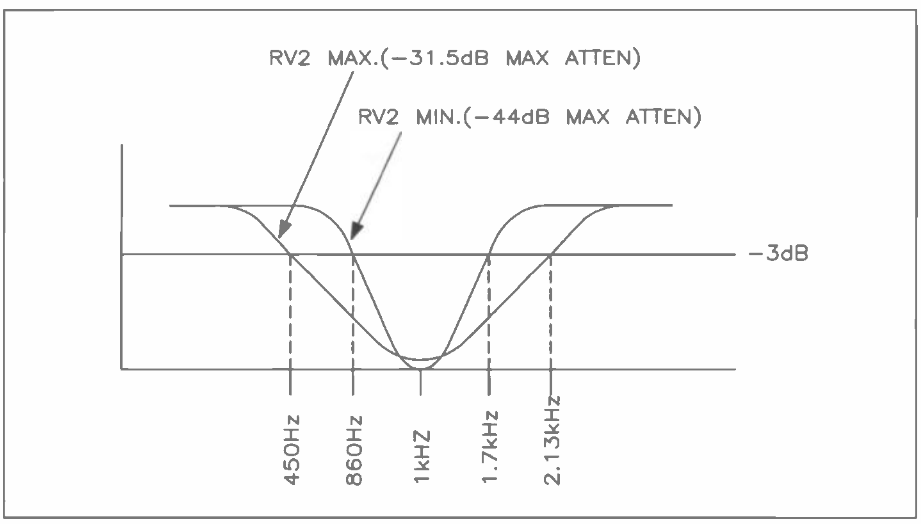


Figure 2. Notch filter bandwidth adjustment.

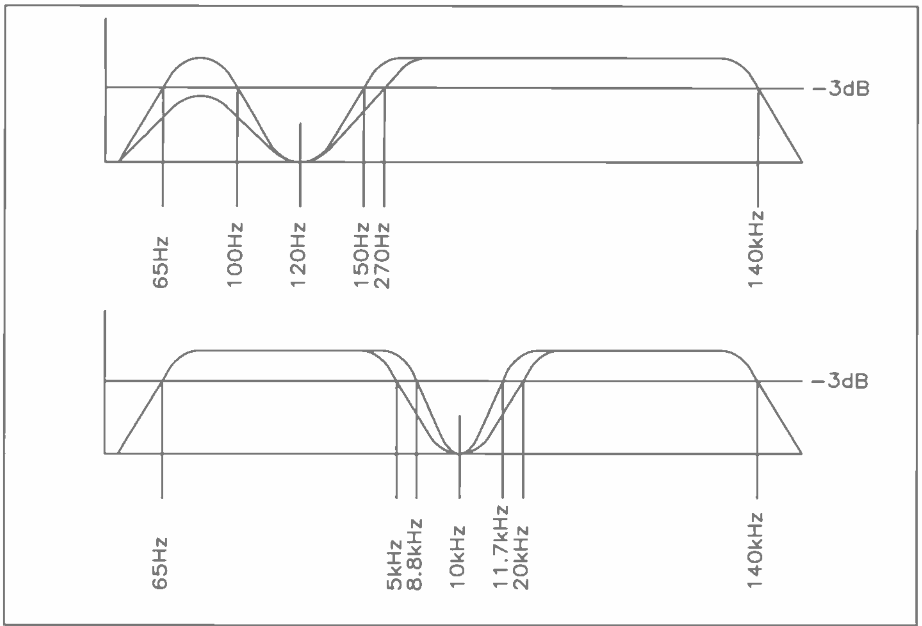


Figure 3. Notch filter centre frequency adjustment.

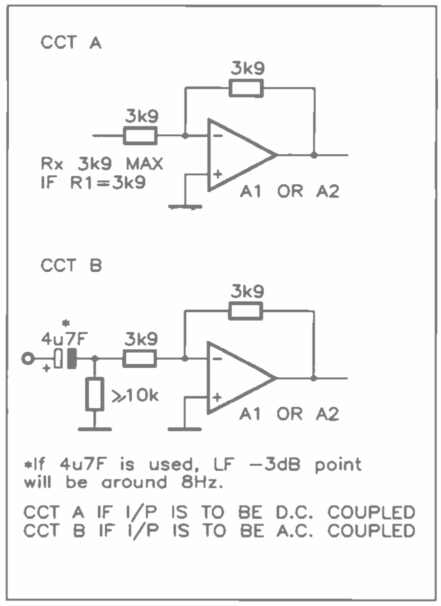


Figure 6. Modifying input stages of Monitor and Effects Module for 'stand-alone' use.

