NDD

15 ELECTRONIC PROJECTS

ETI
ELECTRONICS TODAY INTERNATIONAL

$4.95
NZ: $6.50 INCL G.S.T.

TEST GEAR

COMPILED BY ROGER HARRISON

VOL 4

How to build Test & Measurement Gear for your Home Workshop

BUILD:

LOW DISTORTION OSCILLATOR ★ ANALOGUE BREADBOARD ★ DIGITAL TRANSISTOR TESTER
OP-AMP TESTER ★ ZENER TESTER

CONTENTS

TEST GEAR VOL. 4

Welcome to ETI's Test Gear Volume 4. It's quite a while since we published a Test Gear book, so this compilation of test equipment projects previously published in Electronics Today will prove very popular, we believe.

The projects range from simple, practical, low-cost devices such as the ETI-176 Zener Tester, through unique instruments like the ETI-182 Digital Lux Meter, up to "high-tech" units such as the ETI-169 Low Distortion Oscillator. As we know your interests range far and wide, projects for this compilation have been chosen to satisfy quite a range of fields — from everyday electronics construction and servicing, through computer troubleshooting, to component testing.

So, almost no matter what your field of interest, you should find something of use within these pages. For a guide to the availability of parts, printed circuit boards and kits, don't forget to turn to the "Shoparound" page.

Roger Harrison
Editor

COMPILED BY
Roger Harrison

ETI EDITOR AND MANAGER
Kim Bucknole

PRODUCTION EDITOR
Allison Tait

PRODUCTION
Tracy Douglas

ADVERTISING PRODUCTION
Brett Baker

SECRETARY
Michelle Smith

ELECTRONICS EDITOR
Roger Harrison VK2ZTB

ELECTRONICS SECTION
Compiled by The Apogee Group

EDITORIAL ASSISTANT
Jamye Harrison

DRAFTING
Graeme Knight
Jamye Harrison

Enquiries to The Apogee Group on (02) 555 1646 fax (02) 818 2949

EDITOR-IN-CHIEF
Brad Boxall

PUBLISHER
Michael Hannan

CIRCULATION MANAGER
Michael Prior

HEAD OFFICE
180 Bourke Road, Alexandria, NSW 2015.
PO Box 227, Waterloo, NSW 2017.
Ph: (02) 693 6666.
Federal Facsimile: (02) 693 9935

ADVERTISING
New South Wales: Janathan Paynter, NSW Advertising Manager, Charles Bales, Sales Executive, The Federal Publishing Company, 180 Bourke Road, Alexandria, NSW 2015. Ph: (02) 693 6666 Facsimile: (02) 693 9997.
Victoria and Tasmania: Valerie Newton, Victorian Advertising Manager, The Federal Publishing Company, 221a Bay Street, Port Melbourne, Vic 3207 Ph: (03) 646 3111, Facsimile (03) 646 5494.
Queensland: Graham Smith, The Federal Publishing Company, 26 Chermside Street, Newstead, Qld. Ph: (07) 854 1119. Facsimile: (07) 252 3692
South Australia and Northern Territory: Michael Mullins, C/- Federal Publishing, 98 Jervais Street, Torrensville, SA 5031. Ph: (08) 352 8666. Facsimile: (08) 352 6033
Western Australia: Estelle de San Miguelle, 118 Farrest Street, Cottesloe, WA 6011 Ph: (09) 385 3332. Fax (09) 385 3700
New Zealand: Gordon Marr, The Federal Publishing Company, 67-73 View Rd, Glenfield, Auckland. Ph: 443 0954 Facsimile: 443 1326.
Britain: Peter Halloway, C/- John Fairfax & Sons, 12 Norwich Street, London EC4A 1BH Ph: 353 9321.

ELECTRONICS TODAY INTERNATIONAL is published and distributed monthly by The Federal Publishing Company Pty Limited, inc. in NSW, 180 Bourke Road, Alexandria, NSW 2015 Ph: (02) 693 6666. Printed by Hannanprint, 140 Bourke Road, Alexandria, NSW 2015. 1989. Distributed by Newsagents Direct Distribution, Alexandria, NSW 2015. Distributed in New Zealand by Network Distributors Ltd, 67-73 View Road, Glenfield, Auckland. Ph: 443 0245 Facsimile 443 0249 *Maximum and recommended Australian retail price only.

LOW DISTORTION AUDIO OSCILLATOR

Part 1

Our gnomes in the market research field have told us that ETI readers are more interested in hi-fi than the average. No great revelation, you probably built your own system? Well this project puts that to the test. It's an audio oscillator that allows you to accurately measure audio performance so you can be more authoritative than the average.

Ian Thomas

JUST ABOUT EVERYBODY in this civilised country of ours has a hi-fi system of some sort or another. Since you're reading this magazine you almost certainly take a more than passing interest in how well it works and have probably built some, maybe all of it yourself. If, like most of us, you aren't blessed with infinite financial resources then you probably put it together and said "well it seems to sound OK" and left the testing of your masterpiece at that. You probably knew that this wasn't the most definitive of tests but put the acquiring of good audio test equipment in the "too hard (expensive)" basket. At the request of our editor I've finally gotten around to doing something to rectify this situation.

The very first and most important piece of test equipment you'll need is an audio oscillator to provide a signal source whose performance is at least an order of magnitude better than the equipment you want to test. Audio oscillators have been around for an awful long time now and people have built very large and successful businesses around them. I've been assured that it really is true that two gentlemen by the names of Bill Hewlett and Dave Packard started building simple one valve Wien bridge audio oscillators in their garage in 1939. Their company is now worth billions of dollars so there has to be a future in it! Perhaps you could duplicate their efforts (if you do please remember where you got the idea!).

In modern hi-fi equipment the most commonly measured and quoted gauge of performance is referred to as the total harmonic distortion of the equipment and is usually given as a percentage although in the professional literature it is often given in dB. When a signal is passed through equipment such as an amplifier the signal that emerges from the equipment is not an exact replica of the signal that went in. The amplifier tends to change the waveform slightly in the process of amplifying it.

In order to test the equipment it is necessary to apply a signal to the input that is exactly known so that some form of test can be applied to the output to see if it has been corrupted. Normal voice or music signals are very complex and it is impossible to apply any form of quantitative test to them apart from "well it sort of sounds OK". By far the simplest form of signal to handle both mathematically and in test equipment is a sine wave which can be described as

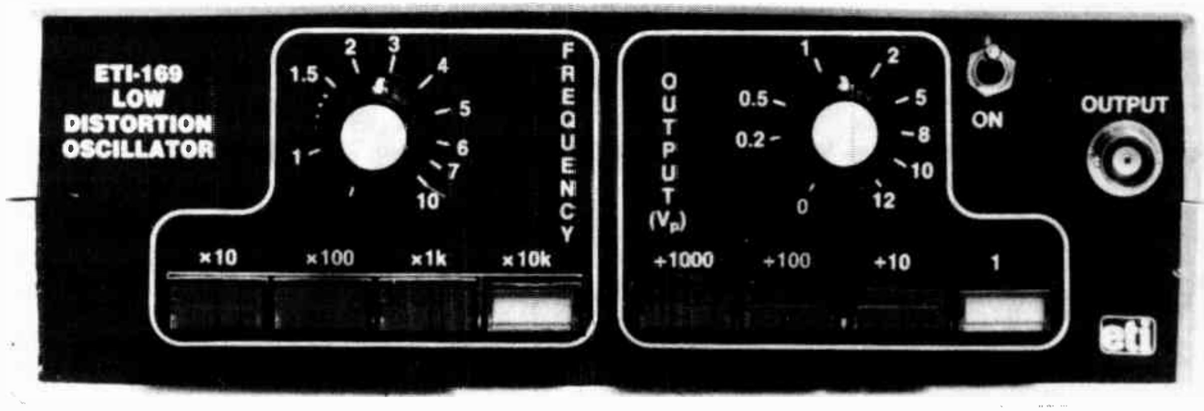
$$V = V_0 \sin(2\pi ft)$$

where f is the frequency of the sine wave. If this signal is applied to the input of an imperfect amplifier (as they all are) what emerges will be the input sine wave multiplied by the gain of the amplifier *plus* other signals as well. These other signals which are generated in the amplifier will have frequencies of 2f, 3f, 4f and so on. These are called harmonics of the input signal and are caused by distortion in the amplifier itself.

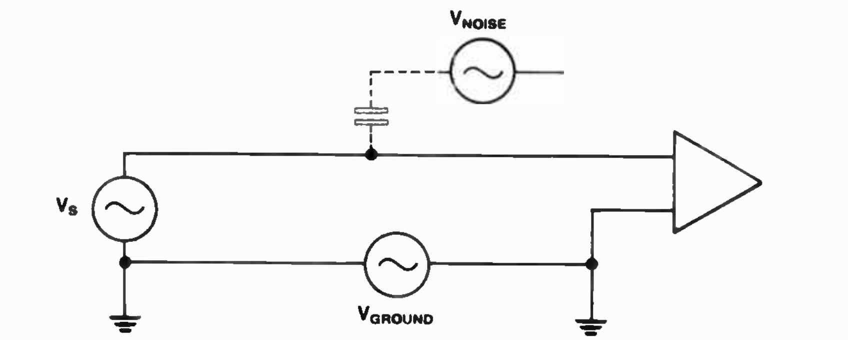
A bloke by the name of Fourier way back when put all this on a mathematical footing which I won't bore you with (assuming I could remember it) but the short of it is that if you have a regular periodic waveform which isn't a sine wave then it can be made up by the sum of a lot of sine waves with frequencies that are harmonically related. As the signal that emerges from an amplifier when a perfect sine wave is put in is no longer a perfect sine wave, the output can be dissected into the amplified perfect sine wave and the resultant Fourier components which are harmonics of the input signal. Because they are harmonics of the sine wave input the distortion is called harmonic distortion; if all the harmonics are lumped together and measured the result is a total of all the harmonics — hence total harmonic distortion! Now you know what it really means when you see 'thd' on the data sheet.

For more consumer equipment the harmonic distortion is always given as a total but in many applications the level of each individual harmonic is given separately or odd and even harmonics may be separated out. This can help a lot in identifying the mechanisms that are causing the distortion.

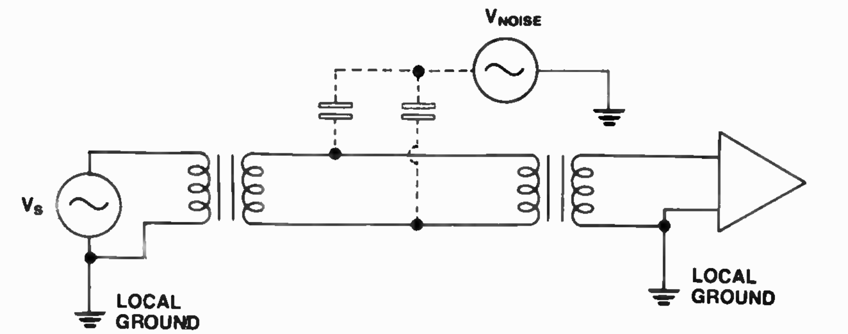
The nub of all this is that the signal to be applied to the amplifier must not contain any harmonics of its own or the harmonics from the oscillator can't be separated from those generated in the amplifier and spuriously high (or even low!) readings are obtained. Hence the need for a low distortion oscillator. Most cheapie sine wave oscillators are absolutely useless for testing audio equipment as their disortion is usually around 1% or −40 dB. That is, the total of all the harmonics is one hundredth of a fundamental sine wave. Even the grottiest amplifier can beat this hands down; in fact you'd have to work at it to build one this bad.



BALANCING OUTPUTS



UNBALANCED
In an unbalanced condition the signal seen at the input of the receiving amplifier is degraded by the addition of any ground loop voltages plus any stray signals that are capacitively coupled into the line.



BALANCED
With balance, ground loop voltages have no effect on the input as the input transformer 'floats' the wanted signal on top of any ground signals and the amplifier is able to ignore them. Capacitively coupled signals tend to be equally coupled into both lines of the balanced pair to be removed by the transformer.

Amplifier distortions normally run from 0.1% or −60 dB (crummy) to 0.001% or −100 dB (so good it's very hard to measure and doesn't really matter!). This latter figure sets the required performance of an oscillator if it's to be used in thd testing. That is, the sum of all the harmonics that come out of the oscillator should be about −100 dB or 0.001% of the fundamental. This is a pretty tall order but it seems a good place to start.

There are other factors that must be considered as well as the oscillator. So far I've only discussed the harmonics of the fundamental signal. As well as these any electronic circuit will generate electrical noise. This is a signal that is characterised by having components at all frequencies and sounds like the hiss you get from a tape recorder with a blank tape. Fields will also be picked up from mains operated equipment such as transformers and add in components usually at 50 Hz and 150 Hz. As the usual method to measure thd is to use a very narrow selective filter to remove the fundamental and then measure what's left, all these unwanted signals will degrade the measurement of distortion figures.

The range of frequencies to be covered by a useful oscillator is usually set by how much you want to pay for it. The lowest frequency is set by the control circuitry to stabilise the oscillator (more of this later) and for most purposes is set at 10 Hz (you can't hear this low anyway!). The upper frequency is set by the choice of operational amplifiers to be used. To keep costs under control I decided to base all designs around the old trusty NE5534 which sets an upper limit around 100 kHz. The last thing to be decided is the type of output the oscillator should have. The output level is also set by the choice of operational amplifier and limits output swing to ±12 volts peak.

If an op-amp output is brought directly to the output terminals, this enables people to do very bad things to the output circuitry so it's nicer to have some sort of series resistor in the output. In the industry this is normally chosen as 600 ohms (for a lot of reasons) and ensures that no matter how the output is mistreated (within reason) the oscillator can carry on. Another good reason to set the output impedance at 600 ohms is that then attenuators can be placed after the final amplifier. These will attenuate both the signal and any noise and distortion equally and thus will maintain the distortion performance of the oscillator at very low levels. This means that if you want to measure the performance of, say, a preamplifier then you can be sure the signal is still clean. To generate these low level signals I decided to include attenuators of 20, 40 and 60 dB or 10x, 100x and 1000x and allowed for a continuous adjustment between.

The last major decision to be made about the output was to decide whether it should be balanced or unbalanced. In a lot of the ▶

lder oscillators it was very common to have transformer in the output. This created /hat is known as a balanced output which as many advantages. The most important ne relates to earthing in an audio system see box) and makes an awful lot of prob- ems simply go away! For this reason alone ll broadcasting and many recording studios se *all* balanced circuits. The big disadvan- age is cost. Using today's design techniques neans transformers aren't necessary any nore but it still doubles the complexity of he output so I decided to settle for an un- alanced output.

Which oscillator circuit?

Oscillator circuits are as many and varied s brands of soap powder and all have their lusses and minuses. The first type that can e ruled out is any form of inductor based scillator. To build an LC oscillator that vorked at 10 Hz you'd need an inductor the ize of a brick that'd work just great as an ntenna to pick up mains hum. Clearly ome form of resistor capacitor oscillator nust be used. Probably the most common scillator configuration used is the Wien ridge (see Figure 1). This design goes way ack to before Hewlett-Packard made good se of it and is still used in designs today.

The Wien bridge serves to illustrate what s required for an oscillator to work. Refer- ing to Figure 1 you'll see that the oscillator onsists of an op-amp (actually any amplifier will do) with negative feedback around it to control the gain. The elements that form the frequency selective part of the oscillator are the two capacitors and the two variable resistors. If the attenuation from the output to the positive input is calculated then it can be shown that at a frequency equal to $1/2\pi RC$ the two resistors and capacitors divide the voltage at the output by exactly 3 and the voltage at the positive input is exactly in phase with the output voltage. At higher or lower frequencies the attenuation is greater and the two voltages are no longer in phase.

Suppose now that R_x and R_y are adjusted so the attenuation through them is also exactly 3. As they are only resistors the voltage on the negative input will also be exactly in phase with the output and under these conditions the whole network has infinite gain *but only at one frequency*. For frequencies greater or less than $1/2\pi RC$ the positive feedback drops away and the overall network gain becomes finite again.

If an amplifier configuration has infinite gain, it will oscillate as it takes no input signal to produce an output which is exactly what is wanted and what happens. If the attenuation through R_x and R_y is adjusted to be slightly less than 1/3 then the oscillation will stop; if it is slightly greater than 1/3 then the amplifier output will be driven hard up against the rails so the average gain over the entire swing is correct for oscillation. The gain has to be adjusted to be exactly correct for oscillation without clipping.

In Figure 1 the frequency adjusting resistors are conveniently shown as R where two resistors are being changed at the same time to set the frequency. It is assumed that the two resistors are always exactly equal. In the real world this is very hard to do but if you dig into the mathematics a bit you'll find that oscillation can still occur if they aren't equal. All that's necessary is to change the attenuation of the R_x/R_y leg of the bridge. Similarly if the two capacitors aren't equal then once again adjusting R_x and R_y will do the deed.

This is where practicalities start to intrude. If the two variable resistors don't track exactly then when you change frequency R_x or R_y must be adjusted automatically to set up the right conditions for oscillation again. As the amount of adjustment is (more or less) proportional to the mismatch in the variable resistors, a badly tracking dual gang pot will require considerable adjustment of R_y by electronic means and *all electronically variable resistors cause distortion!* (R_y is normally chosen as one end is connected to ground.) This is just one of the rules of the game. In order to build a very low distortion oscillator the two variable pots must track to within a per cent or so. You can buy them but you won't get much change out of $100. This just isn't a proposition for a cheap oscillator.

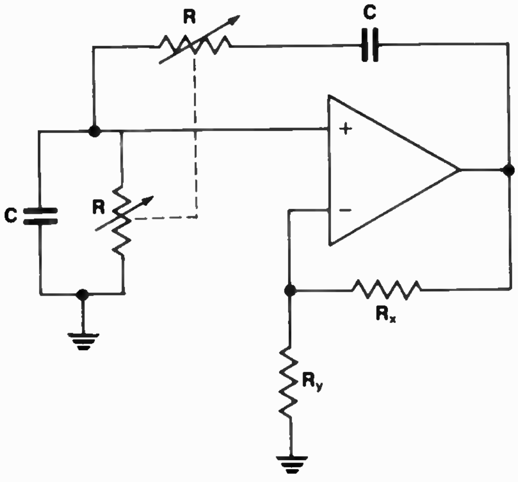


Figure 1. The Wien bridge.

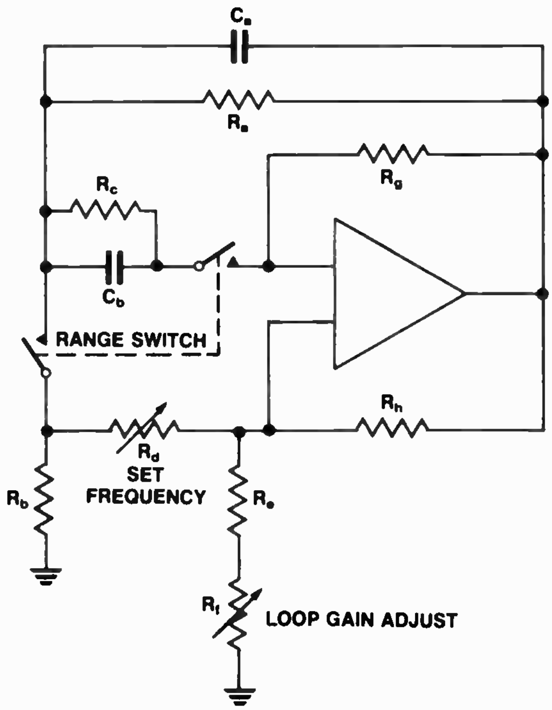
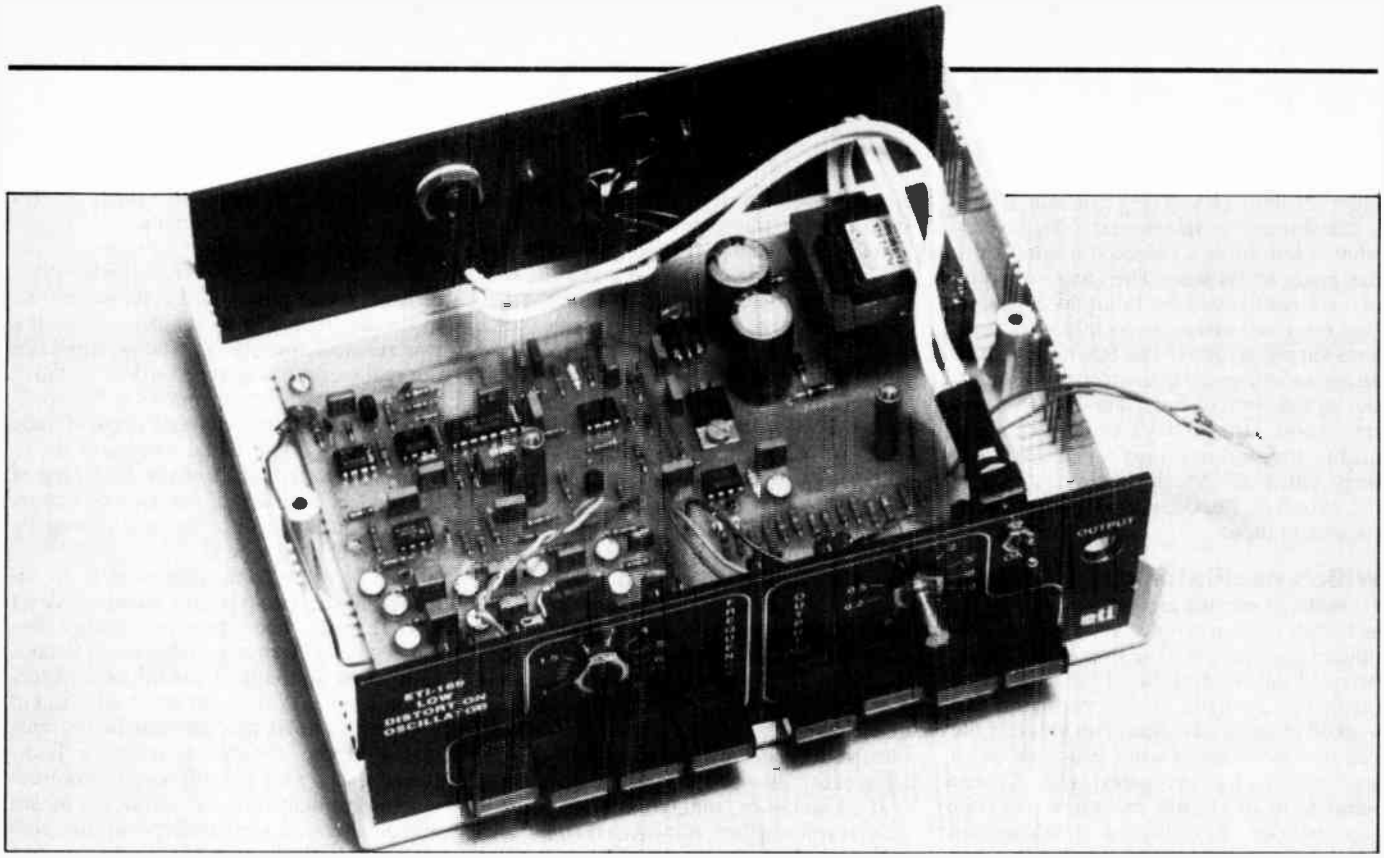


Figure 2. Network for oscillator circuit.



The capacitors also cause a similar problem as it is normal to only use the variable resistors to give a frequency range of 10 to 1 and then switch capacitors to switch ranges. In theory the same capacitors could be used for the entire range but in practice the impedances that would have to be driven at 10 Hz and 100 Hz become silly. Thus in the Wien bridge capacitors with tolerances of 1% are needed. Once again you can get them if you've got the dollars but I wanted this oscillator to be cheap (but, of course, superb!).

A little research into oscillator configurations showed that there has been a fair bit of work done on single control element oscillators. Much of it seems to have been done by Indian gentlemen with much brains but a research budget that wouldn't keep a mouse in cheese. This tends to restrict efforts to tinkering with exotic circuits on paper.

The type of circuit I was after was one where trimpots could be included to adjust out the tolerance variations in all the capacitors and, if possible, a ninth trimmer to adjust for the absolute value of the single gang frequency setting potentiometer (there are eight capacitors for the four frequency ranges). I couldn't find any configuration that was ideal but an article by V. Prem Pyara, S.C. Dutta Roy and S.S. Jamuar gave a method of finding a class of single control element oscillators so I could stir things around for myself.

The network I finally settled on is shown in Figure 2. Obviously it isn't anything like a Wien bridge but all the basic rules still apply. There is a condition that must be preserved in order that the circuit oscillate and if this condition is maintained then a simple resistance-frequency law can be established. Cranking through pages of mathematics I was able to derive the simple law for the condition for oscillation if $R_b = R_f$:

$$\frac{C_a}{C_b} = \frac{\frac{1}{R_h} - \left(\frac{1}{R_g} + \frac{1}{R_a}\right)}{\frac{1}{R_g} + \frac{1}{R_c}}$$

This may seem all a bit overwhelming but it tells one very important thing. R_d, the frequency control pot, *does not appear in the condition for oscillation* (bewdy!).

More thrashing about with algebra showed that if R_a, R_c and R_g were made adjustable then not only could the oscillation condition above be set up but the frequency of oscillation could be made equal to:

$$(2\pi f)^2 = \frac{2}{R_d R_g C_a C_b}$$

Thus by adjusting R_g it is possible to adjust out any tolerancing errors in the frequency set pot, R_d. Then by adjusting R_a and R_c tolerances in C_a and C_b can be adjusted out.