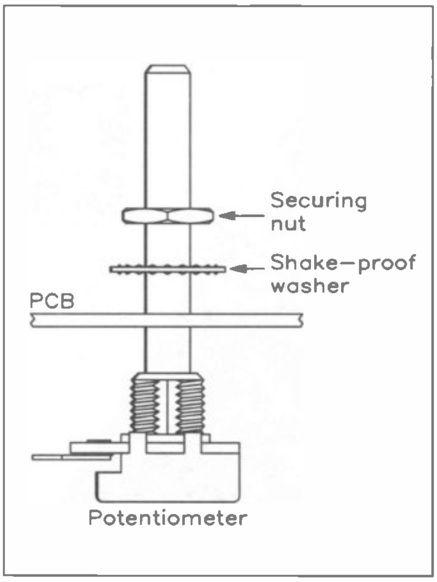


Figure 4. Fitting the potentiometers.

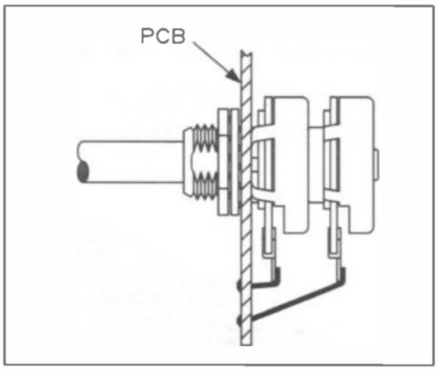


Figure 5. Connecting the potentiometers.

If the Monitor and Effects Module is being used with the Modular Mixing System, R1 should be 8k2 for a 6-channel system, or 3k9 for a 12-channel system. If the circuit is to be used on its own, then IC2(a) and IC2(b) must be fitted with input resistors as shown in Figure 6. This is so that the gain of the input buffers is kept at unity; of course if gain is required, different values may be used.

Testing and Installation

The best form of testing is to use the Monitor and Effects Module in its intended application. Ensure that each power supply rail can provide the required current. For best dynamic range, the full ±15V supply voltage should be used – see Table 1.
The module should be installed away from any strong mains fields (power transformers and the like), in a screened case. If the completed board is to form part of a modular mixing system, it should be built into a decent metal case anyway! Screened cable (such as XR15R) should be used for all audio connections to reduce the possibility of hum pick-up. Once the installation has been inspected, the ICs can be inserted and the system powered up.

MONITOR AND EFFECTS MODULE PARTS LIST

RESISTORS: All 0·25W Metal Film (Unless specified)

R1,2	3k9 or 8k2 (see text)	2 of each value
R3-9	10k	7
R10-14	1k	5
R15,16	100k	2
R17-19	1k5	3
R20,21	4k7	2
R22	1k8	1
R23,24	120Ω	2
R25,26	2k7	2
RV1	2k2 or 2k5 Lin Pot	1
RV2	100k Log Pot	1
RV3,4	47k Log Pot	2
RV5 4k7	Log Pot	1
RV6 10k	Dual Log Pot	1

CAPACITORS

C1	180pF Ceramic	1
C2,3	1nF Ceramic	2
C4	220nF Poly Layer	1
C5,6	100nF Monores	2
C7,8	10μF 10V PCB Elect	2
C9	1μF 50V PCB Elect	1
C10,11	2μ2F 50V PCB Elect	2

SEMICONDUCTORS

IC1,2	TLO74CN Quad Op Amp	2

MISCELLANEOUS

PCB	1
DIL Socket 14-pin	2
PCB pins	2 Pkts
Construction Leaflet	
Tinned Copper Wire	

OPTIONAL (Not in Kit)

3k9 5% Metal Film	2	M3K9
4·7μF Minelect 63V	2	RA53H
Constructor's Guide	1	XH79L
Screened Cable	As required	XR15R

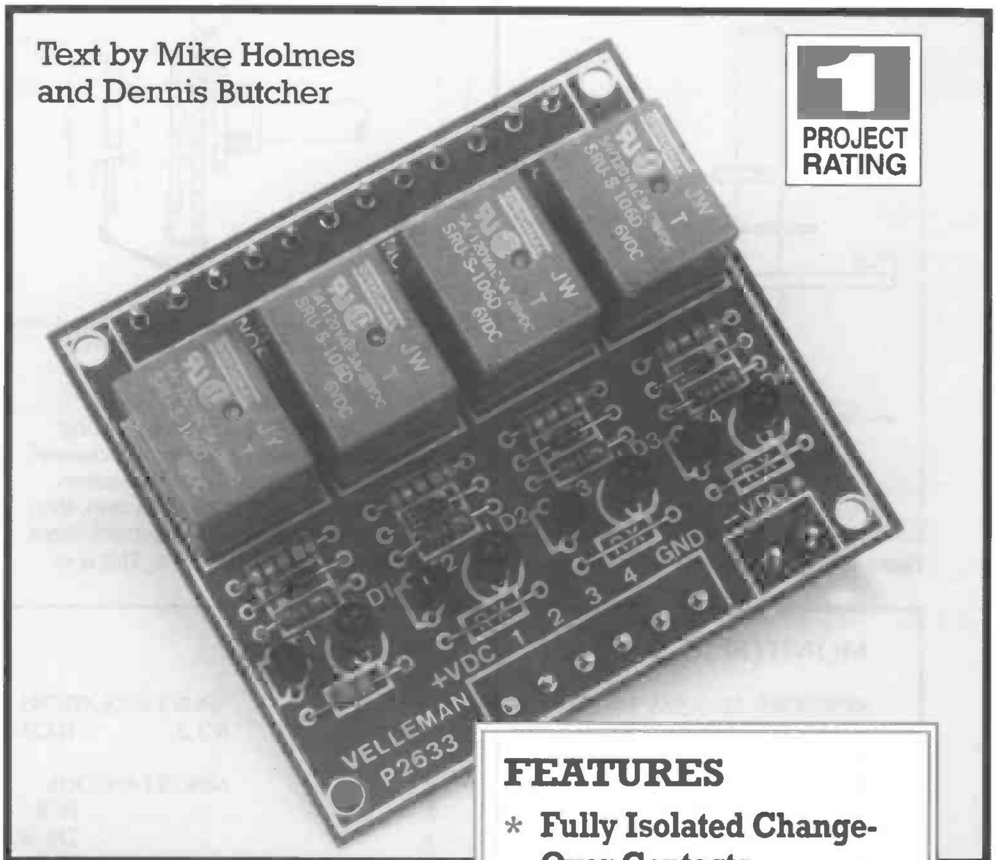
The Maplin 'Get-You-Working' Service is available for this project, see Constructors' Guide or current Maplin Catalogue for details.
The above items (excluding Optional) are available in kit form only.
Order As VE34M (Monitor and Effects Module)
Please Note: Some parts, which are specific to this project (e.g., PCB) are not available separately.

RS232 SERIAL PORT EXPANSION SYSTEM

PART FOUR

Add-on Relay Card

Text by Mike Holmes and Dennis Butcher



The completed Analogue to Digital Converter Card.

This module is intended for use with the Open-Collector Card (VE92A) previously described in 'Electronics' Issue 60, and serves to increase the versatility of the RS232 Serial Port Extension System still further (although of course there is nothing to stop you using the Relay Card in other applications).

The module enables the RS232 Extension System to switch heavier loads with isolated supplies than would otherwise be possible with the transistor switches on the plug-in Open-Collector Card. To save cost the module only has four relays (all of which are single-pole change-over types), but if you need more, two modules can be connected to one Open-Collector Card thus providing eight independently operated relays. In fact, if the Extension Card is filled with four plug-in Open-Collector Cards, it is possible for the Extension System to drive a total of 32 relays.