A quick rats' nest verified that all the mathematics told no lies and quite dramatic control of frequency could be obtained without affecting the oscillation amplitude. According to the numbers the frequency could be taken to infinity if R_d was made equal to 0, and this was very nearly what happened. The only limit was the bandwidth of the operational amplifier and the only problem that remained was the fact that the frequency is proportional to the square root of $1/R_d$. This makes the frequency scale very open for low frequencies and squeezed up at the upper end. However a bit of searching showed that I could get an inverse log law potentiometer from Allen Bradley in Sydney that more or less cancelled the nonlinearity caused by the square root law of the oscillator and gave a frequency-pot rotation law that was usable. This seemed to sort out all the problems with the oscillator itself. The last thing to be dealt with was the control of oscillator level.

In the final oscillator circuit R_h was chosen as the resistor to be varied to set oscillator loop gain. If one had the patience of Job and reflexes that would make a cat look sluggish then one could sit there and fiddle a trimpot to hold the loop gain steady but an automatic loop gain adjust is better. And that is where distortion mechanisms start to appear! R_h need only have about a ±2% adjust but this must be done by a control voltage. About the only voltage variable resistors that can be bought easily in Australia are FETs. If a field effect transistor has zero drain-source voltage applied to it, then it acts as a resistor whose value is varied by the gate bias. However if the applied ac signal becomes too large then it starts to produce distortion as the resistance of the channel is affected by the bias. This can be minimised by applying an ac component to the gate as well and I found that for the small range of resistance control needed it worked just fine.

A far worse problem than this is the possibility of components of the oscillator output voltage getting back to the control gate. The oscillator stabilisation loop consists of an output level detector and a control loop amplifier — loop filter to generate a control ▶

voltage for the FET. It is imperative that the output level detector produce as near as possible a dc voltage that is proportional to the ac out of tne oscillator. If any ac component is left in the detector output then it will modulate the FET and cause distortion. This dc control voltage must be generated for input frequencies between 10 Hz and 100 kHz which says that any form of rectifying and filtering is out of the question as the detector is part of a control loop.

The way I chose to do it is by using a peak detector which is reset every cycle of the oscillator output (see "How It Works"). This generates a dc voltage equal to the oscillator peak output voltage for a bit more than half the period of the oscillator. When the dc output is stable it is connected to the control integrator and when it is changing the integrator input is switched off. This very nicely removes any ac component from the control output and ensures that the FET only sees dc.

Like any control loop the oscillator control loop must be stabilised and this presents its own set of problems. The loop is stabilised by an extended RC circuit in the integrator that compares the dc from the peak detector with a reference dc set up by a zener diode. This makes sure that the control loop gain and phase are right over the entire 10 Hz to 100 kHz range. All this may seem a little complicated when compared to some other techniques you may have seen (such as a light bulb in the feedback loop!) but this level control circuit is the essence of producing low distortion signals. The oscillator circuit alone, if left to itself and operated away from clipping, will produce almost no distortion (NE5534s are very good) and in practical circuits almost all the distortion is added in by the stabilisation loop. This stabilisation loop solves that problem almost completely. It is only necessary to adjust the capacitor compensation trimmers so the FET operates in the best region to get distortion performance that was so low I couldn't measure it.

The output circuit and attenuators presented little difficulty as the switch bank I chose to use (you can get it from either Jaycar or Geoff Wood Electronics) has two contact changeovers. This let me use a 'T' type switching configuration to minimise capacitive hopover. The problem is that if the −60 dB attenuator is selected then signal from the higher outputs is capacitively coupled to the output and you don't get −60 dB at 100 kHz (see Figure 3). Using a 'T' configuration shorts out all the capacitive hopover and gives the correct level. ●

Details of the construction, circuit, overlay and parts will be given in the November issue of ETI.

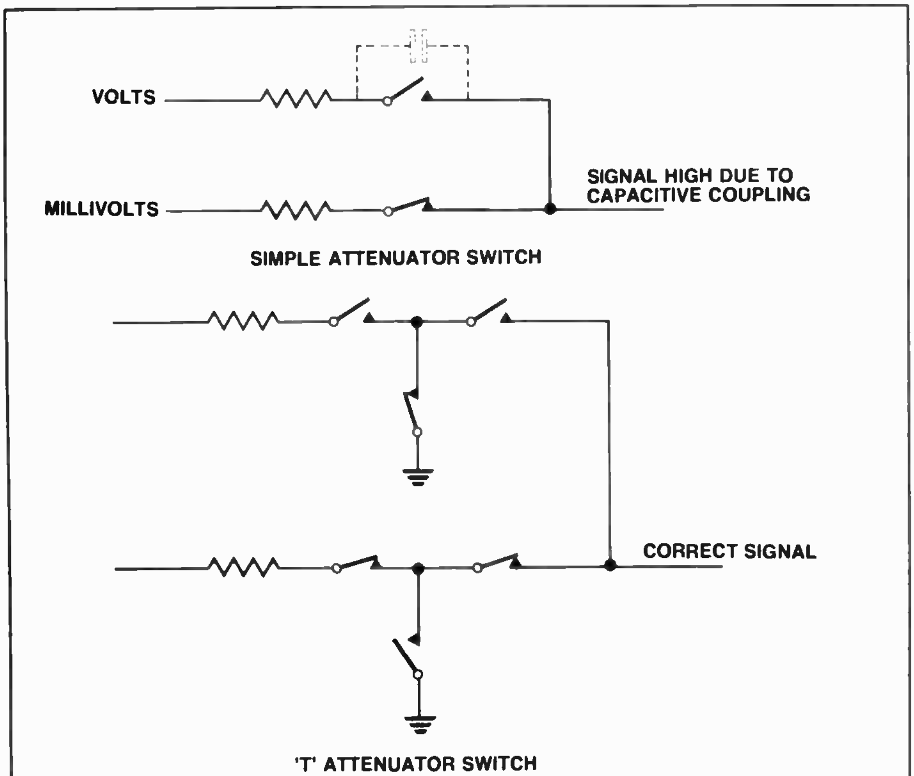


Figure 3. 'T' attenuator switch.

Part 2

LOW DISTORTION AUDIO OSCILLATOR

Last month we told you the principles and compromises in designing this impressive audio oscillator. It was such a comprehensive discussion we had to split the project into two parts. Here we resume with a guide to construction, circuit details and testing advice.

Ian Thomas

SO MUCH FOR all the reasons for the eventual circuit last month. Now for how to use it.

Construction

I chose to use a plastic instrument case available from Geoff Wood, Jaycar or Altronics. I think just about every electronics store will stock one that's OK. The plastic boxes are neater but the metal ones do make the oscillator less prone to hum pickup. You pays your money and makes your choice!

If you want to do your own layout and make the board fit a different case, the most important thing of all is to keep the RC parts of the oscillator *as far as possible from the power transformer*. Power transformers radiate 50 Hz fields and these are entirely too easy to get into the oscillator. If you look at the layout of the board, you'll see that they are on opposite diagonal corners of the board and I still had some trouble. For the same reason (hum pickup) I chose to use the miniature Bourns trimpots to minimise board area used by the selective components in the oscillator. These are also available from Geoff Wood and were, in fact, a lot cheaper than the normal cermet trimmers. On the subject of trimmers it's *absolutely essential* that good quality cermet trimpots be used as the cruddy old carbon film ones really aren't stable enough. All the trimpots are in frequency determining networks and if an older type is used, the output frequency would be all over the place like a dog's breakfast. Similar remarks apply to all the fixed resistors. Ideally they should be ±1% metal film resistors not so much for the ±1% as for the stability with time and temperature although in the prototype they were mainly ±2% and seemed to work OK.

The capacitors used in the frequency determining networks are all metallised polyester and are encapsulated in plastic cases. The types that're suitable are ERO (Roederstein) type MKT1817 or MKT1826 or Wima type PR21 or RS21 or some fair dinkum equivalent. If you do decide to go for a rock bottom budget unit and use greencaps let me know how they work (by mail — not in person!). The point about the capacitors I specified is that they have a known temperature characteristic which only gives a ±0.5% change over the normal operating temperature range.

The safest way to get the board layout is to copy the one given or buy a mask from ETI (if you put any value on your time this is really far cheaper). The board is made from single sided 1/16 inch epoxy glass board. I don't recommend using paper phenolic type boards as their leakage performance is a bit sus. If you are doing your own layout take particular care to get the spacing right for the switch banks pins (they're *@#!! not on 0.1 inch centres).

Etch and drill the board normally and start assembly with the power supply. Assemble only the power transformer, rectifier, diodes, filter electrolytics and voltage regulators. I VERY strongly recommend using printed circuit terminals to get 240 volts onto the board as I have been bitten far too many times.

NEXT COVER ALL THE TRACKS THAT CARRY MAINS VOLTAGE WITH INSULATING TAPE. If you don't you'll probably kill yourself. Also there are two solder pins that carry mains that stick out from the side of the little Ferguson transformer. Tape them up too. I missed them and my CRO earth clip brushed across them on the prototype (BANG!! palpitate!).

Next create for yourself a death machine — that is a length of 2 or 3 core flex with a mains plug on one end and the other end with bare wires. Connect the wires to the input terminals on the board then power the beast up. Check that you have plus and minus 15 volts coming out of the regulators. If all is well remove the plug from the mains, remove the bare ends from the terminals and lock the damn think away in a safe or something. Now that you know that the voltage regulators work you can carry on with the rest of the assembly with some confidence that the whole thing won't go up in flames when you turn it on.

As always make sure that all the ICs are in the right way and also the diodes. It's particularly important that the leads be neat and short around the oscillator (mains hum again). In the prototype I used two 1M and two 200k resistors to make up the 1.2M needed for the −60 dB attenuator as this gives the exact value. There are three wire jumpers that run just behind the frequency select switch which should be as short and straight as possible — resistor leads that've been cropped off do just fine. The last components that should go in are the two switch banks.

Testing and alignment

Before you try to mount the oscillator in the box it's a good idea to do a preliminary alignment and make sure all the ranges are working. Once the board is completely assembled reconnect the death machine to the power terminals and turn the unit on. When I do this to a brand new board I normally run my hands lightly over the components for 30 seconds or so to see if anything is getting too hot too fast (that's another reason to check the power supply first). Next check that the supply rails are still sitting at ±15 volts. If this is right then you're ready for the fun bit.

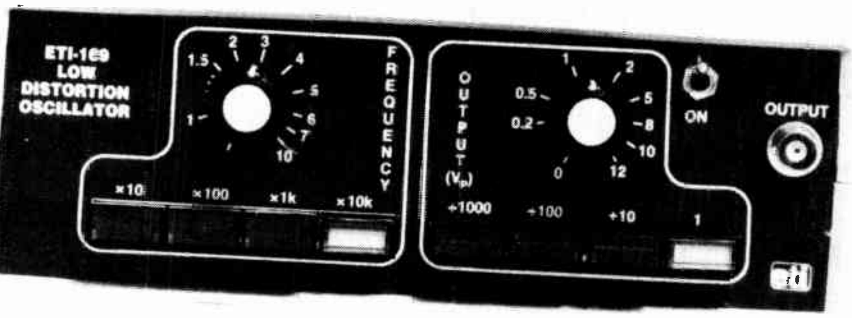
To set up the pots you really need a counter although a CRO will do in a pinch. The purpose of this adjustment is to remove the ±10% or so tolerances in the oscillator components and it can't be done properly if you can only measure frequency to ±20%. You can certainly make sure that everything is working but the frequency scale on the front panel won't be worth much.

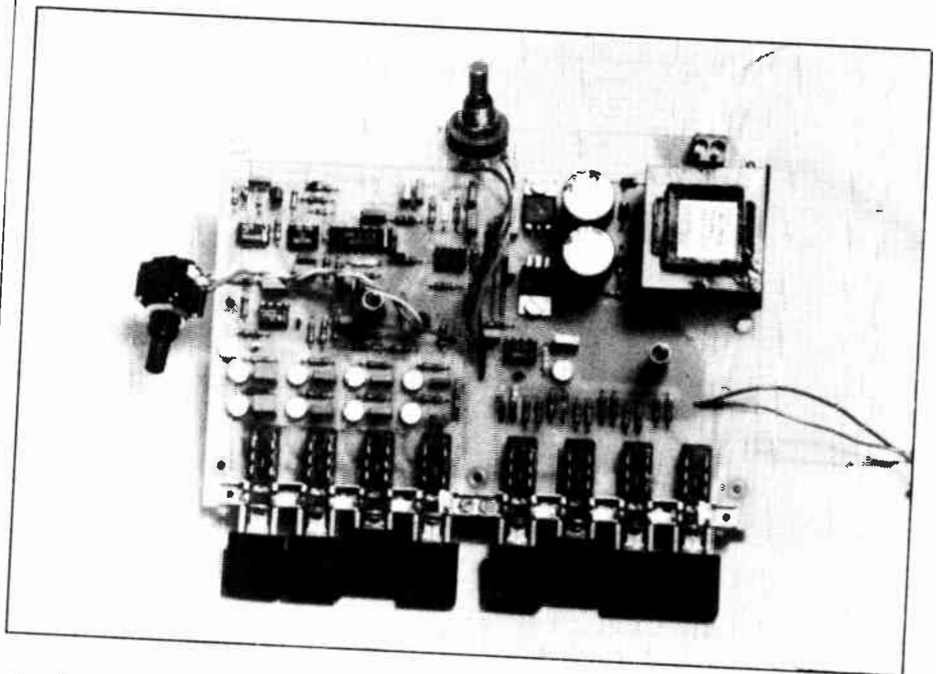
Given that you've got, begged, borrowed or whatever a counter with a 1 second timebase at least (10 seconds is better) connect it to the output and select the 100 to 1000 Hz frequency range and the 0 dB attenuation range. Connect the frequency adjustment potentiometer temporarily to its appropriate holes and tack a piece of wire across the level adjust pot holes so you get full output. Turn all nine trimpots full clockwise and turn the beast on. If all is normal the output should go up against the rails and stay there. Set RV10 (the frequency select pot) for minimum resistance or maximum frequency then start winding RV9 anticlockwise while monitoring the output. Bring it back to about mid position then start winding RV3 counterclockwise. Somewhere around mid position oscillation should start and it should be at around 1000 Hz. If this is OK, everything is looking good and you can start checking out the level control circuitry.

First check that the squarer IC2 is working. Pin 6 of IC2 should have a square wave on it which is in opposite phase to the sine wave input. The output should show no sign of oscillation during transitions. The input to the squarer on pin 2 should show the input sine wave during positive half cycles and should be diode clamped to about −0.7 volts during negative excursions. The output from the squarer should appear on pin 10 of IC5, the CD4053. C14 and R25 should differentiate the squarer output and a very narrow negative going pulse should appear on pin 9. Finally pin 11 of IC5 should be sitting near ground. Momentarily stop the squarer by shorting its output to ground and pin 11 should rise to about 12 volts if you're using a 10M input impedance CRO probe.

Next comes the peak detector. Check that the sine wave output from the oscillator appears on pin 3 of IC3. Next check the signal on the negative input pin 2. It should be sitting at a dc voltage equal to the peak positive swing of the oscillator with a negative step occurring every time the input signal crosses zero volts going positive. If the oscillator is not being properly level controlled due to other problems then the peak detector output may not be able to get as far positive as the oscillator input but the step should still appear every time pin 9 of IC5 is pulsed negative. If this is what you get it's time to proceed to the integrator IC4.

Integrater IC4 is just a simple inverting operational amplifier with a rather messy RC network in the feedback circuit. Check that pin 3 has +6.8 volts on it and that pin 1 of IC5 has the peak detector output on it. As the analogue switch in IC5 is turned off when the peak detector is reset the negative





step doesn't appear on pin 1. If the oscillator output level isn't being controlled and is swinging up against the rails then the output of the integrator IC4 pin 1 should be far negative and the output of the inverting amplifier following the integrator should be hard positive. If the level control circuitry is working (that is if the oscillator has been adjusted so it has enough control range) the integrator output will probably be sitting at a few volts positive, and the output of the inverting amplifier, a few volts negative.

Last of all check that the control voltage on the gate terminal of the FET is at one half the voltage of the inverting amplifier, and the control circuitry should be OK. If the oscillator output is giving bursts of oscillation there is probably a short in the RC network around the integrator. As a final check on the level control try adjusting RV3 again until the level control circuit can take over. You should see a beautiful clean sine wave with a 6.8 volt peak swing on the oscillator output. Pretty — isn't it?

Now back to setting up the oscillator frequency. When the level control is working it's a lot easier to adjust the oscillator as monitoring the control voltage out of pin 7 of IC4 tells how the adjustment is going. Connect a voltmeter there and adjust RV9 until it reads about −4 volts. The next step is to adjust R7 until the oscillator output frequency is 1100 Hz and the control voltage is at −3 volts. To do this, it will be necessary to adjust both RV3 and RV4 to keep the control voltage where it's wanted. You will find that rotating RV3 clockwise lowers the control voltage, and rotating RV4 clockwise raises it. Leave RV3 in the mid position and start adjusting R4 clockwise to raise the control voltage, adjusting RV9 counterclockwise to raise the output frequency until it reads 1100 Hz. Repeat until the correct frequency is obtained with a control voltage of −3 volts.

Next set the frequency pot to lowest (highest resistance). Leave RV9 alone and adjust both RV3 and RV4 until the output frequency is 90 Hz and the control voltage is steady at −2 volts. You will find that rotating RV3 counterclockwise lowers the frequency and raises the control voltage. Rotating RV4 counterclockwise raises both the voltage and frequency. It's easy to adjust one against the other to get the required result. Return now to the maximum frequency setting on RV10 and check that it is still 1100 Hz. If necessary readjust RV9 to the correct 1100 Hz. Return to the lowest frequency setting and reset RV3 and RV4 for 90 Hz. Repeat until it's perfect, but it should only take two or three tries. That's the first range setup completed and you can move on to the 900-11,000 Hz range. ▶

For starters adjust both RV5 and RV6 to mid position and select the lowest frequency setting on RV10. Adjust RV5 and RV6 until the oscillator starts; then adjust exactly as before except that you want 900 Hz. When this is right adjust RV10 to maximum frequency and check that it reads 11,000 Hz. It may not be exact and if the difference is too big for you to live with try adjusting RV9 for correct 11,000 Hz then repeating the realignment on the 100-1000 Hz range.

Exactly the same alignment process must then be done on the 9-110 Hz range. The control voltage will take a little longer to get there. The highest frequency range is a little different in that the control voltage should be adjusted for −1 volt at 9000 Hz and will go down to about −4 to −5 volts at 110,000 Hz. Once you've slogged through this lot you should have a pretty good idea of what you've built.

Finally check that the attenuators are working correctly (each step down should give an output voltage one tenth the step before) and the board is ready to go in the box. Assuming you've used the same box as I did, proceed as follows.

First locate the board in the box and note the plastic mounting pillars that will actually support the board. Next attack the box bottom with a large pair of side cutters and remove the pillars that aren't wanted. There must be about 15 pillars there and only seven are needed. Next carefully mark off exactly where the holes are needed to allow the switch banks to come through the front panel and cut the holes. As this is the front panel it pays to take a little care: if you make a mess of it your mistake will be staring at you for years!

Mark off where you want the frequency and level pots to come through and drill the holes. The same applies to the power switch and output terminals. Assemble all the components on to the front panel then slide it into the box bottom. Solder lengths of wire into the board suitable for connecting the level and frequency pots. At this stage in the assembly of the prototype, a major problem reared its ugly head.

When the board was powered up in the box with all the mains wiring in place, I found that a *large* amount of mains hum was ▶

PARTS LIST — ETI-169

Resistors	all 0.4 W, 2% unless noted
R1 to 9	5k1
R10, 13	750R
R11	6k2
R12	5k8
R14	1k5
R15, 21, 53	1k0
R16, 17, 26, 28	1M0
R18	220R
R19, 20	100k
R22, 23	33R
R24	5R1
R25	10k
R27	2M2
R29	220k
R30	330k
R31	470k
R32, 33	4k7
R34	100R
R35	3k3
R36, 37, 48, 49	1k2
R38, 39	12k
R40	680R
R41	33k
R42, 43	120k
R44	620R
R45	27k
R46	1M0
R47	1M5
R50, 51, 52	10R
RV2, 4, 5, 8, 9, 13	2k0 trimpot cermet*
RV1, 3, 5, 7	5k0 trimpot cermet*
RV10	100k pot freq set Allen Bradley 72J1N056S104B
R11	100k trimpot cermet*
RV12	2k linear law pot level set

* *Bourns type*

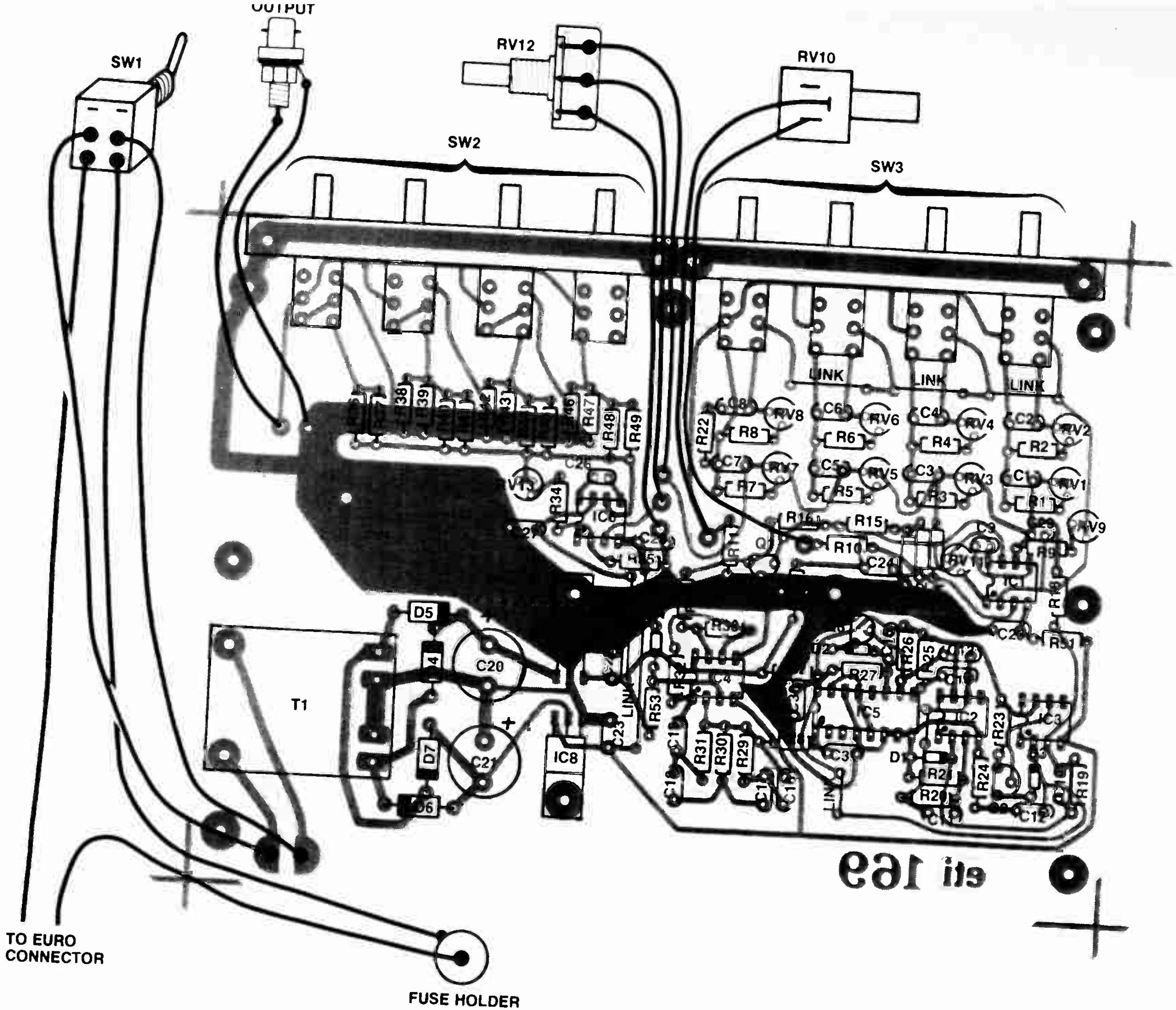
Capacitors	
C1, 2	1µ ±10% 50 V met poly*
C3, 4, 13, 15	100n ±10% 63 V met poly*
C5, 6, 17, 31	10n ±10% 63 V met poly*
C7, 8	1n ±10% 63 V met poly*
C9, 24	10p ±5% ceramic plate
C10	47p ±5% ceramic plate
C11	22p ±5% ceramic plate
C12, 18	47n ±10% 63 V met poly*
C14	220p ±5% ceramic plate
C16	2n2 ±10% 63 V met poly*
C19	220n ±10% 63 V met poly*
C20, 21	470µ 25 V RB electro
C22, 23, 24, 25, 27, 28, 30	100n ceramic monolythic
C26	33p ±5% ceramic plate