Circuit Description

The circuitry for the card is, as you might imagine, quite simple and merely consists of four duplications of the circuit shown in Figure 1. In this instance the input is from one output of an Open-Collector Card and component 'Rx' should be ignored for the moment (more on this later).

RY1 is the relay, and is energised by transistor switch T1. Because the Open-Collector Card output is effectively open circuit when 'off', R1 is included to ensure that T1 is fully off by pulling up its base to +V DC. The recommended level for +V DC is 9V; this provides sufficient 'overhead' for reliable operation of RY1, which is a 6V operating type. The difference is taken up by R5, and in fact the circuit is quite tolerant of supply voltage range which may be increased up to +15V, at which point the 'on-time' current consumption of RY1 will still only be 61mA, it being a low current type.

D1 is provided to protect T1 from the usual collapsing-field generated, voltage spike across RY1 at switch-off. Visual indication that this relay is active is provided by LD1, which is included in the input path. This means that the input current needs to be >=12mA to achieve sufficient voltage drop across R1 to bias on T1. When connected to the output of an Open-Collector Card, 'Rx' is omitted and replaced with a wire link, since the output path of the Open-Collector Card includes a 560Ω resistor (R18–25, also see Issue 61). You may consider that this arrangement is somewhat unusual, but it means that the card is *current*, as opposed to voltage, operated, which will ensure excellent immunity from interference over long cable runs. This is important if the card is remote from the Open-Collector Card.

It's worth noting that, with these values and with the Relay Card powered from a 9V source, this divider chain somewhat precisely sets the bias level for T1 base at just over 0·6V, taking into account the voltage drop across LD1, which may result in unreliable operation of the Relay Card should its supply level fall below +9V. (This may be obviated by reducing the value of the appropriate resistor(s) at position(s) R18–25 on the Open-Collector Card to 470Ω, for example.)

If the values of resistors R18–25 on the Open-Collector Card are significantly reduced, or replaced with wire links for some (very good) reason, or if the Relay Card is intended to be used separately from the Serial Port I/O System for some other purpose, then you MUST install a current limiting resistor at position 'Rx', and NOT use a wire link! (In the latter case either LD1, the base/emitter junction of T1, or both will 'blow' as soon

FEATURES

* Fully Isolated Change-Over Contacts
* LED 'On' Indicators
* Remote Installation

APPLICATIONS

* Model Train Control
* Flashing Signs
* Motor and Solenoid Control
* Security Applications, Door Locks, etc.

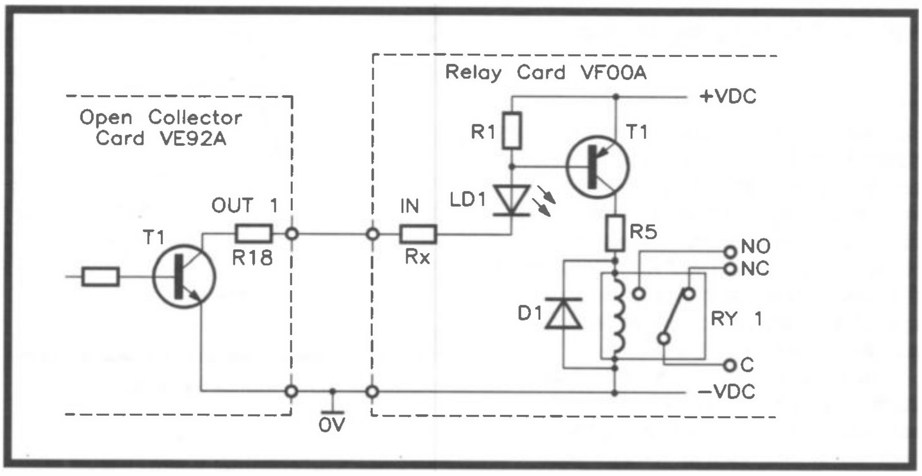


Figure 1. Circuit of one relay and driver on the Relay Card.

as the input is connected or switched to 0V.) Four 820Ω resistors are provided in the kit for use in the positions 'Rx' for this purpose, but be advised that these values will only really be suitable in situations where the card is powered from a DC supply in the range of 12 to 13V. For 9V operation they should be 560Ω each in order to maintain the LED current and base bias for T1.