* *see text for types suitable*

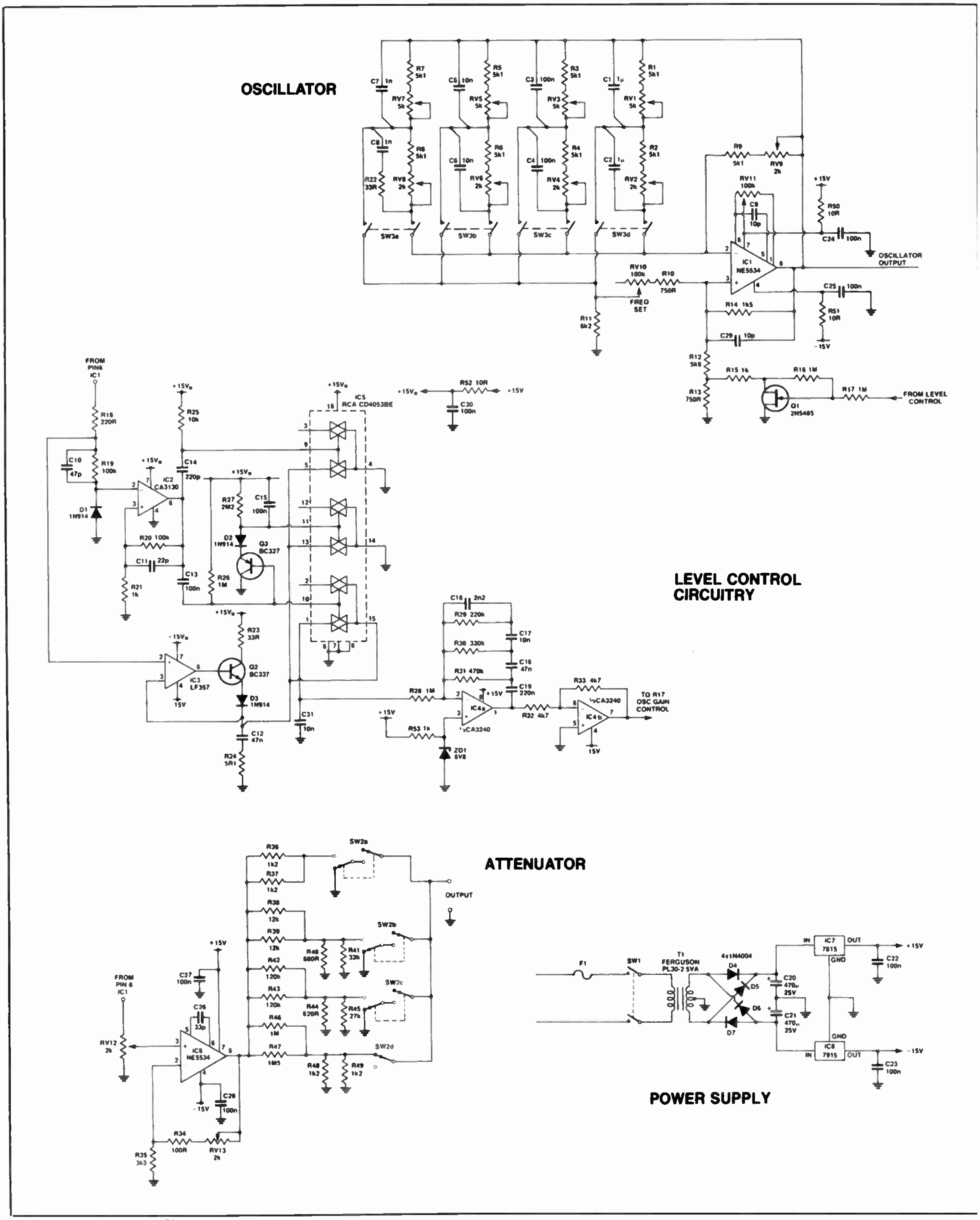
Semiconductors	
IC1, 8	NE5534
IC2	RCA CA 3130
IC3	LF357
IC4	RCA CA 3240
IC5	RCA CD4053BE
IC7	7815 + 15 V reg
IC8	7915 − 15 V reg
Q1	2N5485
Q2	BC337
Q3	BC327
D1, 2, 3	1N914
D4, 5, 6, 7	1N4004
ZD1	BZX79 C6V8
Miscellaneous	
T1	Ferguson PL30/2.5 VA
SW1	DPDT mains toggle switch
SW2, 3	4-way pc mount switch bank and knobs
F1	350 mA 2AG fuse

ETI-169 pc board; 90 x 60 mm piece sheet copper; 120 x 170 mm sheet aluminium; Euro connector; 2AG fuse holder; 2 knobs for pots; 2 x 25 mm pc board spacer tapped at both ends; BNC panel mount socket; 200 x 160 x 70 mm plastic instrument case; Scotchcal front label; mains hookup wire; light hookup wire; assorted nuts and bolts.

Price estimate: $99.50



SW1
OUTPUT
RV12
RV10
SW2
SW3
LINK
LINK
LINK
T1
D5
D7
C20
C21
IC8
TO EURO CONNECTOR
FUSE HOLDER


OSCILLATOR
LEVEL CONTROL CIRCUITRY
ATTENUATOR
POWER SUPPLY

HOW IT WORKS — ETI-169

The audio oscillator can be separated into four main areas:
(1) the power supply;
(2) the oscillator;
(3) the output buffer and attenuator; and
(4) the level control circuitry which may be further broken down into
 (a) the peak detector;
 (b) the squarer; and
 (c) the integrator — loop filter.

The power supply is a conventional mains operated ±15 volt integrated regulator. The transformer TR1 is mounted on the printed circuit board and has a centre tapped secondary with outputs of ±15 volts ac. With this brand of transformer the secondary is designed to give rated voltage under full load so in the oscillator the voltages on the two filter capacitors C20 and C21 are at about ±24 volts. The current to charge C20 and C21 is provided from the full wave rectifier bridge D5 to D8. ICs 7 and 8 regulate the output from the filter capacitors to provide a clean ±15 volt supply for the oscillator.

The oscillator is based on an NE5534 operational amplifier with both positive and negative feedback. SW1, the frequency band switch, selects one of four RC networks to provide frequency dependent negative feedback through two resistors and two capacitors. The two resistors associated with each of the four networks are made adjustable to make up for the fact that only 10% tolerance capacitors are used. Direct negative feedback is also provided via R9 and RV9, and RV9 allows compensation for the fact that the frequency select potentiometer, RV10, is also only ±10% tolerance.

Negative feedback to preserve the correct conditions for oscillation is provided via R14. R12, R13, R15 and the level control FET, Q1, also form part of the negative feedback network. R16 and R17 ensure that the voltage on the gate of the FET is exactly one half of the gate-source voltage. This minimises distortion introduced by the FET and results in predominantly third order distortion.

RV10 is the frequency adjust pot and by varying its value from 100k to 0 ohms a frequency range of 0.9 to 11 can be achieved. As this frequency variation is proportional to the square root of the resistance it is necessary to use an inverse log law pot to avoid the frequency change being bunched up at one end of the pot rotation.

Both C24 in the negative feedback path and R34 in the highest frequency band select network are to compensate for stray capacitances around the loop and ensure that the correct conditions for oscillation are maintained.

The level control circuitry is fed from the output of the oscillator via R18 which is included to provide some isolation of high frequency spikes generated in the level control circuit. The oscillator output is squared by the squarer IC2 and its associated components. R20, R21 and C11 provide positive feedback for the squarer, which is really only an op-amp operating open loop. As IC2 and the circuitry it drives is CMOS it is necessary to only power it from +15 volts and ground. The op-amp will not tolerate large negative swings on its input under these conditions so R19 and D1 clip the negative swing at the op-amp negative input. C10 provides phase correction at high frequencies.

The output of the squarer drives a CMOS analogue switch CD4053. This switch has three sections, one of which is used to switch the error signal to the loop integrator and the other two are used to control the peak detector reset function.

The peak detector is formed by IC3 and its associated circuitry. IC3 is used as a non-inverting voltage follower with a transistor-diode in the feedback path. To understand how the peak detector works, first consider that C12 has no charge on it. This means that the voltage on the negative input of IC3 is zero volts. As the voltage on the positive input is taken positive by the oscillator output, the output of IC3 will also go positive turning on Q2 and forcing charge into C12 via diode D4. D4 is only included as the reverse bias breakdown of a transistor emitter base junction is normally only 6 or 7 volts and in theory could be left out (in practice it can't). Thus feedback is provided around the op-amp through the transistor and the negative input is held at the same voltage as the positive input. In the process C12 is charged to the same voltage as the positive input. R24 is included in series with C12 so the op-amp doesn't have to drive a purely capacitive load which makes the loop unstable.

When the input from the oscillator reaches its peak value and starts to swing negative again the transistor-diode prevents charge being removed from the capacitor and C12 is left charged to exactly the peak value of the oscillator output. The capacitor is left charged while the oscillator output swings through the complete negative part of its cycle and starts to come positive again. When it passes through zero volts positive going the squarer output IC2 pin 6 switches negative. This sharp edge is differentiated through C14 and R25 to produce a narrow negative spike on the input of one of the analogue switches. This turns on the analogue switch IC5 pins 4 and 5 and partially discharges the peak detector capacitor C12 ready for the next peak detection cycle.

The resultant voltage on the peak detector is thus a dc voltage equal to the peak ac value of the oscillator output with small negative steps every time the oscillator output crosses zero positive going.

A second section of the analogue switch is driven directly from the squarer output such that pins 1 and 15 are on during the negative half of the oscillator output and off during the positive half. Thus when the peak detector output is stable the analogue switch is on and when it is being reset it is off.

One problem that exists with this type of level control circuit is that it is possible for the oscillator to stop running because the gain control is set too low. If the peak detector has been charged to a high voltage by a transient then, as the oscillator is not running, no reset pulse is generated from the squarer output and the oscillator cannot start. To prevent this C15 is continually discharged to ground by the squarer output if it is running by Q3 and D2.

If the squarer stops running or fails to start then R27 charges C15 to the positive rail and turns on the analogue switch pins 13 and 14. This completely discharges the peak detector. At the same time the squarer input to the analogue switch (pins 1 and 15) is ac coupled through C13 and R26 and thus under no signal conditions IC5 (pins 1 and 15) is turned hard on. This ensures that the oscillator will always get a 'kick start' if it fails to start normally.

The loop integrator is formed by one half of IC4, a MOS input op-amp. The input to the integrator is from pin 1 of the analogue switch which is at the peak detector output voltage for half the time and open circuit for the other half. The positive input of the op-amp is set to a reference voltage by the zener diode, ZD1, biased by R29.

A rather complex feedback network is formed around the op-amp by capacitors C16, C17, C18, C19 and resistors R29, R30 and R31. All these components are necessary to preserve the correct gain/phase conditions for all oscillator frequencies and conditions. The most important feature of the network is that C19 has no resistor in parallel with it and hence for low frequencies the amplifier acts as a pure integrator. This means that given sufficient time the dc voltage at the peak detector output will be adjusted by the loop to be exactly equal to the reference voltage no matter how other conditions in the loop vary.

The output of the loop integrator is inverted by the second half of IC4 with resistors R32 and R33. This is because the FET, Q1, requires an increasing dc voltage to reduce the gain and preserves the correct dc operating conditions.

The output buffer amplifier is formed by IC6 and is a simple non-inverting amplifier with adjustable gain so the correct output level can be set. RV12 is the level adjust potentiometer on the front panel which allows the oscillator output to be adjusted from maximum to nothing and RV13 in the feedback is the fine adjust to preset the amplifier gain. The output of IC6 drives three 'L' pad attenuators and a series resistor in parallel to generate four outputs, all of which have a 600 ohm output impedance and output levels of 0, −20, −40 and −60 dB referred to the output of IC6. One of these four outputs is selected by SW2 and connected to the output terminals. SW2 is connected in a 'T' configuration with the centre of the off switches connected to ground to minimise capacitive coupling of the higher level outputs to the low level output.

---

being coupled into the output. There were three ways that this could occur. The first was that there was hum on the power supply lines but as the crud in the output was 50 Hz this tended to rule it out. Any unfiltered and regulated grot on the rails would be at 100 Hz (full wave rectifier bridge). Also a quick check showed the rails were as pure as the driven slush.

The next way was through some capacitive coupling from the mains area to the oscillator circuit. As the problem seemed to occur only for the lowest frequencies where the frequency set pot has its highest resistance, this seemed most likely. Capacitive coupling is a high impedance effect which is, praise be, easy to stop. By holding my hands around the oscillator I found that I could snuff the coupling with ease. It appeared that I was getting capacitive hopover from the mains wiring to and from the ▶

switch and the high impedance oscillator circuit. Annoying but not disastrous. If it had been stray magnetic fields from the transformer it would have meant deep trouble; an electrostatic field will be stopped by almost anything but a magnetic field takes kilos of iron.

To fix it I cut a piece of sheet metal to fit in the bottom of the box so it covered the complete area under the oscillator. This must include the area under the switch bank right up to the front panel. It should extend from the centre of the box right to the left hand edge. I then completely removed what was left of the unwanted mounting pillars with a hammer and sharp wood chisel and fitted the screen to the case. I connected a piece of wire to the screen and brought it out so I could earth it and reassembled the oscillator. About a 10 dB improvement but still not nearly good enough!

I found that holding my hands over the oscillator part of the circuit still had a very marked effect so a screen above the circuit was needed too. I mounted two 1" spacers off the board in grounded areas and cut a piece of aluminium so it completely covered the top of the board, with a notch out of one corner to clear the transformer and filter capacitors. When I screwed it down there was considerable improvement but still not good enough.

If a piece of metal was held down the left hand side of the box near the oscillator frequency select elements, it helped a bit so I bent up a small bracket to protrude down 25 mm from the top screen, extending from the front edge of the screen to the centre of the side where there is a major mounting pillar. This was easily attached with two of the self tappers that hold the board down. Another test and almost good enough!

After a few more magic gestures with my hands the last culprit was found. The field was coupling into the frequency pot itself which is visible above the screen. A bit more quick work with sheet aluminium and a little box was screwed in place around the pot. In case you have some trouble getting the sheet aluminium, I used a cheap oven baking tray that was on special for a dollar — it seemed to be pretty good stuff too! This completed the screening and got the hum down to acceptable levels. Even if you chose to use a metal instrument case it would still be necessary to do the top part of the screening as the mains wiring and the oscillator are inside the same box.

After you've gone through all the hassles of making screens as above it should be a snip to screw the board in place. Wire up the frequency select pot *using as short a wire as possible*, the output level pot and the mains wiring to the switch. The mains connection on the rear panel was through an IEC socket so if some oaf trips over the mains cord it just pulls the plug rather than ripping the cord out by the roots. Both the mains IEC connector and fuse are mounted off the rear panel and holes have to be cut for them. It's a good idea to cut the fuseholder hole just the right size and cut the locating notch in the side so it cannot rotate. *All mains wiring connections should be securely made by looping the bared end of the wire through the terminal tag then soldering it.* All connections should be completely insulated as you'll be the one to cop it if they aren't. The final result should look pretty much as shown in the photo. If this is all OK, remove the top screen and the unit's ready for final test.

More alignment and testing

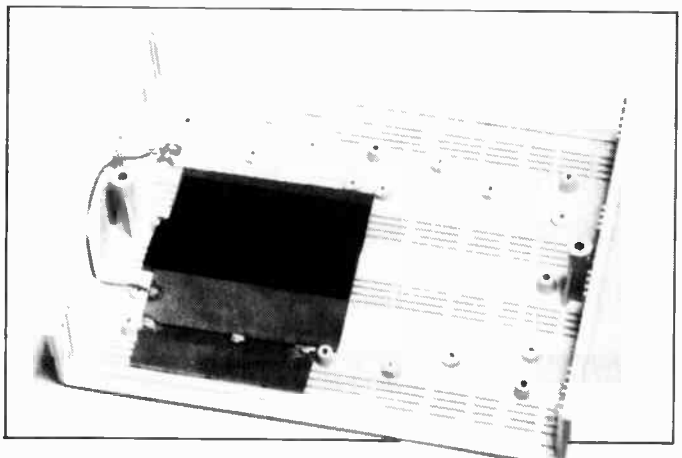
Final alignment follows almost exactly the same routine as laid out in the preliminary testing except that it's a bit easier as the oscillator is working. You'll probably notice things need a bit of tweaking as the stray capacitances have changed. If the FET control voltage is allowed to go much further than −2 volts then the third harmonic distortion will start to rise. It will also be noticed that when the frequency pot is rotated from one extreme to the other a dc offset will appear in the output. To remove this adjust RV11 next to the oscillator op-amp until there is no discernible shift over the entire range.

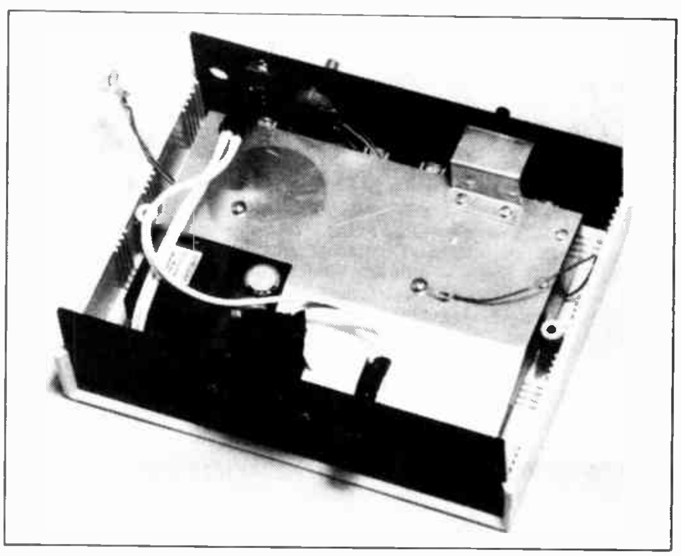
The only adjustment left to be made is to set the output level. To a certain extent this is a matter of choosing the units you like to work in. My choice was to terminate the output in 600 ohms then set the level pot so the maximum level out was +12 dBm or just a shade over 3 Vrms. 0 dBm is a very commonly used reference level in the audio industry and is exactly 1 mW into 600 ohms or, in volts 0.775 Vrms. The oscillator is capable of putting out up to 4 Vrms or +16 dBm.

At this stage the top screen can be screwed into place and the small cover screwed over the pot. It has to be put on afterwards but I imagine you've discovered that already! Screw on the top cover, attach knobs to the two pot shafts and the unit is ready to have the front panel marked. To calibrate the frequency pot set the knob on the pot so the knob pointer's rotation is symmetrical about the vertical axis, then set for far counterclockwise. Select the 900-11,000 Hz range and connect the output to your counter. The counter should read 900 Hz give or take. Mark this point with a pencil and label it. Then proceed up the scale in 1000 Hz increments marking and labelling each point. You will probably want finer calibration points between 1000 and about 4000 Hz but above that the scale starts to close up. The number of points you plot is largely a matter of taste and how you want the front panel to look.

Exactly the same procedure is followed for the level pot except that a level meter (or, at a pinch, an oscilloscope) is connected. If you're calibrating in dBm don't forget to terminate the output in 600 ohms. You can then permanently mark the front panel with an engraving tool or dry transfer lettering to give the unit a completed look. The switch banks also need to be labelled in some way too. When this is all done you've added another weapon to your arsenal of electronic weaponry! ●

Below. Bottom of the box showing the piece of sheet metal covering the area beneath the oscillator. Insulation tape is used to prevent shorting of tracks. **Right.** Aluminium completely covering the top of the board with a small bracket protruding down from the top screen stops the electrostatic field. (View from rear of box.)





ETI-185: VERSI-PLY

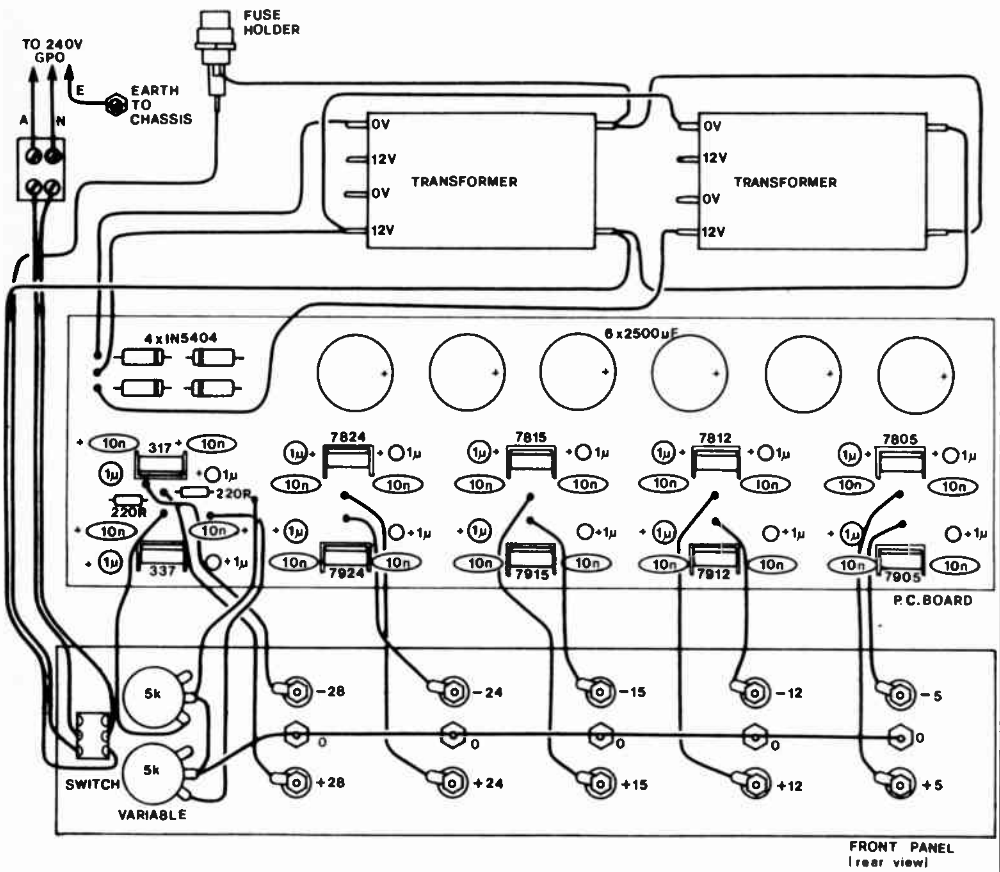
multiple voltage power supply

A power supply that has ten voltages available simultaneously.

MEETING THE VOLTAGE needs of modern electronic devices often poses more than a minor inconvenience for technicians and experimenters.

Having individual voltage rails available to power such things as TTL, CMOS, ECL, MOSFET, and relay coils can cause problems — especially if a breadboard type project is on the bench and requires all voltages simultaneously!

All Electronic Components have recognised this problem, and have designed this power supply to be simple, economical, and as versatile as possible — without placing it in the "upmarket" category.



This project was designed by and is available as a kit from All Electronic Components, Lonsdale St, Melbourne Vic for $257.21, (03) 662-3506.

The Versi-Ply provides eight popular fixed voltages of +5 V, −5 V, +12 V, −12 V, +15 V, −15 V, +24 V, −24 V; and two variable voltage range of +1.3 to +28 V, and −1.3 to −28 V. These ranges should be capable of supplying almost any voltage requirement.

The project is just about as simple as its possible to make it, and exploits to the full features built into modern regulator ICs. They display some attractive characteristics; in fact a veritable wish list of power supply features. For instance: built in thermal shutdown, over voltage protection and current foldback. They virtually guarantee distruction proof operation. Indeed, they are fully short circuit proof, so in the event of an accidental short on the output, they should still survive.

They are also capable of supplying currents of up to eight amps in short bursts. This is more than adequate for most of the requirements of the typical electronic work bench. The result is that we have been able to eliminate the series pass transistors and their associated protection and control circuitry which adds so much complexity and expense to the typical power supply.

The performance of the Versi-Ply has come up to our expectations. Maximum ripple is of the order of the 1.8 Vdc on the supply rails, and each can draw a maximum load of 1.2 amps. The two positive and negative busses are supplies by Ferguson PL24/40VA type transformers. After rectification and filtering to get rid of the ripple they produce a rail at 32.8 Vdc positive and negative.

The bus thus provides more than the required three volts drop across the 24 volt regulator, which is the worst case situation. The TO220 three terminal regulators used here require their input voltages to be three volts higher than the output. This additional three volts is known as the dropout voltage, and is the minimum input voltage the device needs to regulate.

Of course, as is usual in designing electronic circuits, there is no such thing as a free lunch. Even simplicity has a price. In our case, remember that the maximum current available per bus is 1.2 amps. The safe current output on each output voltage depends on the voltage input (V_{in}) minus

ETI-185 PARTS LIST

Resistors
R1, R2 .... 220 ohm CR37
RV1, RV2 .... 5K Linear Potentiometers

Capacitors
C1-C20 .... 10nf 100 V Ceramic
C21-C30 .... 1µF 35 V Tantalum
C31-C40 .... 1µF 50 V RBLL Electrolytic
C41-C46 .... 2500µF 80 V RP Electrolytic

Semiconductors
D1-D4 .... 1N5406 Silicon Diodes
IC1 .... 7905 TO-220 Voltage Regulator
IC2 .... 7912 TO-220 Voltage Regulator
IC3 .... 7915 TO-220 Voltage Regulator
IC4 .... 7924 TO-220 Voltage Regulator
IC5 .... LM337T TO-220 Voltage Regulator
IC6 .... 7805 TO-220 Voltage Regulator
IC7 .... 7812 TO-220 Voltage Regulator
IC8 .... 7815 TO-220 Voltage Regulator
IC9 .... 7824 TO-220 Voltage Regulator
IC10 .... LM317T TO-220 Voltage Regulator

Miscellaneous
Two PL24/40VA Transformers, C1063 Case, 10 Green 4mm Binding Posts, 10 Red 4mm Binding Posts, 10 Black 4mm Binding Posts, DPDT Toggle Switch, AEC 86-10-1 pc Board (Tinned), Silk-Screened Aluminium front panel, ½" Plastic Spacers, 18 ½" x ⅛" Bolts, 4 1" x ⅛" Bolts, 23 ⅛" Hex Nuts, Solder, Hook-up Wire, Strain Relief Bush, mains cord, cable and plug, 10 only TO-220 Vertical Mount Heatsinks.

the output voltage (V_{out}) multiplied by the current (I), and should not exceed 15 watts power dissipation (P_{dis}). The current (I) available on each voltage rail can be calculated from:

$$V_{in} - V_{out} = V_{diff}$$

and:

$$P_{dis} = V_{diff} \times I$$

so that:

$$P_{dis} = (V_{in} - V_{out}) \times I$$

and therefore:

$$I = \frac{P_{dis}}{(V_{in} - V_{out})}$$

P_{dis} should not exceed 15 watts, or the TO-220 package regulators will overheat. Standard parameters for this circuit are: V_{in} = 32.8 volts and P_{dis} (max) = 15 watts.

The implication is that the low voltage. regulators have a very large voltage across them, and in consequence will be limited in the current they can supply. For instance, the five volt regulator can supply a maximum current (I) of:

$$I = \frac{15}{32.8-5} = 540\text{ mA}$$

while for the 24 Volt regulator the picture is much better:

$$I = \frac{15}{32.8-24} = 1.7\text{ A}$$

indicating that the input will be limited by the current supply in the bus, not the regulator. The number of occasions where more than 500 mA will be required at 5 V will be few and far between so these limitations do not undermine the usefulness of the device. And even in the rare case where one might have a requirement for more than half an amp, the variable supply based on the 317 could be pressed into service to double the available current.

Construction:

Construction is straightforward. Check the pc board for bridges, links, undrilled holes and non-continuous tracks. Mount the small capacitors first, observing polarity requirements. Then insert resistors and flying leads for the pots and output terminals. The bridge diodes and large electrolytic capacitors may now be mounted and soldered. Once again, observe polarity. It will be necessary to clip off the No Contact (NC) pin on the capacitors.

Taking standard over-head precautions (heatsink clamps, or the burnt finger test!), install the voltage regulators. Visually check the board for dry joins and bridges.

Care must be taken when fitting and wiring the transformers, because if they are wired out of phase, the ensuing explosion will be both costly and smelly! Check the diagram carefully.

Cross refer the board against the diagram overlay, and if satisfied, connec ac power and check all output voltages. If all is not correct, switch off immediately, and begin to recheck your work. Should fault finding be required, be sure to discharge the capacitor banks by shorting with a low value resistor.

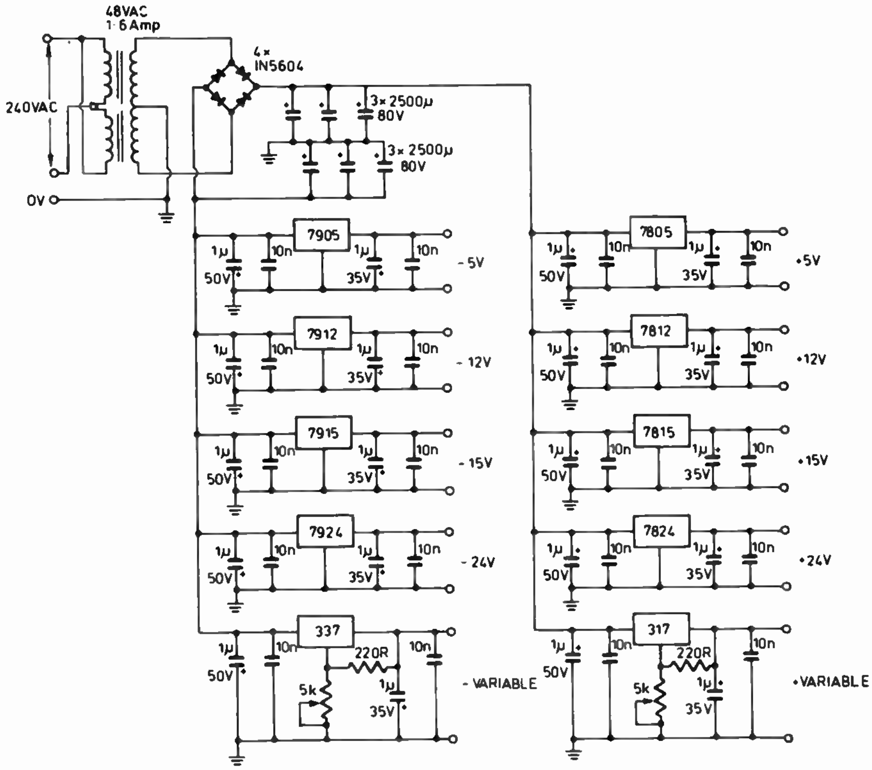
ETI-185 — HOW IT WORKS

The two transformers have the secondaries in series to give the required 32.8-0-32.8 Vdc after rectification. Transformer output before rectification is 24 Vac at 1.6 Amps per transformer.

The input voltage for the regulators is taken from the positive or negative rail, so that each works independently of the others. Filter capacitors are used for stability in the supply. The bank is of 2500µF electrolytics have a very low internal resistance, and also the necessary filtering to give a continuous, uninterrupted supply.

Due to the wide range of load impedances that will be encountered the inputs and outputs of the regulators are bypassed with 10nF Ceramics and 1µF Tantalums (or RBLL low leakage electrolytics). This ensures stable regulation, and stops self oscillation. These bypass capacitors are located as close as possible to the regulators to minimise the risk of rf interference entering the device. It is also for this reason that a double-sided circuit board is used.

On the positive and negative rail, approximately 1.2 Amps is available — however, the regulators' outputs should not exceed 1 Amp. It should also be remembered that a maximum 15 watts dissipation is allowable on these devices; that is, at 5 Volts output, the current drawn should not exceed 0.5 Amps. The full 1 Amp can be drawn from approximately 17 Volts.



The variable regulators (LM317 and LM337) work by maintaining a constant 5 V between their output and Common pins. The voltage between output and ground can thus be controlled by devising a voltage divider with R1 and VR1. A constant current of (5/220) = 23 mA flows through the 220R resistor, and a linear voltage drop will thus be created across VR1.

OP-AMP POWER SUPPLY

Robert Irwin

An ideal supply for op-amp experimenters and those with solderless breadboards. The ETI-251 provides ±12 V rails at 1 A and solves those 'split rail blues'.

A DUAL RAIL supply is a handy piece of equipment for anyone who is even thinking of playing around with analogue ICs. The ETI-251 is a simple, easy-to-build, low cost supply that will be ideal for breadboarding up circuits which require single or split 12 V rails. The ETI-251 provides regulated positive and negative 12 V rails and can supply up to 0.5 A from each. An overload LED on each rail gives a visual indication when you try to draw too much current from the supply. All the components used are very common and most could probably be found in the average hobbyist's 'bits-and-pieces' draw. The supply is relatively easy to build and should be suitable for even inexperienced constructors, although not recommended as a very first project. The construction section has been made very detailed to accommodate any beginners who wish to build this supply.

Design details

The circuit is designed around the very widely used LM7XXX series of three terminal regulators. The LM7812 and LM7912 provide +12 V and −12 V respectively. Both ICs have built in short circuit foldback current limiting and thermal protection and are therefore very hardy devices. As well as the internal protection built in to the regulator ICs, several external protection diodes are included in the circuit to guard against any accidental load faults that may otherwise destroy the regulators.

The transformer used is a widely avail-

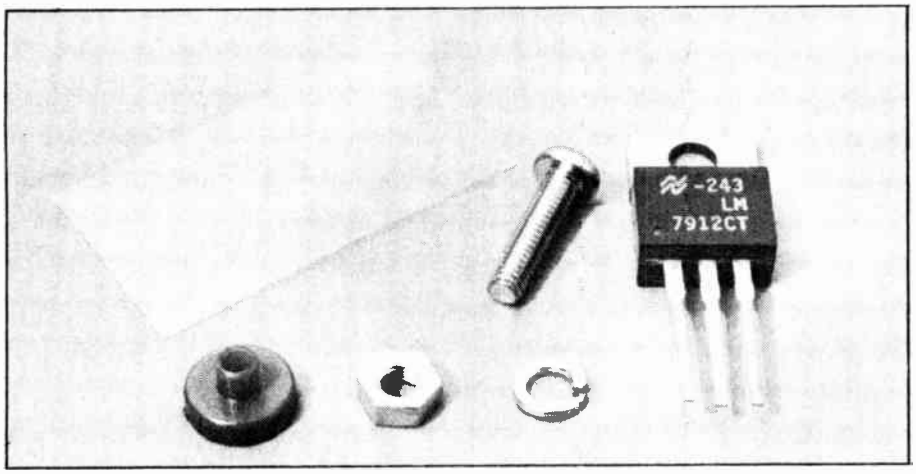
HOW IT WORKS — ETI-251

The circuit is very simple as the three terminal regulators are basically autonomous, requiring no external circuitry to make them work. Protection from unusual load conditions is needed though. Referring to the circuit diagram, the transformer output is 30 Vac with a centre tap. This gives two 15 Vac signals which are rectified by a D1, 2, 3, 4 which form a full wave bridge rectifier. This produces both a positive and a negative rectified output with reference to the centre tap. The rectified signals are smoothed by C1 and C2 which, with the values chosen, will give a peak output of 21 V and a ripple of about 4 V p-p at 1 A output.

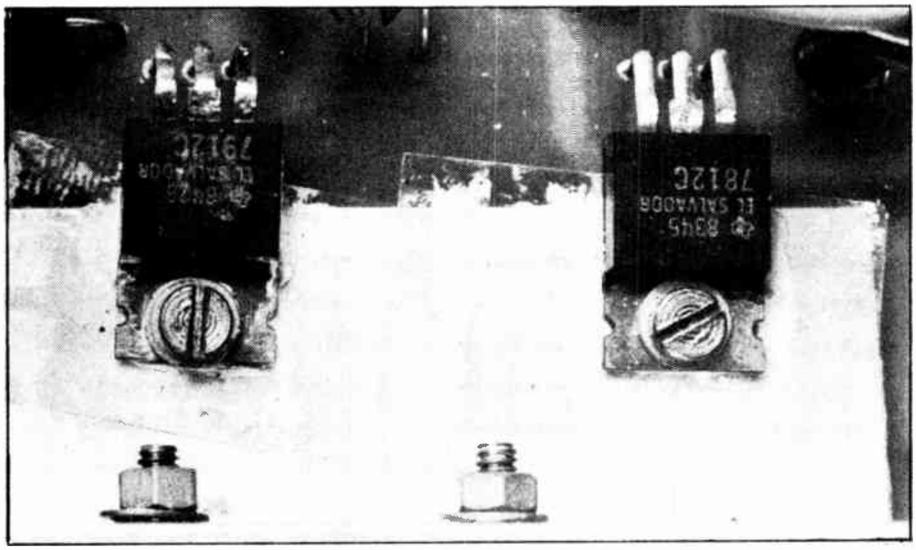
Before going to the regulators, the current is monitored by the overcurrent circuitry. Both the positive and negative circuits are identical (except for the direction of current flow) so we will just look at the positive overcurrent circuit. R1 is in series with the supply current and will develop a voltage across it which is given by Ohm's law, V=IR. The emitter of Q1 is connected to the supply side of R1 and the base is connected to the load side. When the load current reaches 0.6 amps, 0.6 V is developed between the base and emitter of Q1 and it begins to turn on. This will allow current to flow through LED2 which will cause it to light and indicate an overcurrent condition. R5 limits the current through the LED to about 15 mA.

IC1 is a positive regulator which takes the unregulated input and gives a regulated +12 V output. IC2 does the same for the negative side and gives a regulated −12 V output. One problem that sometimes occurs when using regulators in a split rail supply is that of start up under a common load. The negative regulator tends to establish itself first and, under a shared load (particularly a capacitive one), it may drag the output of the positive regulator negative and prevent it starting up. It may also cause the IC to be destroyed. To help prevent this, R3 and D7 are incorporated into the design. This helps the positive regulator start up under common loads by providing isolation of the common pins on the two ICs and, if the output is dragged negative, the current can be shunted by D7 and allow the positive regulator to establish itself. R4 is included in the circuit to maintain a voltage balance between IC1 and IC2. Without R4, the quiescent current in the common terminal (about 6 mA) would raise the output voltage on the positive regulator slightly.

As an added precaution against destruction of the regulators, D8 and D9 prevent any reverse polarity voltages from developing on the outputs of the regulators. D5 and D6 will protect the regulators from any overvoltages on the outputs which may occur when reactive loads are being driven. C6 and C7 are not crucial but provide some filtering to the input of the regulators. C3 and C4 improve the transient response of the regulators and prevent high frequency instabilities. C5 ac couples the power supply earth to the chassis which allows the metal case to act as an electrostatic shield and prevent any rf inteference in the supply. There should be no dc connection between chassis and power supply earth. LED1 and R7 provide a power on indication.



Above: IC mounting hardware. This pic shows the various bits and pieces you will need to mount each regulator. The mica washer is at the top left, and the insulating washer is at the bottom left.
Below: The regulators mounted on the heatsink bracket. The 'messy white stuff' all over the bracket is the thermal grease used to provide good heat transfer.



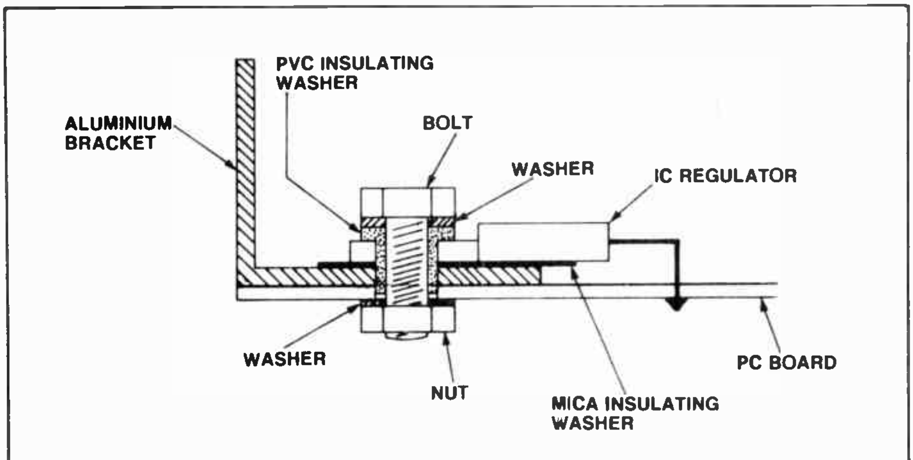


Figure 1. When mounting the IC regulators to the heatsink bracket, make sure the tabs on the regulators are electrically insulated from one another. This is done by placing a PVC insulating washer around the bolt and a mica insulating washer between the IC and the bracket.

able multi-tapped secondary type which provides 15 V and 30 V taps and is rated at 30 VA maximum. The main reason for the choice of this transformer is that it is cheap and easy to get. It should be noted however, that under a direct short circuit between the positive and negative terminals, the output current will be just over 2 A. The transformer will handle this sort of overload for quite a few minutes without damage but may heat up if the short is left for long periods. This will not be a problem in normal operation but if the supply were, for example, to be used to power a circuit which was to be left running overnight, it would be a good idea to use a transformer with a higher output current rating so that any sustained short that may occur will not thermally stress the transformer. A short from either the positive or negative to ground will only cause about an amp to flow and can be handled indefinitely by the specified transformer. A PL30/60 VA is an ideal substitute but is quite expensive. For most applications, though, the specified transformer will be more than adequate.

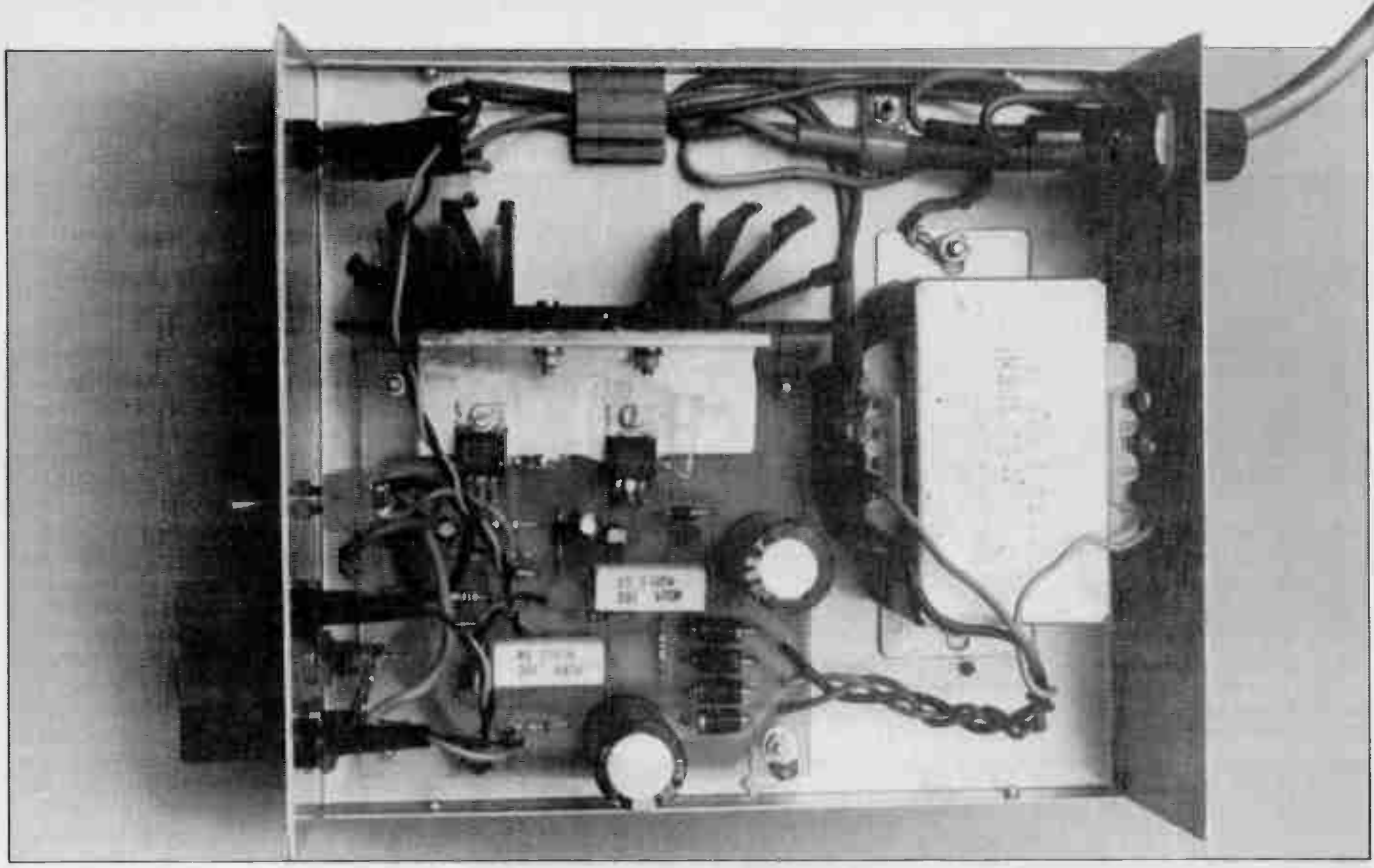
Incorporated into the design is an overload indication for each rail. This is set to indicate that a current of 0.6 A or more is being drawn from the rail. This overload in no way damages the supply but is there to indicate excessive current drain which may be cause to investigate the circuit you are powering for shorts.

Construction

The construction of the power supply presents no real problems. However, I will describe the construction in detail for those who may not be as familiar with a soldering iron as others.

It is recommended that the circuit be constructed on the ETI pc board. If you wish (and you know what you're doing) you can, of course, use Veroboard or the like but using the pc board will greatly simplify construction and minimize the chance of a wiring error. Having said that, once you get a pc board check it very carefully for faults on the copper side. The most common faults are broken or shorted tracks caused by problems in the etching stage. If a track is thin, then over-etching or faults in the resist can cause the track to be etched away in parts and thus be open circuit, so check all thin tracks for breaks. Where two tracks come close together, under-etching or dust on the negative can cause the copper between the tracks to remain and thus cause a short so check all points where tracks come close together. Finally, check that all the holes have been drilled. Once you are satisfied that the pc board is in good shape, you can move on to the soldering in of the components.

Referring to the overlay diagram, locate the position of the wire link. This link



should be made on the pc board with a piece of tinned copper wire (a discarded piece of component lead is ideal). Once this is in position locate and solder in all the resistors. R1 and R2 are high power resistors and in the course of normal operation may get quite hot. To help cooling and to prevent scorching the board these two resistors should be mounted so that they stand off the board by about a millimetre or so. The parts list specifies 1 ohm, 5 watt resistors for R1 and R2 but two 2.2 ohm, 1 watt resistors in parallel can be substituted for each resistor if you wish. There are extra holes on the board to allow for this.

Next, locate and solder in all the capacitors. Take very special note of the way these are put in as they are polarized and may be destroyed rather spectacularly if they are put in the wrong way round. Note that C5 is only soldered to the board at one end. The other lead will be bolted to the pc board mounting bolt at a later stage. For the moment just leave it dangling over the side.

The semiconductors can be soldered next. Start with the diodes. Mount the large rectifier diodes off the board as you did with the power resistors as these too may get hot. Once again pay attention to the way the diodes go in as they are also polarized components.

The two transistors can now be soldered in. The only remaining components to mount are the IC regulators. These will be a bit of a problem in that they mount on an aluminium heatsinking bracket but must be electrically isolated from one another. The first thing to do is to prepare the bracket.

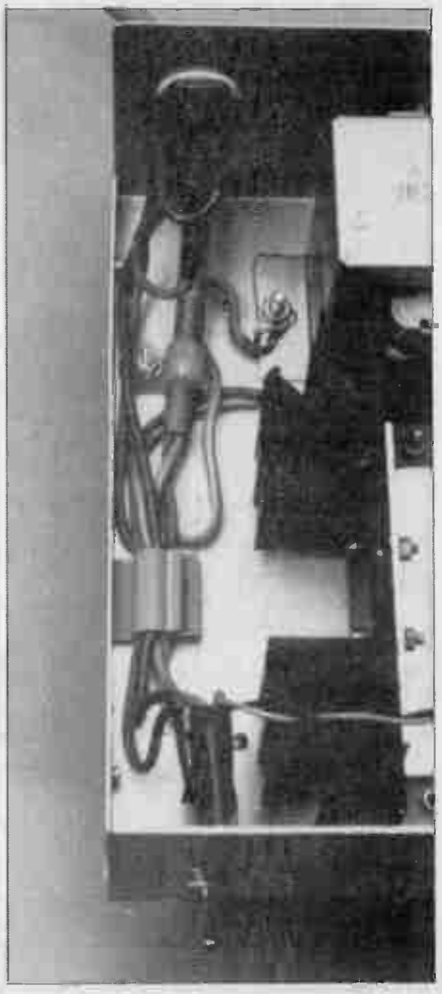
Cut a 70 mm length of 1 inch aluminium

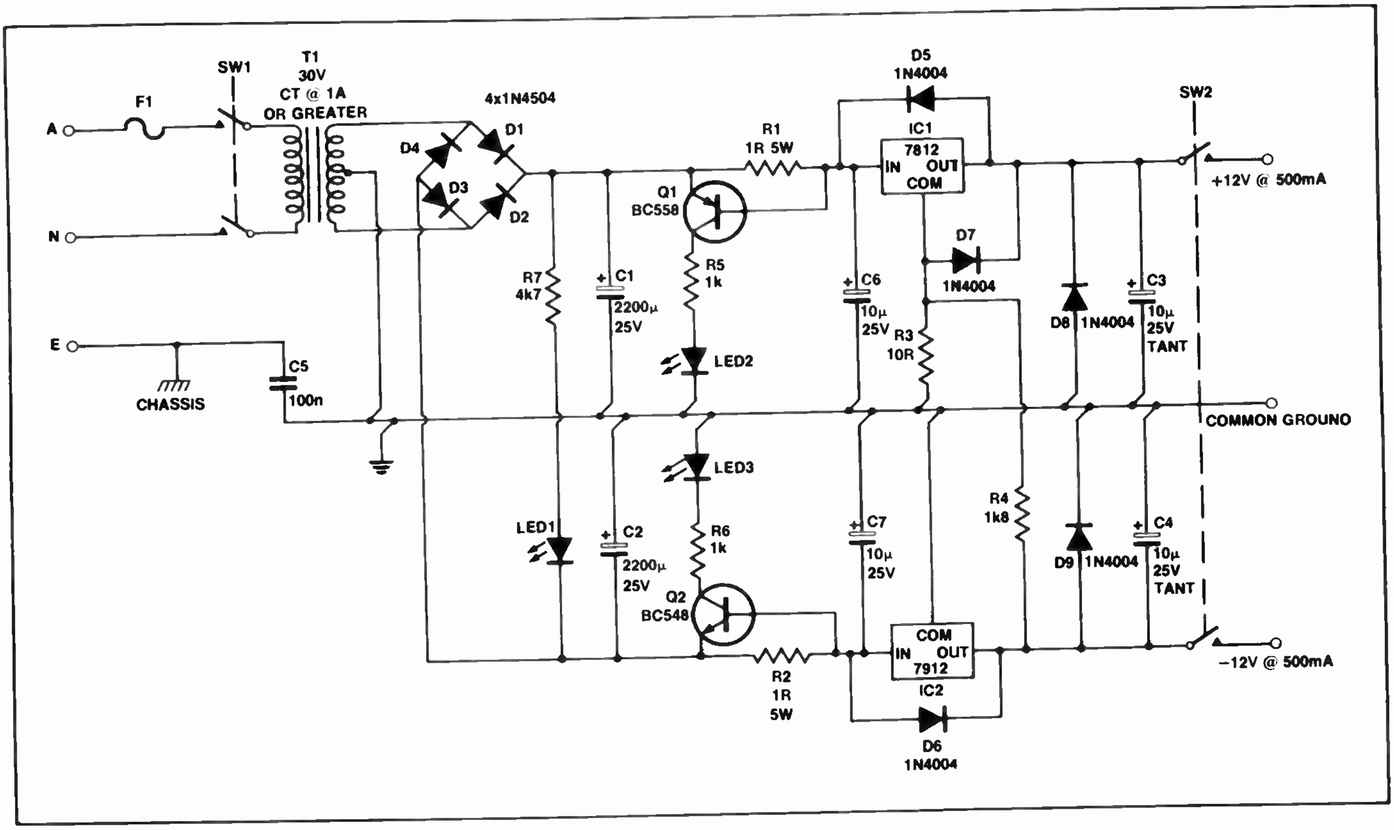
Above: The general layout inside the case. Make sure that the transformer and heatsink are clear of one another.