To calculate the value of 'Rx' from a given supply voltage the following simple formula can be used:

$$RX = \frac{(+Ve-2{\cdot}7)}{0{\cdot}012}$$

For example if +Vs = 15V, then

$$\frac{(15-2{\cdot}7)}{0{\cdot}012} = 10250$$

where 2·7 is the combined voltage drop across both LD1 and T1 base/emitter, and 0·012 is the recommended LED current. From the above example the next lowest resistance value in the E24 scale of resistor values will be 1000Ω, to be used in position 'Rx'.

Construction

The board is quite simple to build, once you have decided how it is going to be used, as explained above. It will be helpful to leave the relays until last, as these take up the most space and can restrict access to some of the smaller components. Begin by inserting the double-ended PCB pins at positions '+VDC', '1' to '4' and 'GND' on the PCB, and also at '–VDC +'.

The remaining pins may be inserted at the opposite edge of the card at all positions 'NO', 'C', 'NC', etc., but at this stage you can decide to opt for PCB mounting screw terminal blocks instead (see Optional Parts List), allowing more convenient, non-permanent connections to the relay contacts. Four 3-Way PCB terminal blocks can be stacked and fitted in these positions forming a single 12-Way block.

Next fit resistors R1 to R8, and then use offcuts from these to form the wire links for the four locations 'RX' if the card is going to be driven from an Open-Collector Card. Otherwise current limiting resistors must be fitted, as already mentioned, choosing a value according to the Relay Card external supply level as indicated in the following table:

Card Supply	Rx Value
9–11V	560Ω
12–13V	820Ω
14V	910Ω
15V	1k

You can now fit diodes D1 to D4, making sure to align the cathode identifiers (black stripe on the diode body) with the white markers on the legend (the orientation alternates across the board). Transistors T1 to T4 can then be installed, followed by the four red LEDs. Note that the card can be fitted on pillars behind a panel, in a situation where it is remote from the remainder of the Extension System, and these LEDs can then be visible indicators through appropriately positioned holes. In this event you should make the tops of the LEDs slightly higher that the tops of the relays (21mm) in order to reach the panel.

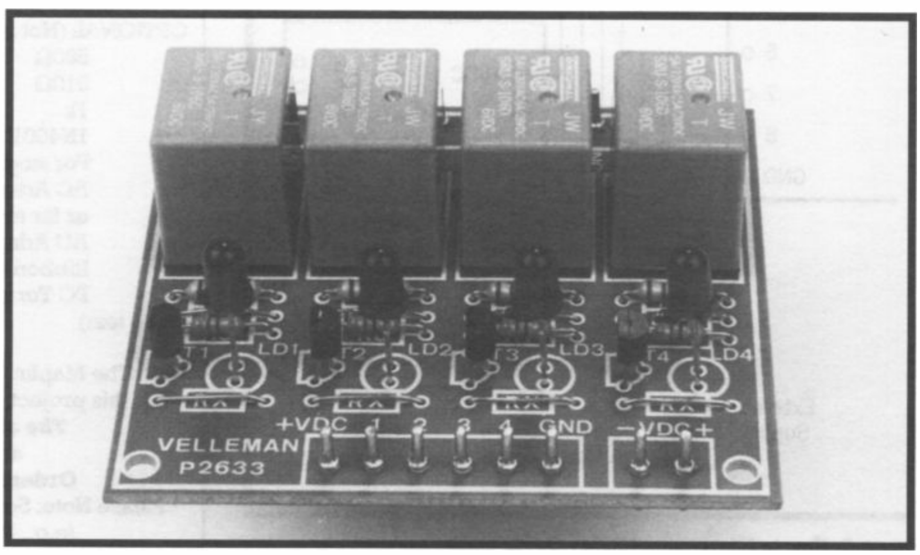
When fitting each LED, note that the shorter of the two leads is the cathode (corresponding to the flat on the LED body) and must be inserted in the hole nearest positions 'RX'. To set the height accurately, lightly solder one lead only, and then briefly re-melt to alter the height then hold it till the solder sets. When satisfied, the other lead can be soldered permanently and the first lead re-soldered properly, before removing the excess leads with side-cutters.