Right: The mains wiring is run down the left hand side of the box. Try to keep it neat and insulate all connectors on the fuse and switch so that you can't accidentally brush against a live terminal.

angle. Study the photographs and drawings and position the angle at the edge of the board so that it sits between the pc board mounting holes on the component side. The side should point upwards from the board. Use a felt pen or scribe to mark the position of the two mounting holes for the ICs. These holes should be drilled large enough to fit the IC mounting bolts. The heatsink (a 35 mm length of radial fin type) mounts vertically on the side of the aluminium angle. This should be positioned against the angle and the centres for two holes to mount the heatsink should be marked. It is best to drill the holes in the heatsink first and then use these to mark and drill the holes in the angle. This way they'll line up. All the holes in both the bracket and heatsink should be carefully de-burred and the edges made smooth. Thermal conduction to the heatsink is dependent on how well contact is made between the two surfaces. If the edge of the hole is rough it may prevent the two surfaces from contacting each other properly.

To mount the two ICs examine Figure 1 carefully. Position the bracket and lay the ICs on it. Bend the legs of the ICs in the appropriate place and push them through their mounting holes. Take a TO220 package mica mounting washer and thinly coat it on both sides with a layer of thermally conduc-





PARTS LIST — ETI-251

Resistors ....all ¼ W, 5% unless noted
- R1, 2 ....1R0, 5 watt
- R3 ....10R
- R4 ....1k8
- R5, 6 ....1k
- R7 ....4k7

Capacitors
- C1, 2 ....2200μ, 25 V, RB electro
- C3, 4, 6, 7 ....10μ, 25 V tantalum
- C5 ....100n ceramic bypass

Semiconductors
- IC1 ....LM7812
- IC2 ....LM7912
- D1, 2, 3, 4 ....1N5404
- D5, 6, 7, 8, 9 ....1N4004
- LED1, 2, 3 ....5 mm red LED

Miscellaneous
- T1 ....30 V CT, 1 A sec (Arista type A-6672 or similar)
- SW1, 2 ....DPDT toggle
- F1 ....500 mA 2AG fuse

Fuse holder; 185x70x160 mm metal cabinet (DS-H-2744 or similar); mains flex and plug; mains clamping grommet; 3 x 5 mm LED mounting grommets; ETI-251 pc board; Scotchcal front panel; 70 mm length of 1 inch aluminium angle; 2 x TO220 mounting kits (mica washer, insulating washer and nuts and bolts); 3 x 4 mm banana socket binding posts (red, black and green); 35 mm length of radial fin heatsink; heavy duty hookup wire; 4 x 6 mm stand-off pc board spacers; 4 x pc board mounting nuts and bolts; 4 x 15 mm 4BA nuts and bolts; 2 x solder lugs; silicon thermal grease.

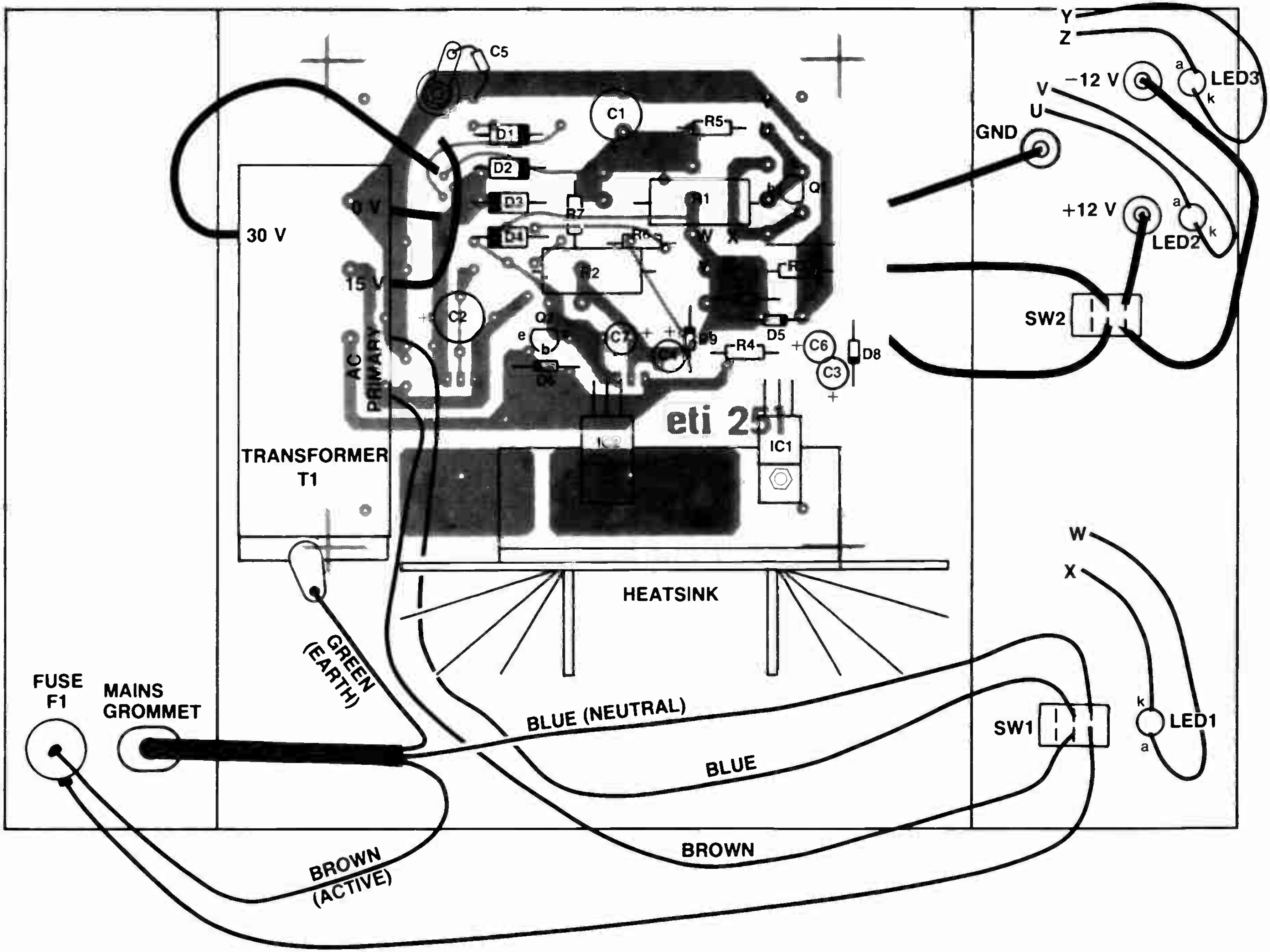
Price estimate: $45-$50

tive silicon grease. This will ensure good contact from the IC to the bracket. Put the mica washer in place over one of the mounting holes on the bracket and position the hole in the IC over the mounting hole. The bolt and washers can then be put through the hole and tightened up. Note that the bolt is isolated from the metal tab on the IC by a PVC insulating washer specially made for the TO220 package. Repeat the procedure for the other regulator.

To test whether the regulators are isolated from the heatsink, measure the resistance from the centre leg of one IC to the center leg of the other with a multimeter. You should get a high resistance (open circuit) reading. If not then take the ICs off and try again with fresh mica washers. After the ICs have been mounted correctly you can solder the legs to the pc board. The heatsink can now be mounted to the bracket. Use silicon grease to ensure good contact and screw the bolts up fairly tightly.

The next step is to solder flying leads to the input, output and LED mounting points on the pc board. These should be colour coded so you can easily identify the positive, negative or ground when you come to wire the LEDs or terminals. Light hookup wire is adequate for the LEDs but heavier wire (5 amp or more) should be used on the ac input and dc output points. If the pc board is placed temporarily in position in the case, you can get an idea of the length of wire needed. Always allow a bit more length than you think you'll need as the wire can be trimmed later. Once all the wires are attached the pc board is completed and you can turn your attention to preparing the case.

The prototype was housed in an inexpensive aluminium instrument case (see parts list for details). Take the case apart and don't lose the screws. If you examine the photographs you can get an idea of the general layout of the inside of the case. Looking from the front, the pc board is mounted on the right-hand side at the front. The transformer mounts directly behind this. The mains cord and wiring is all on the left-hand side. Position the transformer and pc board in the box and, after you have ensured that there will be no possible shorting between transformer and the case or the transformer and pc board, mark the positions of the mounting holes for the transformer and board, and drill the holes to fit a 6BA bolt. Holes for the mains cord grommet and the fuse should be marked and drilled in the back panel. If necessary, drill a hole in the floor of the box for a mains cord clamp. To mark out the front panel you can either use the drilling diagram or you can use the front panel artwork as a template and centrepunch the holes from this. Drill the front panel holes to the appropriate sizes. Remove any burrs from the holes and smooth any rough edges. ▶

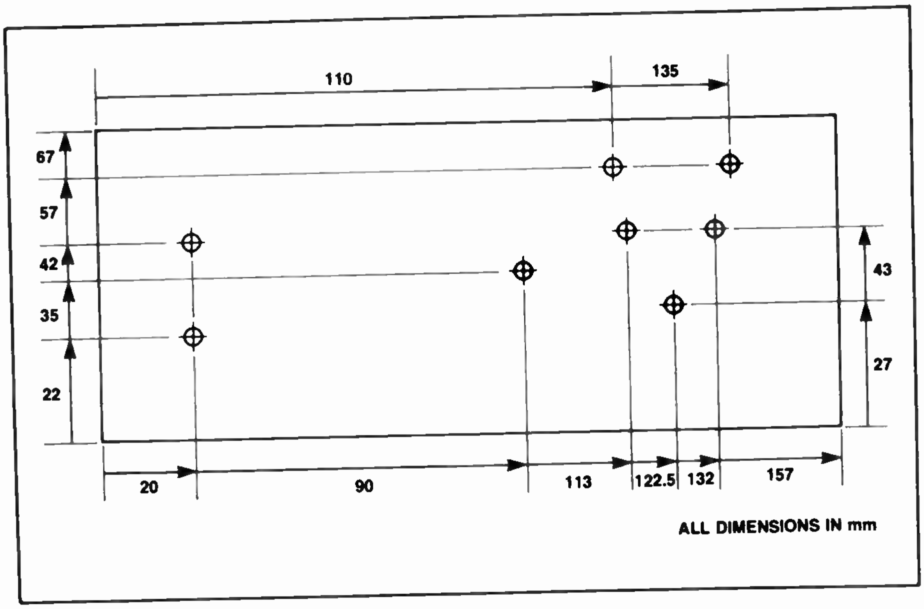

C5
C1
R5
D1
D2
D3
D4
R7
R6
R1
Q1
W
X
R2
R3
0 V
15 V
30 V
AC PRIMARY
C2
Q2
e
b
C7
C4
R9
R4
D5
C6
C3
D8
D6
IC2
IC1
eti 251
TRANSFORMER T1
HEATSINK
Y
Z
−12 V
LED3
a
k
V
U
GND
+12 V
LED2
SW2
W
X
k
LED1
a
SW1
FUSE F1
MAINS GROMMET
GREEN (EARTH)
BLUE (NEUTRAL)
BLUE
BROWN
BROWN (ACTIVE)

The next step is to attach the Scotchcal label to the front panel if it is needed. Firstly drill small pilot holes at the centres of all the mounting holes. Do a trial fit to see how the pilot holes line up with the holes in the front panel. If they are OK then peel off the backing paper and place the Scotchcal sticky side up on the bench. The front panel can now be carefully positioned above the Scotchcal. Be careful not to touch the Scotchcal as it will stick fast and is hard to get off again. When you are satisfied that the front panel is lined up, carefully lower it onto the Scotchcal and press down. The Scotchcal should now be stuck to the front panel. The holes can be cut out with a sharp knife or scalpel and the edges trimmed. Be careful not to tear the Scotchcal when doing this. Screw on the four rubber feet and the case is now ready.

The next step is to do the mains wiring. Be *very* careful with this as mains voltages are *lethal*. Try to keep all mains wiring neat and tidy and out of the way of everything else. Strip 200 mm of the outer insulation off a length of mains flex. Thread this through the hole in the back panel and clamp it with a clamping grommet. Make sure it is secure and will not pull out. If necessary use a screw-on mains cord clamp as well. Mount the fuse holder on the back panel and the mains switch on the front panel. Following the wiring diagram carefully, wire up the switch and fuse. All exposed terminals and joins in the mains wiring should be well insulated so that they cannot be accidentally touched. This is best done with heatshrink tubing which is placed over the wire before soldering. When the wire is soldered in place the heatshrink is pushed up over the join till it covers the exposed area. The heatshrink is then heated with a hairdryer or soldering iron and it will shrink to form a tight seal over the join. This should be done for all the connections on the mains switch and fuse holder. As added protection, insulating tape can be wrapped round the entire switch and fuse assemblies.

The transformer should now be mounted using 4BA nuts and bolts. Note that the earth wire of the mains bolts to the transformer mounting bolt. The earth wire should be long enough to ensure that if the mains cord is pulled out of the box the earth will be the last to break. A solder lug should be soldered on to the end of the earth to ensure that it mounts securely. The transformer primary can now be wired up. Once again heatshrink should be used to insulate the terminals. On the transformer specified the primary connections are at the bottom. This can be a bit awkward to get at so be careful and make very sure that the terminal joins do not short on the floor of the box and are well insulated. This, then, completes the mains wiring. Double check that everything is correct and that no connections are exposed.

The pc board should be mounted next. The mounting bolts should be put through the floor of the box and secured with a nut (see Figure 2). The board then mounts on 6 mm standoff spacers and is bolted down. Solder a lug on the remaining lead on the 100n bypass cap and bolt it to the nearest pc board mounting bolt. The switches, terminal posts and LEDs should now be mounted on the front panel. These can be wired up according to the wiring diagram. When wiring the LEDs be sure to get the wires the correct way round. Finally, the transformer secondary can be wired to the board. A 500 mA fuse should be fitted to the fuse holder and a mains plug (if not already fitted) should be wired to the mains flex (be sure to get the connections right). You are now ready to test the supply out.



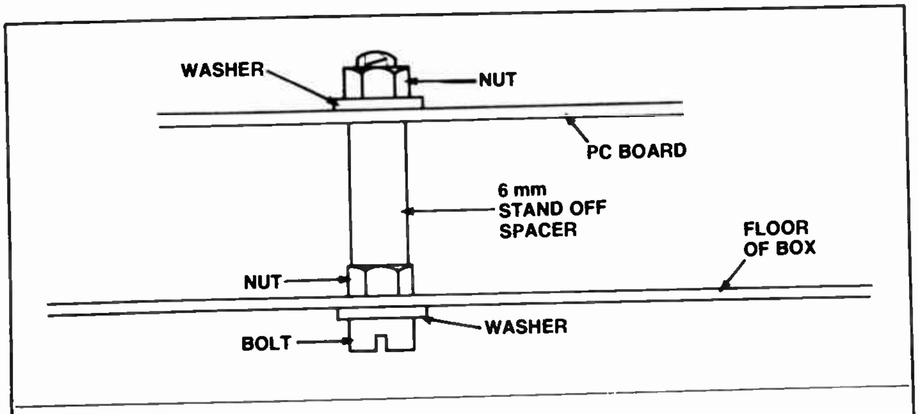
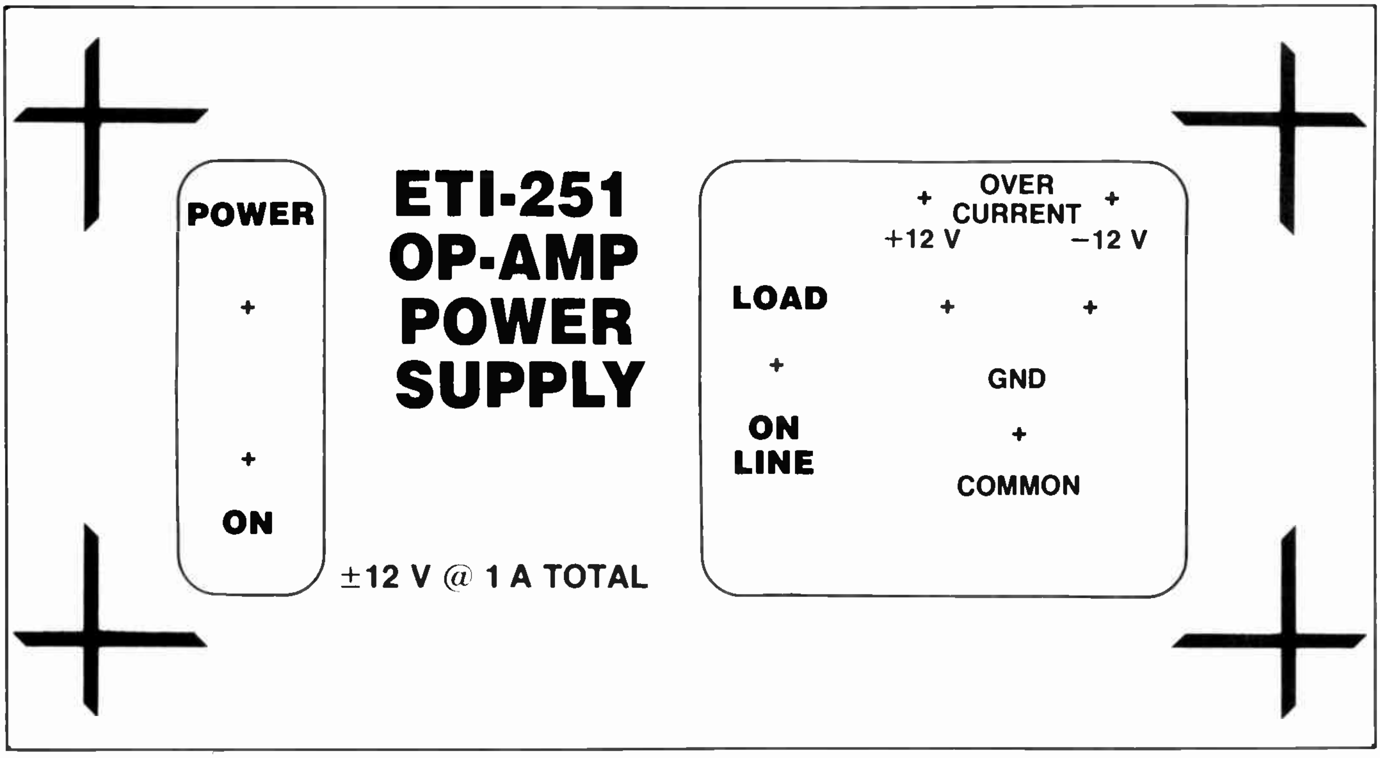


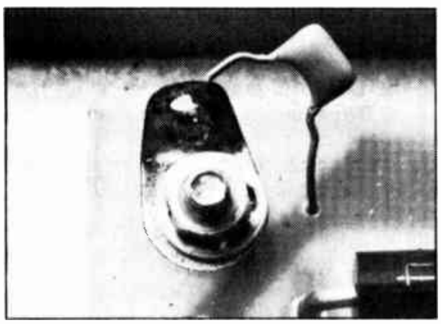
Figure 2. Mounting the pc board.

Testing and using it

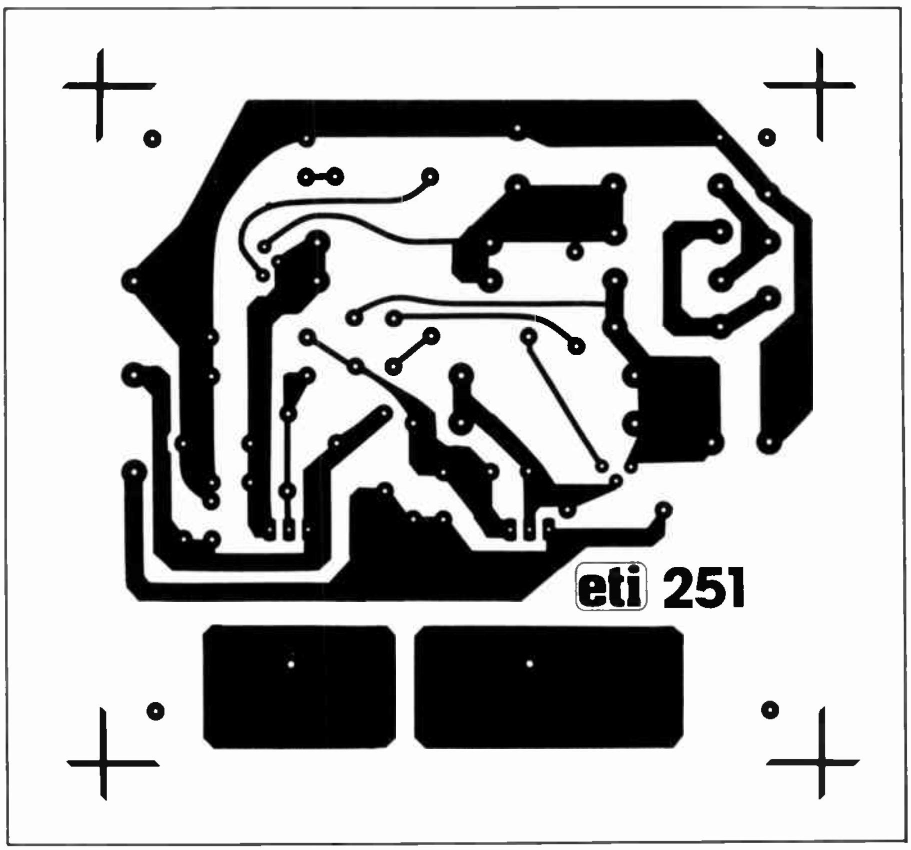
Plug the supply into a mains socket. With the LOAD switch off switch the power on. The power indicator LED should glow and nothing else should happen. With nothing connected to the output terminals, switch the LOAD switch to the ON LINE position. Measure the voltage between the ground and positive terminals. It should read +12 Vdc. Do the same between ground and the negative terminal. It should read −12 Vdc. If this is not the case then unplug, and recheck the wiring and pc board.

If all is well then you can apply a load to the output. Ideally, an 18 ohm, 10 watt power resistor should be wired between the positive terminal and ground. This will draw about 700 mA from the supply and when the supply is turned on with the LOAD switch in the ON LINE position the overcurrent LED above the +12 V terminal should light. If the load is wired across the negative and ground the LED above the −12 V terminal should light. You can accurately determine the current that the LEDs switch at if you have a high power variable resistor or rheostat. This can be





A solder lug should be used to bolt the 100n bypass cap to the chassis. This allows the case to act as an rf shield but still provides for a floating supply.



varied until the LED just lights. The resistance can then be measured and the current determined.

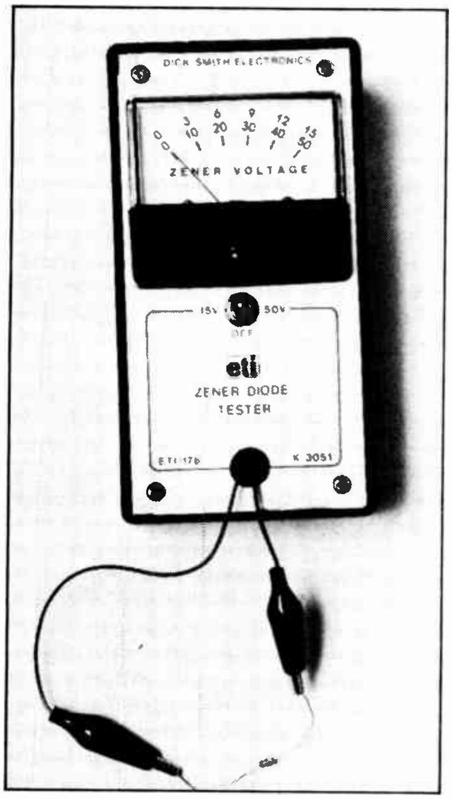
If the supply tests out OK so far, then connect a piece of heavy duty hookup wire between the positive and ground, and switch on. The supply should power up with the +12 V overcurrent LED lit. Do the same for the negative terminal. Finally, do the same thing with a piece of wire between the positive and negative terminals. Both LEDs should light this time. If all is well then switch off, unplug and put the lid back on. You are now ready to use the supply.

The only thing on the front panel that may require a few words is the LOAD switch. This merely disconnects the output terminals from the supply. This was done so that on power up and power down the load can be disconnected from the supply and thus be unaffected by any transients when the regulators power up or down. This is particularly useful when making changes to a breadboard for instance. If the main power switch were used to turn the supply off then you would have to wait a couple of seconds for the capacitors to discharge and the voltage to go to zero before altering the circuit. Using the LOAD switch, the circuit is instantly disconnected from the supply.

If you are powering a circuit and the overload LED comes on you should switch off. Although the supply can maintain overloads for quite long periods it is wise not to run the supply continuously in this mode as thermal stressing of the components can occur which may eventually lead to failure. ●

ETI-176 ZENER DIODE TESTER

REX CALLOHAN



THE ETI-176 IS A simple, battery-powered meter used to measure the zener breakdown voltage of a device connected across its test clips. It is invaluable when sorting devices with indistinct or unfamiliar type numbers and is suitable for measuring devices with zener voltages of up to 50 volts. To improve reading accuracy the meter has two ranges: 0-15 volts and 0-50 volts. If a more accurate reading is required, an analogue or digital voltmeter can be connected directly across the zener while it is connected to the tester.

Ideally, zener diodes should be tested at their manufacturer's rated test current. Because of the diode's "dynamic resistance" the measured breakdown voltage is somewhat dependent upon the current at which it is measured. As a rule of thumb, the test current is roughly that current which causes the zener to dissipate about a quarter of its rated power dissipation. Therefore, to properly test zener diodes, a voltmeter, high voltage power supply and potentiometer or a variable current source would be required. Even if this equipment was readily available, its use would become rather cumbersome, particularly if more than a few devices had to be tested. Additionally, to correctly adjust the test equipment, you would need to know what type of zener you are measuring before you make the measurement to determine what type of zener it is! An easy-to-use unit of the simplicity of the ETI-176 cannot hope to achieve "lab-standard" accuracy with a wide range of zeners, but in practice the small differences between rated and measured zener voltages are of little concern.

Construction

Begin construction by examining the pc board for broken or bridged tracks. When you are satisfied with this, start loading the components, working up to the largest, with the proviso that delicate components like semiconductors ought to be left until last. Be careful to get the polarity of the big 470 µF capacitor, the diodes and the transformer correct. To finish off, cut five lengths of wire to about 150 mm and solder them onto the board in accordance with the wiring diagram. Don't solder the probes on at this stage. You will find this exercise easier if you tin the ends of the wires before you insert them into the holes. Complete this stage by soldering the battery leads into place.

Put the board to one side and begin work on the front panel. Begin by drilling out the required holes in accordance with the drilling diagram, and then stick down the scotchcal. Because the bezels cover the edges of the holes, its not neccesary to get this absolutely precise, but you don't have too much room to spare, so be careful during this operation.

Now mount the meter and the 15/off/50 switch, and insert the rubber gromet for the probes. Pull the probes through their hole and solder them onto the board (Make sure the clips are on the side with the scotcheal, not the other way around! Now solder the leads to the switch and th meter. You will find this process a lo easier if you have used multicoloure wire.

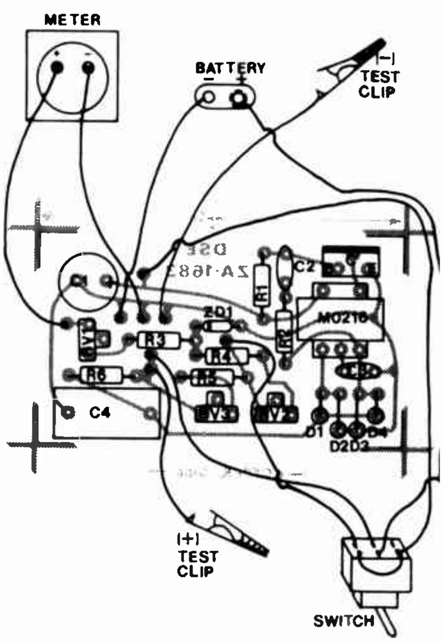
Finally, ensure the switch is in on th off (centre) position and connect the bat tery. The needle should not deflect an the transistor should stay cold. If either i not the case, disconnect the batter immediately and go back over your work checking it carefully. If all is well, procee to calibration.

Calibration

Note that two zener diodes have been included for the calibration procedure. (12V 400mW — IN 963, 30V 1W — IN 4751).

Firstly, with the switch set to the 50V range and the test leads open-circuit, se RV2 and RV3 to maximum resistance and adjust RV1 so the meter reads just ove full scale deflection.

Next, with the switch set on the 15V range, connect the 12V zener diode acros the test clips. Adjust RV2 for a correc reading on the 15V scale. Now switch tc



the 50V range and connect the 30V zener across the test clips. Adjust RV3 for a correct reading. This completes the setup procedure.

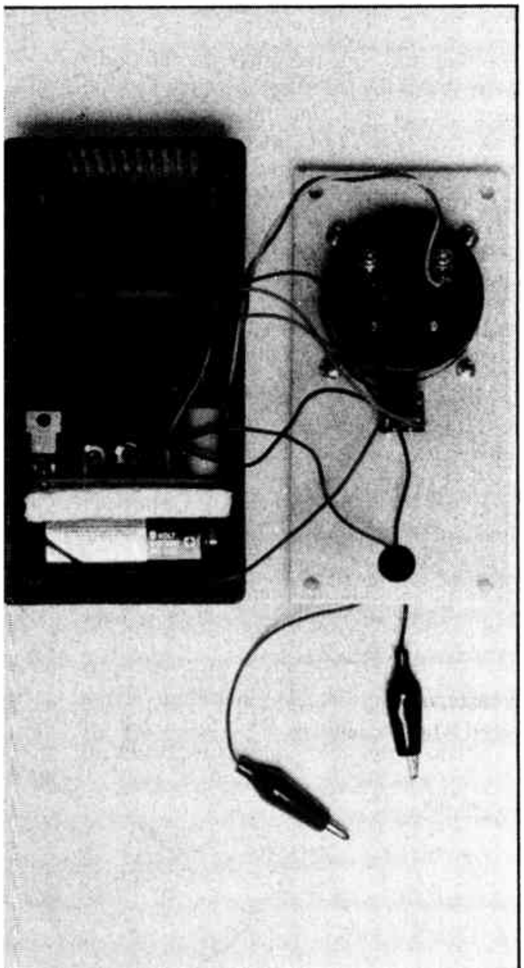
Operation

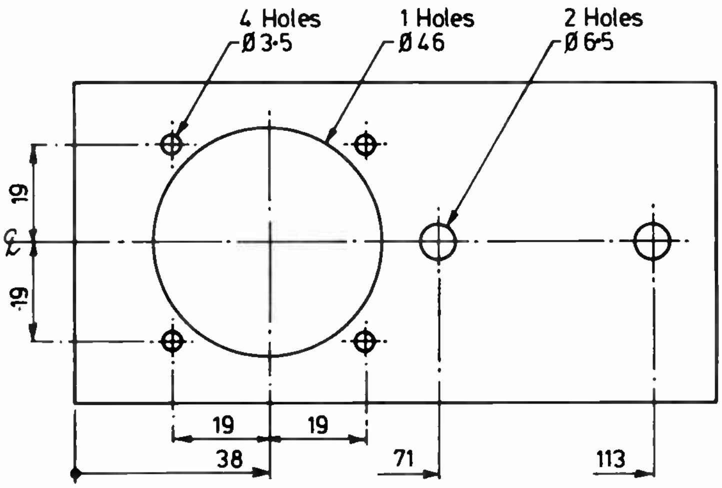
The open-circuit voltage at the test clips is about 100 volts and is current limited to about 8mA. While being quiet safe, there is sufficient current to cause a slight "tingle" if the test clips are touched while the unit is turned on. For this reason, it is recommended that connections only be made to the unit while the power-range switch is in the "Off" position.

As zener diodes are reverse breakdown devices, the cathode of the diode (usually the end with the stripe) is connected to the Red test clip and the anode of the diode is connected to the Black test clip.

After connecting the device to the test clips, switch the power/range switch to the 50 volt range and read the voltage from the meter scale. If the indicated voltage is less than 15 volts, the meter can be switched to the lower range for improved resolution. If the reading is less than about 1 volt, check that the test clips are not reversed. If the reading is greater than 50 volts, the zener may be a high-voltage type or it may be open circuit. ●

Rex Callohan is in the R & D department of Dick Smith Electronics.





PARTS LIST — ETI-176

Semiconductors
- D1,2,3,4 .... IN4002
- Q1 .... MJE 3055
- ZD1 .... 6VZ ZENER 400mW
- TFMR .... M0216 TRANSFORMER

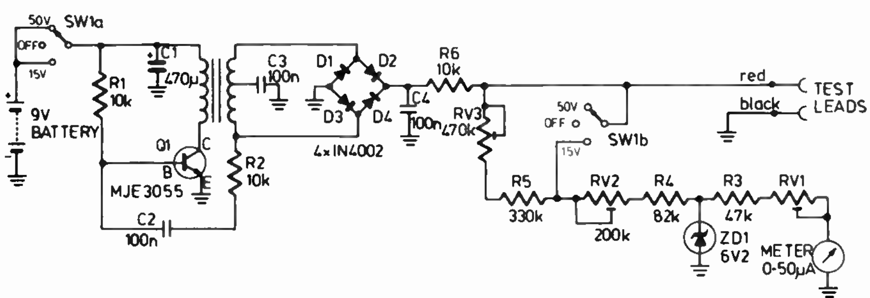
Resistors
(all ¼W unless otherwise stated)
- R1,2 .... 10K
- R3 .... 47K
- R4 .... 82K
- R5 .... 330K
- R6 .... 10K

Capacitors
- C1 .... 470µF 17V RB
- C2 .... .1µF CERAMIC
- C3 .... .1µF CERAMIC
- C4 .... .1µF 630V

Miscellaneous
- SW1 .... DPDT CENTRE — OFF TOGGLE SWITCH
- METER 0 → 50µA ZIPPY BOX
- VR1 .... 200K MINI VERTICAL TRIMPOT
- VR2 .... 200K MINI VERTICAL TRIMPOT
- VR3 .... 500K MINI VERTICAL TRIMPOT
- GROMMET "A1" x 1
- BATTERY SNAP — suit 216-9V
- ALLIGATOR CLIPS
- BLACK x 1
- (MINI) RED x 1
- RIBBON CABLE (200mm)

HOW IT WORKS

An oscillator is formed by Q1 and a transformer having about 1:10 turns ratio. The voltage across the secondary of the transformer is rectified by a diode bridge and results in 80-100 volts dc across capacitor C4, depending on the condition of the 9V battery. This voltage is presented to the zener diode via the current limiting resistor R6. Across the zener diode under test is the voltmeter circuit which scales the voltage into two ranges chosen by the 15V/50V switch. ZD1 acts with RV1 as a meter voltage protection circuit, should there be no connection at the test clips. The maximum available test current is around 8mA. The actual current in the zener-under-test will vary according to its zener voltage.



OP-AMP TESTER

The ETI-183 is a simple op-amp tester which could save your future projects hours in agonizing over using that old op-amp that's been lying in the drawer for the last year.

Robert Irwin

TEN YEARS AGO it was a must for anyone working in electronics to be equipped with some kind of device, (simple or hideously complex), that could test a transistor and determine its questionable state of health. The transistor tester is still a favourite workhorse but with the increase in the use of the op-amp as a basic building block it became essential for the analogue artisan to have a quick and easy means of checking ICs. This project is designed to be a simple stand-alone unit which will tell you, in terms of a few LEDs, whether your op-amp belongs in your next project or on the scrapheap with the vegetable peelings.

Design Details

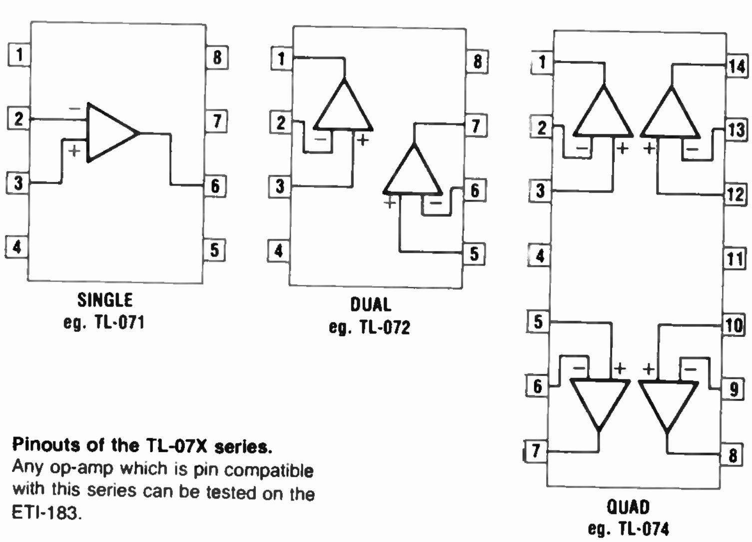
The tester was designed to accommodate single, dual and quad packages with equivalent pinouts to those of the TL08x series. This includes most of the popular general purpose op-amps such as the μA741, LM301 and LF347. Three tests are provided on the tester. Firstly, an excessive power supply current indication tells you that there is a short circuit on the op-amp power supply pins. The other two tests find out whether the op-amp is, in fact, amplifying properly. For both of these tests the op-amp is configured as a non-inverting, dc amplifier with a gain of 20. A dc test can be performed which grounds the input of the op-amp and indicates any excessive dc offset on the output. If the output sits at one of the rails in this situation then it is probably deceased. To make sure the op-amp is in fact amplifying, a square wave is connected to the input. The output is monitored by a window comparator which turns on two LEDs to indicate that the signal is being amplified up to the correct level on both the positive and negative sides.

When dual or quad packages are being tested, each op-amp in the package is tested individually. The appropriate op-amp is selected by a four position rotary switch and the ac and dc tests are then applied to this op-amp only. For op-amps that require compensation, such as the LM301, a 10 pF capacitor can be switched in across pins 1 and 8. This is only necessary for single op-amp packages as all the dual and quad packages that can be used with this tester are internally compensated.

Construction

Construction should begin by carefully checking the pc board for broken or shorted tracks. If the pc board checks out OK then start soldering. The eight wire links should be located and soldered in first. These are made up of pieces of tinned copper wire cut to the appropriate length. Resistors and trimpot can go in next. Note that the resistors in the feedback circuit of the test op-amps are of 1% tolerance. Solder in the capacitors making sure that you get the polarity correct on the two electrolytics. The bipolar electrolytic has no polarity and can be put in either way round. The next step is to solder in the ICs, zeners and transistor. It is vitally important to get these components the right way round. If you wish you can use sockets with the ICs.

Now comes the hard bit! The prototype was mounted on the front panel of a jiffy box so the trick comes in getting all the LEDs, test sockets and switches the correct height. The four position rotary switch can be mounted and soldered in flush with the board. If you do not have a pc board mount type switch you will have to trim the pins on the back with a pair of side cutters so that they will fit through the holes.



Pinouts of the TL-07X series.
Any op-amp which is pin compatible with this series can be tested on the ETI-183.

Next mount the three wirewrap IC sockets that are to be used for the test ICs. The pins on these should all be the same length so mount the sockets to stand up off the pc board about 14 mm making sure that they are all the same height and level. They can be 'tacked' in by soldering just two pins on each socket rather than soldering all pins at this stage. This will make any height adjustments a lot easier.

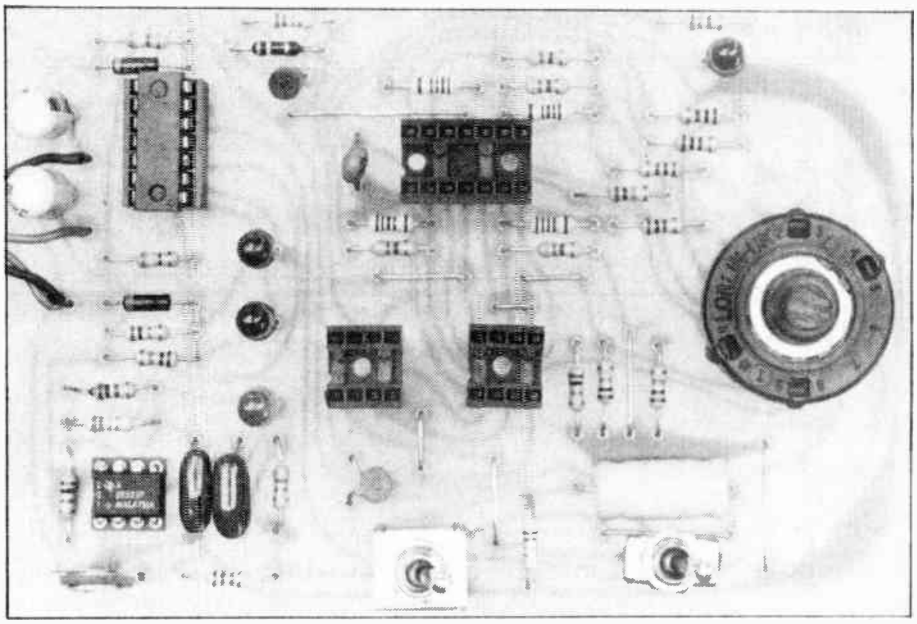
The INPUT and COMPENSATION toggle switches can be mounted in one of two ways. Small lengths of hookup wire (about 20 mm) can be attached to the lugs of the switches and then soldered into place on the pc board or the lengths of tinned copper wire (such as the type used for wire links) can be soldered on to the switch lugs to form 'legs' for the switch to stand on. If you use the latter method then do not, at this stage, solder the switches to the pc board but merely poke the wire legs through the holes on the board and leave them. The height can be adjusted when you mount the board and the switches soldered in then. The four LEDs should also be just pushed into the holes on the pc board and left, then soldered in later.

Finally, the battery terminals and power switch should be wired up with hookup wire according to the wiring diagram and overlay. Take careful note of the polarities of the terminals and leads. There has been provision made on the pc board for a BNC socket to allow a CRO to monitor the output from the DUT (Device Under Test). If you want this the socket should be wired in with hookup wire at this stage. This completes the pc board for now and you can turn your attention to the box.

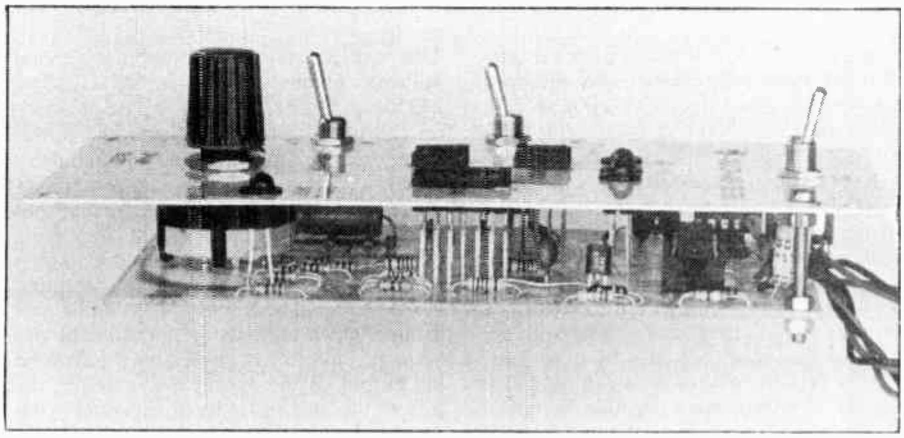
The prototype was housed in a 150 x 90 x 50 mm jiffy box. The pc board mounts on the aluminium lid. The front panel can be marked using the front panel artwork as a template. Mark the positions of all the holes to be drilled and the corner points of the three holes for the IC sockets. The holes for the LEDs and toggles can be drilled using a 6.5 mm drill and the hole for the rotary switch can be drilled using a 9.5 mm one.

Unless you have a suitable square punch set, the easiest way of cutting out the holes for the IC sockets is to drill a hole in the centre of each socket position and file it out to size with a small square file. Be careful not to file too big a hole. If you are including a BNC socket for a CRO output then a suitable hole should be drilled in the side of the box.

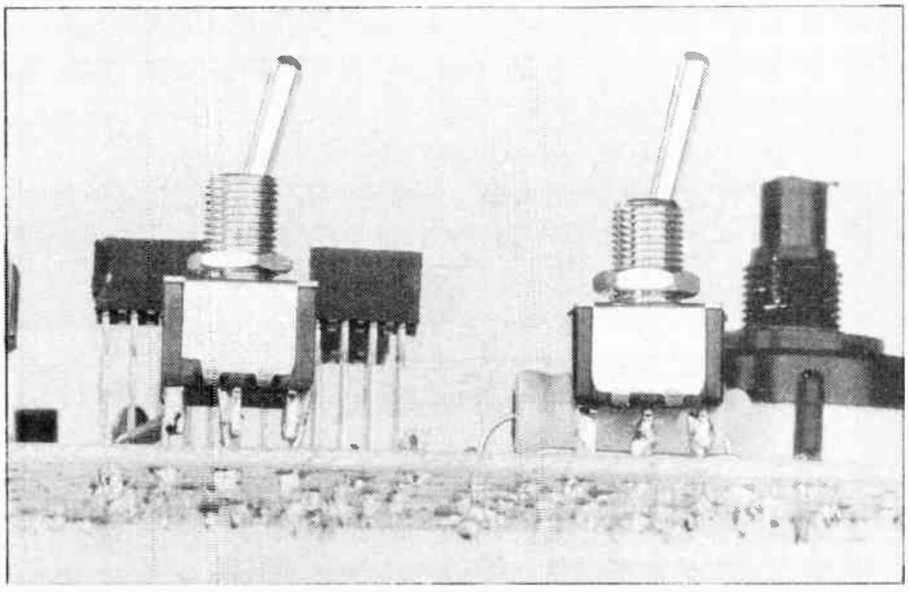
Once you have cut all the holes, do a trial fit to make sure everything lines up OK. Once everything fits, the Scotchcal front panel can be stuck on. If you have a blue on white plastic Scotchcal label it would be advisable to spray the aluminium front panel with white paint before applying the Scotchcal. When it is dry the Scotchcal can be carefully applied. Line it up accurately the first time because once it has stuck you'll have a hard time getting it off again. Trim out the holes in the Scotchcal with a sharp knife or scalpel but be careful not to tear it.



Top view of the pc board showing the position of components. Note the three wirewrap IC sockets used for the test op-amps.



How it all fits together. The board is mounted on the front panel by way of the rotary switch and two mounting bolts. The height of the LEDs should be adjusted after mounting of the board.



Mounting of the two toggle switches. Note the wire legs soldered to the switch lugs to enable them to be pc board-mounted.

HOW IT WORKS — ETI-183

The circuitry for the tester is fairly simple. IC1 is an LM555 timer which, in this case, is configured as a free running astable multivibrator. The frequency of the multivibrator is given by the formula:

$$f = 1.49/(R1 + R2)C3.$$

With the values given this gives a frequency of approximately 1 kHz. The output of the LM555 forms the ac signal that is used to drive the DUT (Device Under Test). A green LED, LED1, is used to indicate an output from the LM555 with R7 limiting the current to about 4 mA. The output from the LM555 is divided down by the resistive divider network formed by R6, R8 and RV1. RV1 controls the level of signal being fed to the DUT. The input can be switched from ac to ground by SW2 and the appropriate op-amp input is selected by the B pole of SW3.

The device test sockets are wired up so that the single package, op-amp 1 of the dual package, and op-amp 1 of the quad package are all wired up in parallel. Op-amp 2 of the dual and op-amp 2 of the quad package are wired in parallel. All test op-amps are configured as non-inverting stages. The feedback network is the same for all op-amps and comprises a 100k feedback resistor with a 4k7 resistor to ground. Op-amp 1 has the addition of a 100p capacitor (C6) across the feedback resistor. This is for extra stability at high frequencies for some single op-amps such as the NE5534 which are prone to oscillations. Capacitor C7 can be switched in between pins 1 and 8 of the op-amp 1 test position to take into account op-amps such as the LM301 which need compensation between these pins for stable operation.

A 100R resistor is included on the output of each op-amp to provide some load isolation and enhance stability, and a 47k resistor is connected from each input to ground to provide a dc path to ground when the op-amp is not selected.

The appropriate output is selected by the C pole of SW3. The output from the op-amp is selected by the other pole of SW2 to be either dc coupled or ac coupled via C5 and R9.

The output is fed to the input of a window comparator formed by IC2a and IC2b. The positive input of IC2a is biased to 3.3 V by ZD1 and the negative input of IC2b is biased to −3.3 V by ZD2. The two spare inputs are connected to the output of the op-amp. When the voltage from the op-amp is between +3.3 and −3.3 V both comparator outputs are high therefore LED2 and LED3 are off. If the op-amp output goes above +3.3 V the output of IC2a swings low and turns on LED2. If the output goes below −3.3 V then the output of IC2b goes low turning on LED3. Therefore, if the output is a symmetrical square-wave, as it should be, each LED will be turned on and off in turn as the output swings positive and negative with the overall effect of both LEDs appearing lit.