Finally each of the four relays RY1 to RY2 can be installed, making sure to use good, strong solder joints. This completes construction of the card and, after double-checking correct placement of components, the quality of solder joints and ensuring that there are no solder bridges, etc., the Relay Card can be tested.

The row of PCB pins opposite the relays will be used to connect the four inputs '1' to '4', and the common supply and input 0V 'GND' to the Open- Collector Card on the Serial Extension System extension board. The pair '–VDC +' are commoned through to 'GND' and '+VDC' respectively, and it is to these that the external supply is connected. Pin '+VDC' is only used to carry this supply to any further cascaded Relay Cards, and must not be connected to any supply pin on the System Extension Card!

Figure 2 illustrates this by showing how two Relay Cards are combined to give full 8-bit control of eight relays. All pins 'GND', including that on the Open-Collector Card, are commoned; inputs '1' to '4' of Relay Card No. 1 connect to outputs '1' to '4' on the Open-Collector Card, while inputs '1' to '4' of Relay Card No. 2 connect to outputs '5' to '8'.

The '–' terminals of both cards for the external 9V supply are commoned together but, while the external supply '+' connects to the '+' of card No. 2, it is transferred via pin '+VDC' to card No. 1. In this way several Relay Cards can be cascaded together. The external 9V supply must be able to supply 300mA for



The assembled Relay Card.

each card. For example mains adaptor XX09K can be used to power one Relay Card, and YM85G for two (see Optional Parts List). The supplies need not be stabilised.

Relay Connection and Testing

Connecting to the relay contacts is very simple: each group is marked 'NO', 'C' and 'NC' where 'NO' = Normally Open, 'C' = Common (the 'pole' of the switch) and 'NC' = Normally Connected when the relay is in the 'off' state. In use the Open-Collector Card merely has to send a logic 0 for 'off' or a logic 1 for 'on' on the relevant data line. The last program listed near the end of the Open-Collector Card article in Issue 60 (under the heading 'Another Test Program') can be used again to perform a running test on the Relay Card, operating each relay in sequence. The Relay LEDs will show which one should be active, and the relay itself should be heard to click. An ohmmeter or continuity tester may then be used to prove that each set of contacts is changing over properly. Initially all relays will be switched on for one second, and then each in turn, starting with number 1 (bit 0), will be switched on individually for one second, and then all off before the cycle resumes. (If only four relays are present there will be a five second delay between number 4 going off and all going on again.)

In Use

Although the leaflet states that each relay contact is rated at 3A at 240V AC, this is not necessarily correct. The actual rating should be displayed on the housings of the relays themselves, and is normally 5A at 28V DC (and 120V AC). Because of the closeness of the PCB tracks connecting the relay contacts, you are strongly advised not to exceed the DC rating.

For switching inductive DC loads such as motors or higher power solenoids, etc., you should connect rectifier diodes such as 1N4001 (see Optional Parts List) across both the load and the relay's contacts, as illustrated in Figure 3. These will prevent arcing across the contacts themselves and reduce RFI.

Ribbon cable is mentioned in the Optional Parts List, which will be most convenient for quickly connecting together Relay Cards and Open-Collector Cards, having ten, colour coded wires in a strip. Of these, 'black' can be used for the common '0V' and 'GND' connections and 'white' for the '+V DC' connections, the remainder used as signal carriers.

Ideas for applications of the Relay Card are limited only by your imagination, especially where such switching capability is teamed up with an intelligent device, a computer. One or two ideas include model railway layout control, where trains can be controlled on independent sections of track and made to start and stop at stations and signals, and points and motorised crossing gates can be operated in unison. Alternatively, as an intelligent 'zoned' security controller, the system can disable and enable switch sensors for an alarm system, such as door contacts, PIRs, pressure mats, etc., for variable periods according to preset times of the day or week. In fact the system is ideal for all manner of timed switching functions.