If the op-amp is only amplifying one side then only one LED will light. In the dc mode, if the output is sitting at one of the rails then the appropriate LED will light. The outputs of the comparators are open collector and can sink a few milliamps. The current is limited by R12 and R13 to about 1.5 mA.

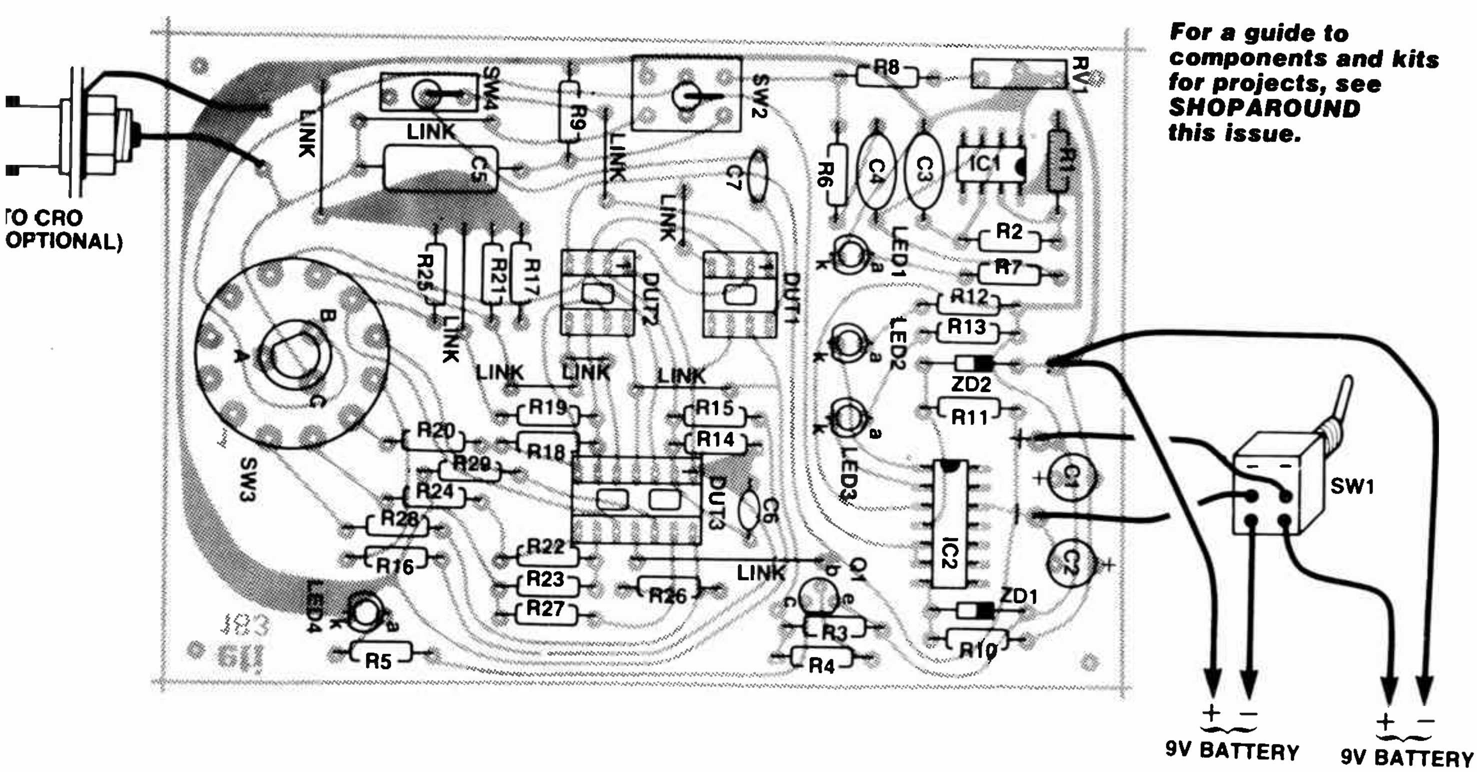
The power is supplied by two 9 V batteries connected to give a split 9-0-9 V supply. This is filtered by C1 and C2. The positive rail to the op-amp test sockets is monitored by an overcurrent indicator formed by Q1, LED4 and associated resistors. R3 is in series with the supply. Q1 is connected across this resistor in such a way that, as the current being drawn from the supply increases, the voltage across R3 increases. When this voltage reaches 0.6 V the transistor, Q1, starts to turn on which turns on LED4 indicating that excessive current is being drawn from the supply. This occurs at about 10 mA. To increase the current that can be drawn before the LED turns on simply decrease R3.

PARTS LIST — ETI-183

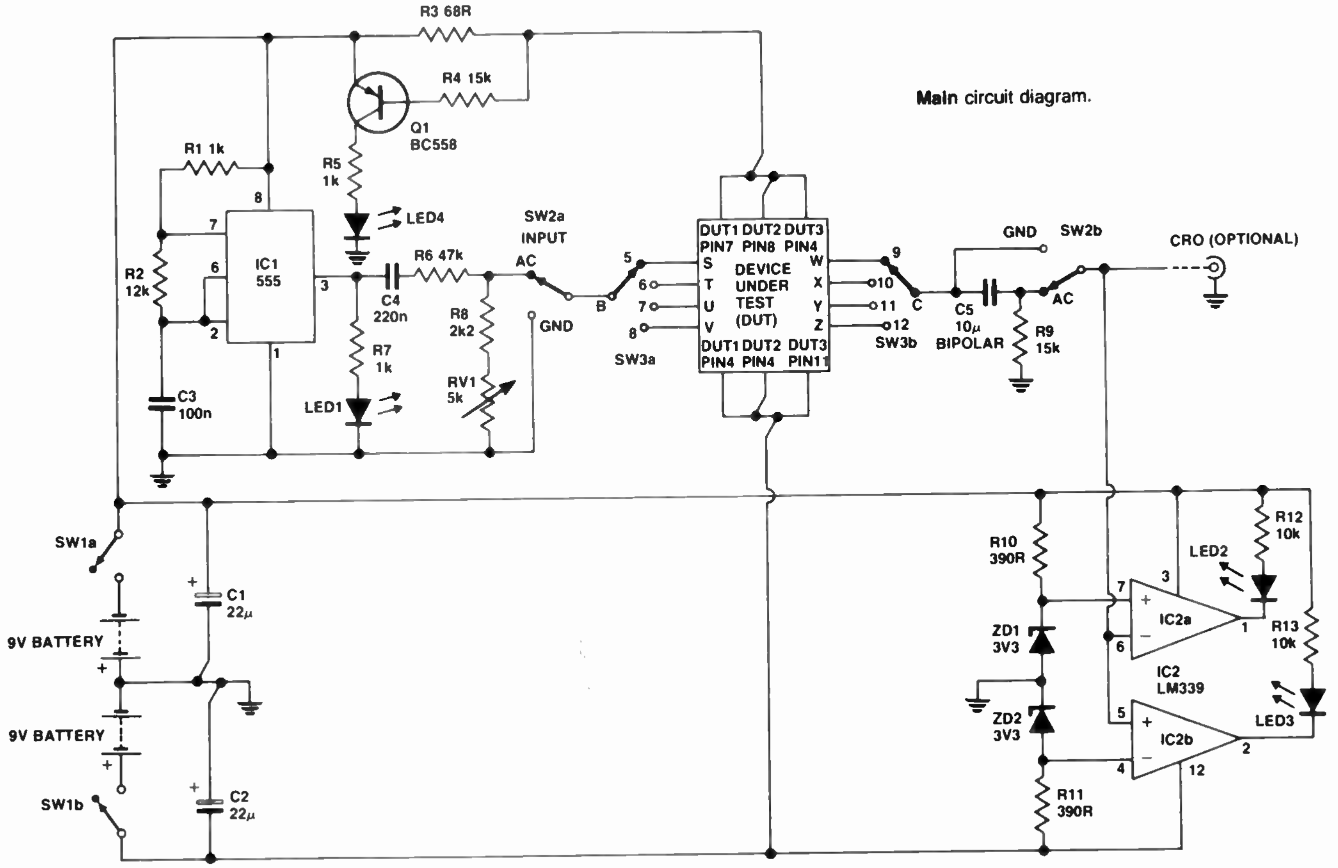
Resistors	all ¼ W 5% unless noted
R1, 5, 7	12k
R2	33k
R3	68R
R4,9	15k
R6	47k
R8	2k2
R10, 11	390R
R12, 13	10k
R14, 18, 22, 26	100k 1%
R15, 19, 23, 27	4k7 1%
R16, 20, 24, 28	100R
R17, 21, 25, 29	47k
RV1	5k trim.
Capacitors	
C1, 2	22µ 25 V RB electro.
C3	100n greencap
C4	220n greencap
C5	10µ 25 V bipolar electro. axial mount
C6	100p ceramic
C7	10p ceramic
Semiconductors	
IC1	LM555
IC2	LM339
ZD1, 2	3V3 400 mW zener
LED1	green 5 mm LED
LED2, 3, 4	red 5 mm LED
Q1	BC558 or similar
Miscellaneous	
SW1	DPST momentary action toggle
SW2	DPDT toggle
SW3	3 pole, four position rotary switch
SW4	SPDT toggle

ETI-183 pc board; 2 x 216 battery terminals; 2 x 8 pin wirewrap IC sockets; 14 pin wirewrap IC socket; 150 x 90 x 50 mm jiffy box; 2 x 6BA 25 mm bolts; 6 x 6BA nuts; 30 cm length tinned copper wire; 4 x LED mounting grommets and washers; hookup wire.

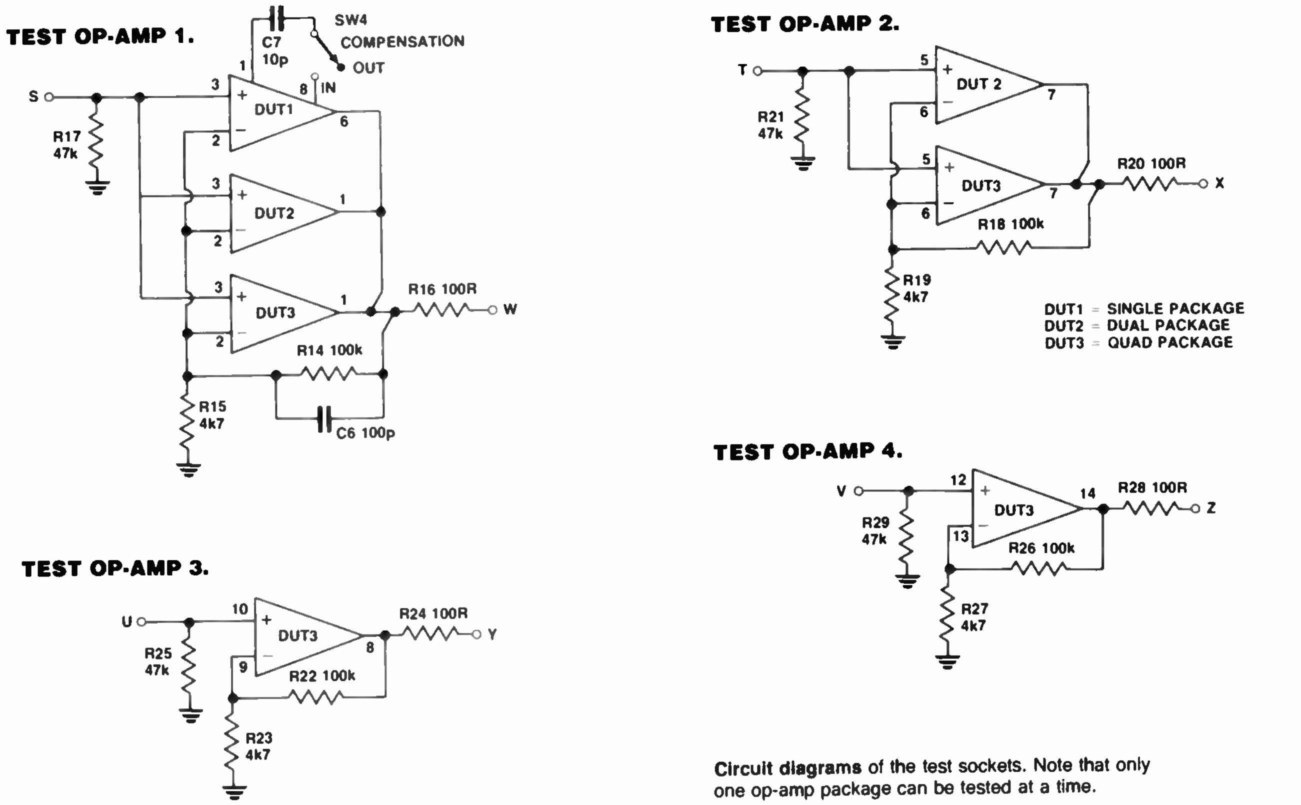
Estimated price: $25



For a guide to components and kits for projects, see SHOPAROUND this issue.



Main circuit diagram.



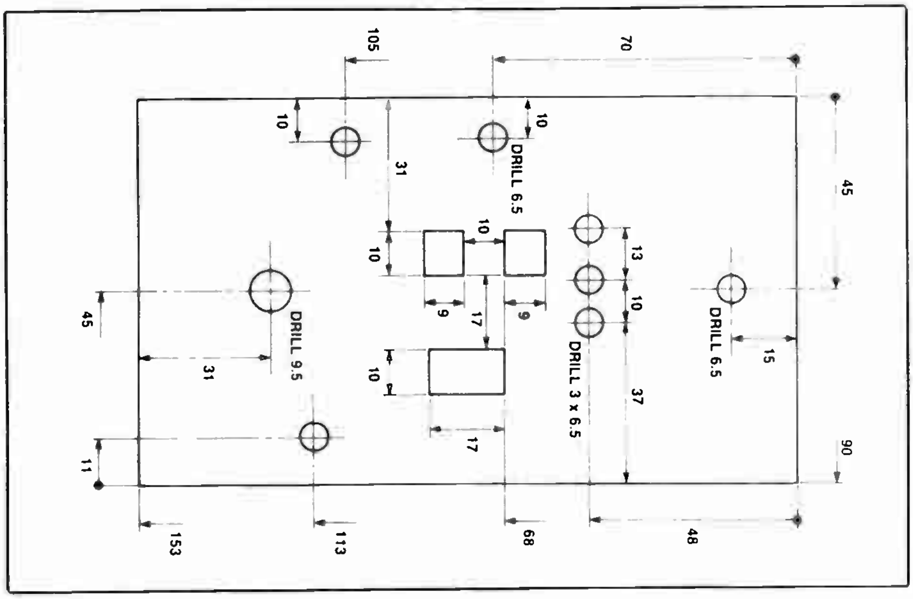
Circuit diagrams of the test sockets. Note that only one op-amp package can be tested at a time.

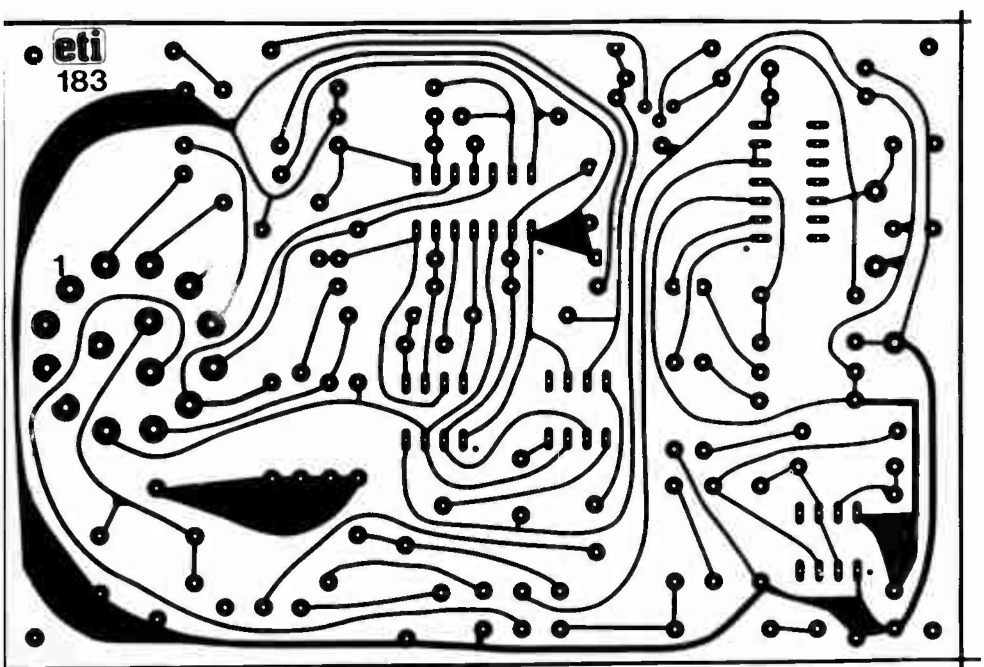
Once the front panel is finished you can begin to mount the pc board. Two 6BA bolts should be put through the front panel at the top and bolted into position. The pc board mounts on these two bolts and the rotary switch. Make sure you have the key washer of the switch adjusted to give four positions. Two more nuts should be screwed about half way down the mounting bolts. The height of these will be adjusted later to allow the pc board to sit level.

The main on/off switch should be mounted first. Locking washers for the LED mounting grommets should be placed over each of the LEDs. Put the pc board in place, gently easing the IC sockets through the holes cut for them. Screw the nut down on the rotary switch and then adjust the nuts on the mounting bolts so that the pc board sits level. The nuts to secure the pc board can now be screwed into place. The nuts for the toggle switches should also be screwed on. The legs of the toggle switches can now be soldered into place on the pc board. LED mounting grommets can be placed in the appropriate holes and the LEDs pushed up into them. The LEDs can then be soldered in and the locking washers slid into place. The IC sockets can also be soldered up.

The tester should now be ready for adjustment. Now comes the chicken and the egg problem. To set the tester up you need an op-amp that you know is working. Place the op-amp in the appropriate socket. Set the INPUT switch to AC and the OP-AMP SELECT switch to 1. If the op-amp is one that needs a compensation capacitor between pins 1 and 8 such as an LM301 switch the COMPENSATION in, otherwise leave it out. Set the trimpot fully clockwise (minimum resistance). Switch the TEST switch on. The green AC LED should light to indicate that the oscillator is on. Slowly turn the trimpot anti-clockwise until both the + and − indicator LEDs are just lit. Turn the trimpot just a little past this point and leave it set there. If the indicator LEDs don't come on at all check your circuit for wiring faults. If you have a CRO handy you can check the output of the '555 to make sure it is oscillating.

If all is well, try replacing the op-amp you have in at present with a quad package and checking each op-amp in turn. To test the supply short indicator try plugging an op-amp in the wrong way round! Reassemble the box.



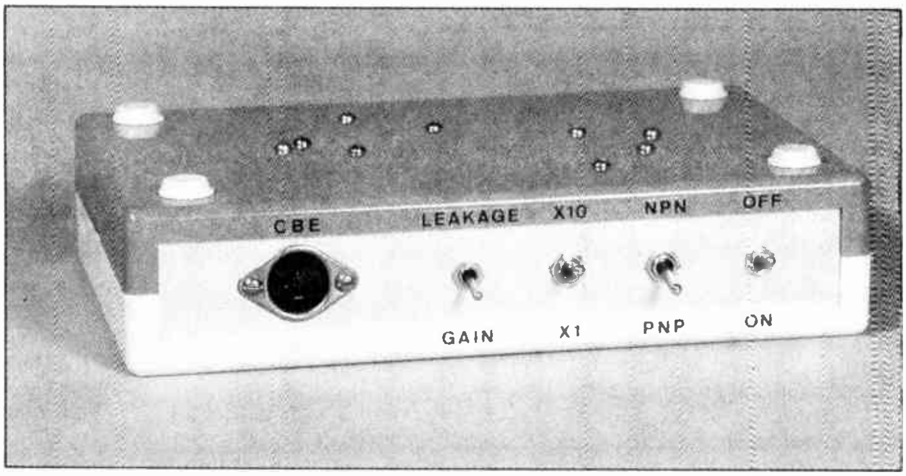


Using it

Using the tester is fairly simple. The op-amp you wish to test should be inserted into the appropriate socket (single, dual or quad). The OP-AMP SELECT switch selects which op-amp in the package is to be tested. Obviously, for a single op-amp package this switch must be set to 1. For a dual package, 1 or 2 can be selected.

The INPUT switch should be set for the test you wish to perform. If GND is selected then the input to the op-amp under test is grounded and the output is dc coupled to the comparator which drives the indicator LEDs. If AC is selected then the input to the op-amp is a square wave derived from the output of the LM555. In this case the output is ac coupled to the comparators. The COMPENSATION switch should be left in the OUT position unless a single package which requires compensation between pins 1 and 8 (such as an LM301) is being tested.

To test the op-amp just press the main switch to the on position. The green LED should always light indicating that the LM555 is outputting something. If this does not light then check your battery voltage. With the INPUT switch set to GND *no* red LEDs should light. If the SUPPLY SHORT LED lights then the op-amp is either in the wrong way round or it has a short on its supply pins indicating that it is malfunctioning. If the + LED lights then the output is sitting near the positive rail. If the − LED is lit then the output is sitting near the negative rail. Both of these conditions indicate a dud op-amp. In the ac input mode both the + and − LEDs should light indicating that the op-amp is amplifying in both the positive and negative directions. If either LED does not light then the op-amp is malfunctioning. If the op-amp passes the ac, dc and supply short tests then it should be OK to use. If it doesn't then it's probably destined for the dustbin. ●



This transistor tester was developed to fulfil the need for a unit that would provide a quick and convenient means of testing DC current gain (h_{fe}) and leakage. Some other testers are inexpensive circuits based on moving coil meters but the relatively high cost of meters these days makes these designs less attractive than they once were. On the other hand, an equivalent circuit using a digital readout is substantially more complex but not necessarily much more expensive. A 2-digit display gives better accuracy than most of the moving coil meters currently on sale and certainly more than adequate accuracy for this application.

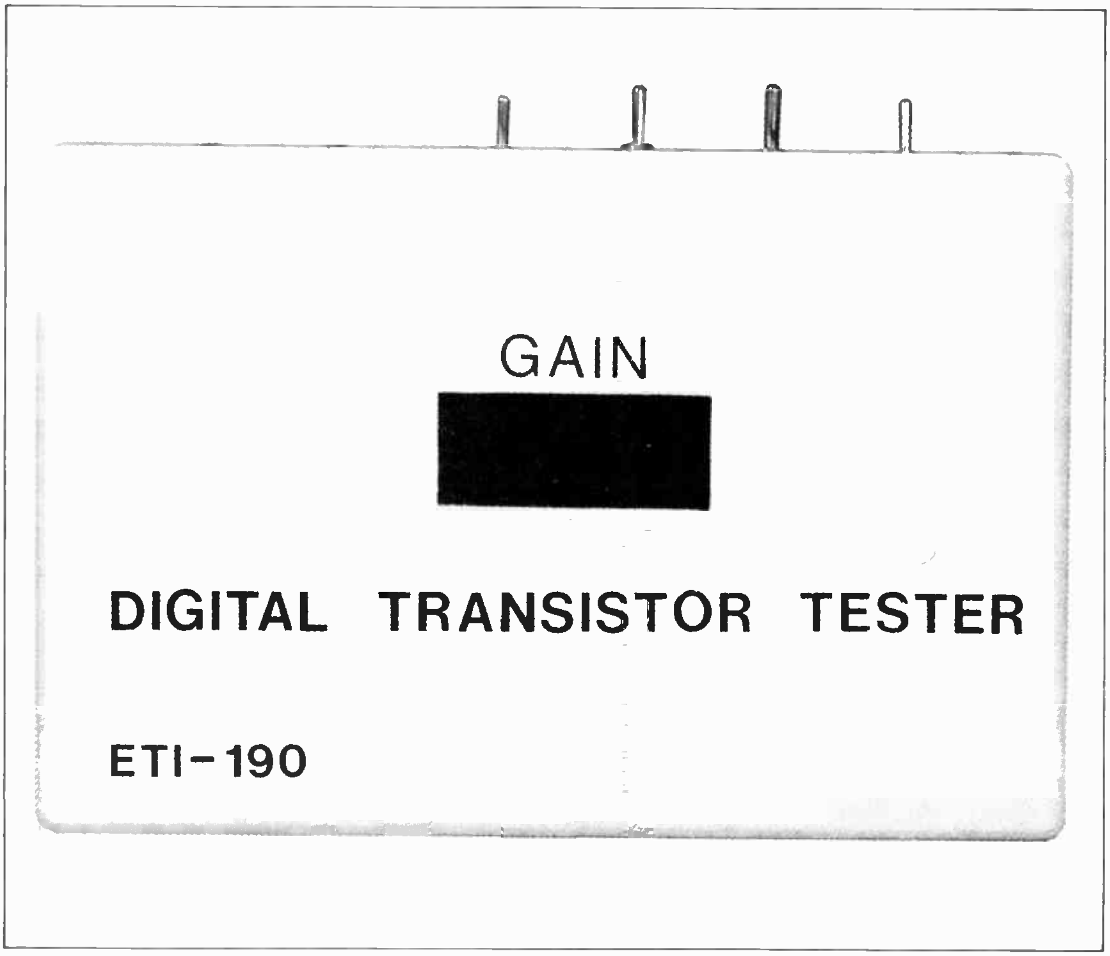
This design has proved to be quick and easy to use in practice. It can test both NPN

DIGITAL DIAGNOSIS

Building a transistor tester

Let the digits do the work and build this handy transistor tester. ETI's Robert Penfold shows you how.