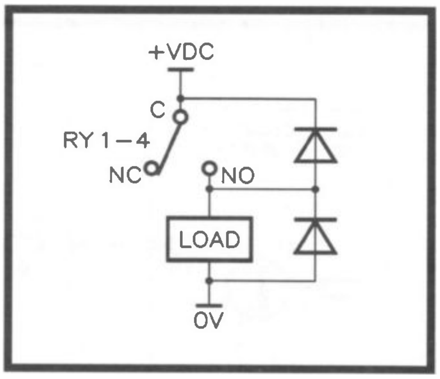


Figure 3. Using protective diodes with inductive loads.

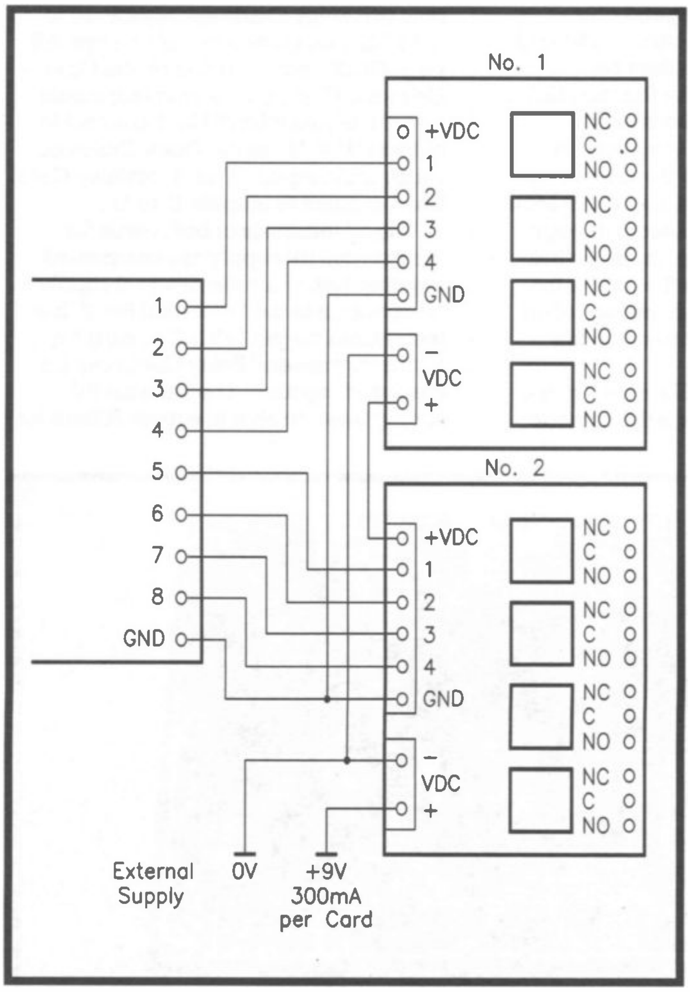


Figure 2. How to interconnect Relay Cards to an Open-Collector Card.

SERIAL SYSTEM RELAY CARD PARTS LIST

RESISTORS: All 5% Metal Film			
R1-4	56Ω	4	
R5-8	82Ω	4	
Rx	820Ω	4	
SEMICONDUCTORS			
D1-4	1N914 (1N4148)	4	
LD1-4	Red LED	4	
T1-4	BC557 (BC558, BC559, BC307, BC308, BC309)	4	
MISCELLANEOUS			
RY1-4	Relay SPCO 5A 28V	4	
	PCB Pins	20	
	PCB	1	
	Instruction Leaflet	1	
	Component Identification Guide	1	
OPTIONAL (Not in Kit)			
	560Ω	4*	(M560R)
	910Ω	4*	(M910R)
	1k	4*	(M1K)
	1N4001	As req.*	(QL73Q)
	For single card:		
	AC Adaptor 9V 300mA	1	(XX09K)
	or for two cards:		
	AC Adaptor 9V 800mA	1	(YM85G)
	Ribbon Cable 10-Way	As req.*	(XR06G)
	PC Terminal Blocks 3-Way	4	(JY94C)

(*see text)

The Maplin 'Get-You-Working' Service is available for this project, see Current Maplin Catalogue for details.
The above items (excluding Optional) are available in kit form only.
Order As VF00A (Serial Sys Relay Crd)
Please Note: Some parts, which are specific to this project (e.g., PCB), are not available separately.

/// Maplin /// Maplin /// Maplin