ELECTRONICS
ETI-190



and PNP devices and has two gain ranges, 0-99 and 0-990. This enables reasonably accurate checks to be made on anything from low gain RF and switching devices through to very high gain audio devices.

An over-range indicator is included on the 2-digit LED display and simple leakage tests can also be made using the unit. Power is obtained from an internal 9 V battery.

Testing theory

Most simple transistor testers operate using the basic test setup of Figure 1a. Battery B1 supplies power to the test device with the correct polarity and meter M1.

A bias generator registers the flow of current in the collector circuit. The current flow from the collector to the emitter is normally very low and would typically be about one microamp or less for a silicon device. This is termed the *leakage* current. Providing a small forward bias current to the base terminal results in a much larger flow of current in the collector circuit. The current gain of the transistor is equal to the collector current divided by the base current.

In this case R1 provides a small reference current to the base of the test device. The higher the gain of the device, the greater the collector current that will be registered on M1. In fact the circuit can be arranged so that M1 provides a readout direct in current gain. For instance with the value of R chosen to give a base current of 1uA and M1 having a full scale value of 1mA (1000μA) M1 would accommodate a current gain range of 0-1000.

This assures the leakage current is very low and is not inflating the collector current flow but as explained previously, with silicon devices the leakage current is almost invariably insignificant.

Figure 1a shows the test setup for NPN transistors but the arrangement for PNP testing is essentially the same and is shown in Figure 1b. It is just a matter of reversing the polarity of the battery and the meter.

This type of testing has a slight flaw in that it is not checking the gain at specific collector currents and voltages. These both vary according to the gain of the test component (high gain giving increased collector current and reduced voltage). The uncertain collector voltage is not of great importance as quite large variations in this factor have a minimal effect on the gain of test components. Variations in collector current have a greater (although still fairly small) influence on current gain. Results are perfectly acceptable in practice, provided test components are not tested at very low collector currents. The use of two or more measuring ranges ensures that low gain devices can be checked at an acceptable current and also that they will give a high enough reading to provide good accuracy.

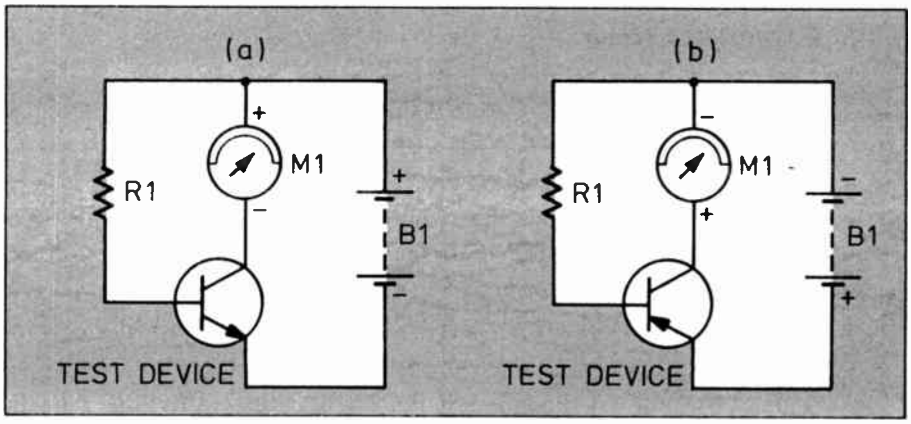


Figure 1: basic test circuits for NPN and PNP transistors.

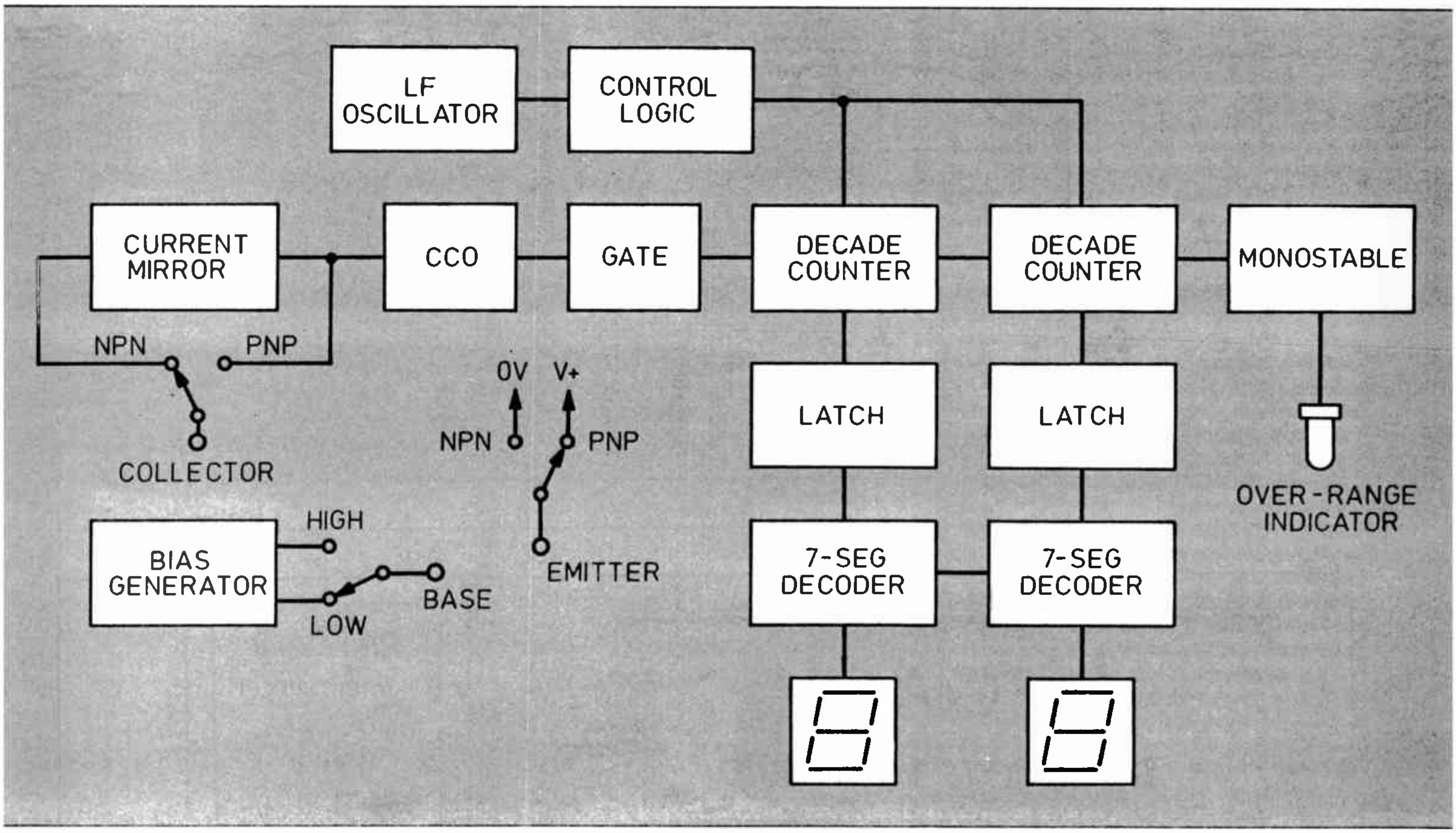


Figure 2: block diagram of the digital transistor tester.

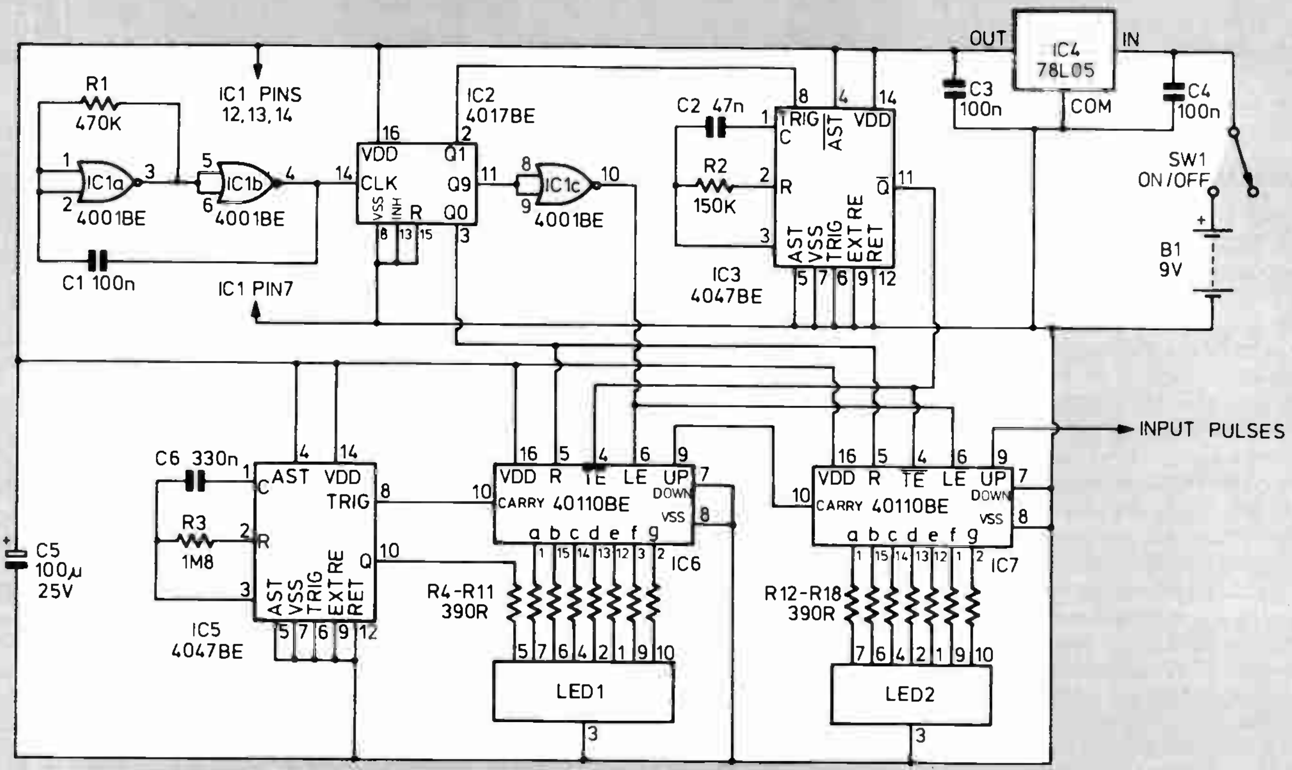


Figure 3: the circuit diagram of the display section.

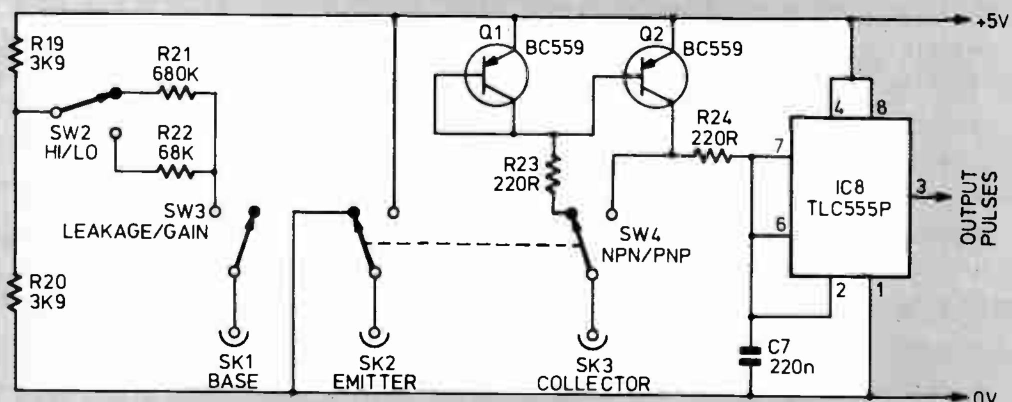


Figure 4: the circuit diagram of the input section.

How it works

The display section (Figure 3) is built around two CMOS 40110BE integrated circuits which each contain a decade counter, latch and a 7-segment decoder/driver. They also 'toggle enable' inputs that effectively give a built-in gate which avoids the need for an external gate circuit. The 40110BE is actually an up/down counter but in this application it is only used as a straightforward up counter.

Unlike most CMOS devices, these have a

high output current capability and they directly drive the common cathode LED displays via current limiting resistors. The carry output of IC6 is issued to drive a positive edge triggered monostable based on IC5 and a trigger signal is only provided to IC5 if an overflow occurs. IC5 then activates the overflow indicator LED for just one second, giving clear warning that the main display reading is erroneous. The overflow indicator is the otherwise unused decimal point segment of the most significant display digit.

The low frequency oscillator uses two gates of IC1 wired as inverters and connected in the standard basis CMOS astable configuration. The oscillator operates at approximately 20 Hz but due to a divide by ten action in the control logic circuit this equates to only about two readings per second.

The control logic circuit is built around IC2, which is a CMOS 4017BE one-of-ten decoder. Output 0 (pin 3) going high resets the counters to zero and output 1 (pin 2) going high generates the gate pulse. It does so by triggering monostable multivibrator IC3. This generates a negative gate pulse of about 17 mS at its Q output which is used to drive the gate inputs of IC6 and IC7.

With this type of display circuit both gate inputs must be driven and not just the one belonging to the least significant digit. Output 9 drives the latch inputs of the counter chips and latches the new reading prior to a new cycle commencing and the counters being reset. IC1c is used to invert this signal so as to give the negative latching pulse required by IC6 and IC7. Note that outputs 2 to 8 of IC2 as well as its carry output are left unused. Also, one gate of IC1 is left unused but its inputs are tied to the positive supply rail in order to prevent spurious operation.

Power is obtained from a 9 V battery but a small 5 V regulator provides a well regulated 5 V supply for the entire circuit. This ensures the unit provides consistent results as the battery voltage drops.

In the input circuit (Figure 4) R19 and R20 form a centre-tap on the supply lines and this drives the base terminal of the test device via one of two switched current limiting resistors (R21-R22). These provide the unit with its two measuring ranges and SW2 is used to select the desired base feed resistor. By driving these resistors from the mid-supply voltage there is no need to bother with any NPN-PNP switching in the base circuit. SW3 can be used to cut off the base bias current so that leakage checks can be made.

Q1 and Q2 form a conventional current mirror circuit, with R23 and R24 providing current limiting in the event of any accidental short circuits or closed circuit devices being checked.

There is no need to switch out the current mirror in the PNP mode and NPN/PNP switching can therefore be achieved using just a DPDT switch (SW4).

In theory, Q1 and Q2 should be a matched pair to obtain an accurate 1:1 ratio of input to output current. In practice, quite wide differences in their gain did not produce any great discrepancies between the NPN and PNP modes. One way of ensuring really accurate results is to use any two BC559s for Q1 and Q2 initially and to then use the unit to select two reasonably well-matched transistors from a batch of (say) half a dozen devices. However, this is by no means essential and unmatched devices should suffice. It is advisable to use transistors from the same gain group (say, two BC559Bs).

The CCO is just a 555 astable circuit. No resistor is used between pins 6 and 7 in order to keep C7's discharge time as short as possible. This makes the period of each cycle almost totally dependent on the charge current and ensures good linearity. A TLC555CP is specified for the IC8 position as this gives a much lower current consumption than the standard 555 and it also seems to be somewhat faster in operation (which again aids good linearity). The collector current at which the devices are tested is dependent on their current gain but is typically around one or two millamps.

System operation

The block diagram of Figure 2 shows the general arrangement used in this transistor tester. The unit breaks down into two distinct sections, one providing the display and the other converting current gain into a suitable driver signal for the display circuit. The bulk of the circuit is used to provide the display.

The display section is a simple frequency counter circuit.

That is controlled by the low frequency oscillator and a simple logic circuit. First the two decade counters are reset to zero and then a gate at their input is activated. Input pulses then flow into the 2-digit counter circuit until the gate pulse ends. Another pulse from the control logic circuit then activates the two latches, which store the count and feed it through to two 7-segment decoder drivers. The 2-digit display therefore shows the number of pulses received during the gate period.

This cycle is then repeated, with the decade counters being reset again. Note though that resetting the counters does not affect the latches and the old count is displayed until a new one has been taken and fed into the latches. The unit accordingly provides a continuous readout which is updated approximately twice per second. If the count goes beyond 99, it is detected by a monostable driven from the second counter, and activates a warning LED.

The display circuit requires the collector current of the test components to be converted into a proportional frequency. This is not difficult and all that is needed is a current controlled oscillator (CCO) having a reasonably linear control characteristic. This leaves a slight problem in that the CCO operates as a current sink which will operate properly with the PNP devices (which act as current sources) but is incompatible with NPN transistors which act as current sinks and must be fed from a current source. The solution to the problem is to drive the CCO direct from PNP transistors and to drive it via a current mirror for NPN transistor testing.

A current mirror is a very simple circuit which provides an output current that is equal but opposite to the input current it receives. One pole of the NPN/PNP switch connects the collector test socket to the input of the current mirror or the CCO, as appropriate. The other pole connects the emitter test socket to the appropriate supply rail. A bias generator provides two switched base bias currents and these provide the unit with its two measuring ranges.

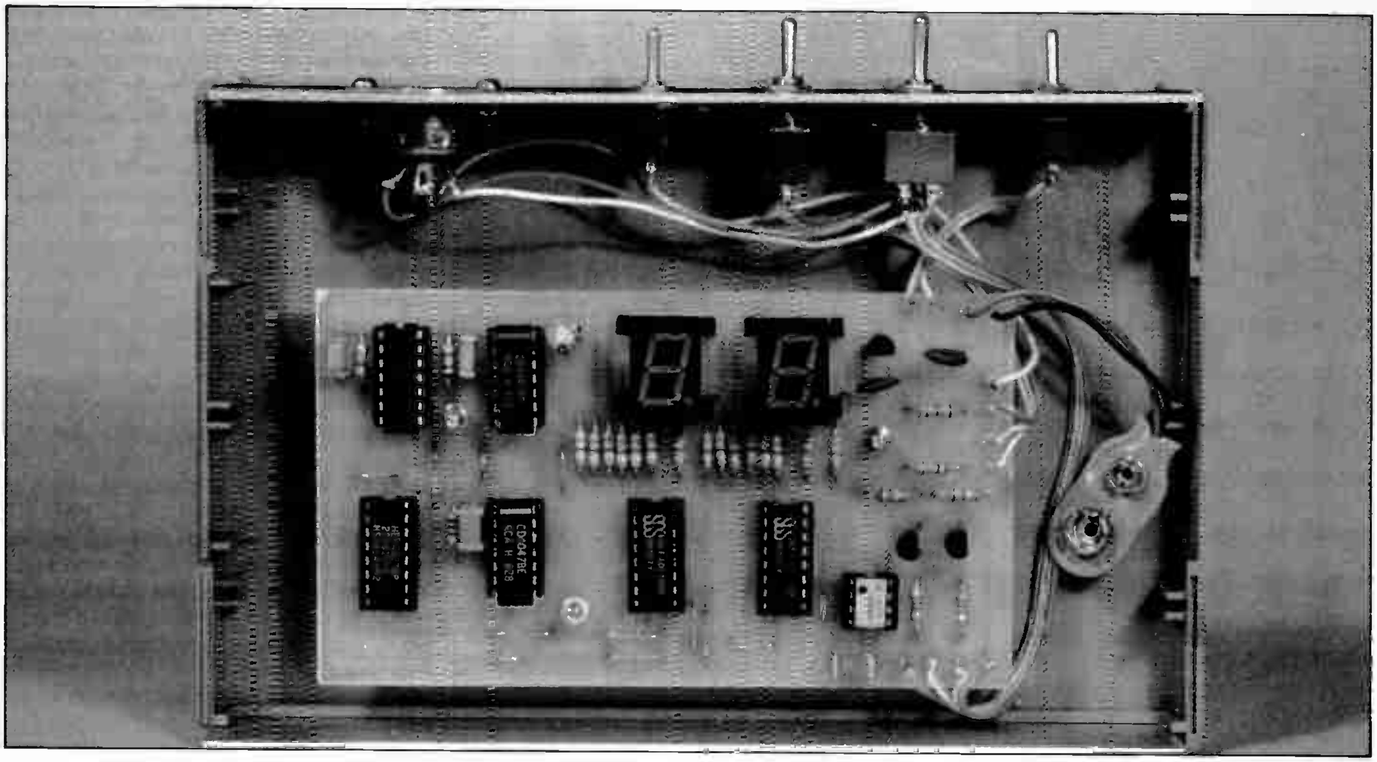
Construction

Apart from the usual off-board components (controls, sockets, and battery) all the components fit onto the pc board, as detailed in the overlay. All the ICs are CMOS types and the usual anti-static handling precautions should be observed when dealing with these. In particular, they should be fitted in sockets but not plugged into the circuit until the unit is completed in all other respects.

It is also advisable to fit the displays in sockets. Apart from eliminating the risk of them sustaining heat damage when they are fitted to the board, this is also advantageous in that it raises them clear of other components on the board. Remember that the displays must be positioned just behind a window cut in the front panel and this will not be possible if other components protrude significantly higher above the board. 12 and 13 mm displays are compatible with the pc board layout but the larger type generally seem to offer slightly higher brightness for a given LED current. The displays must be common cathode types.

A dozen link wires are required and these can be made from the leads trimmed from the resistors. Fit single-sided pins at the points where connections to off-board components will be made.

A plastic or metal case having approximate outside dimensions of 180 x 120 x 39 mm will comfortably



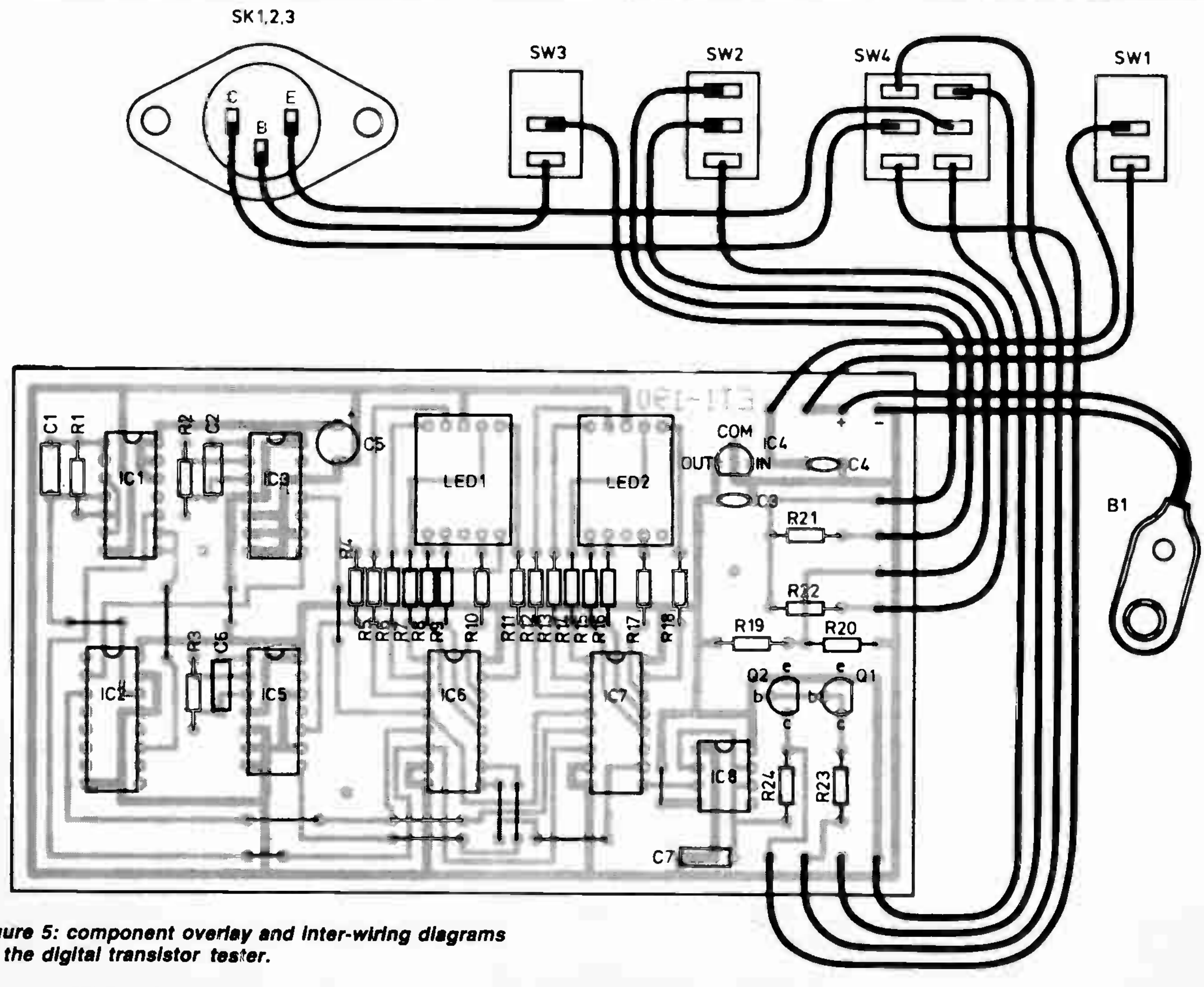
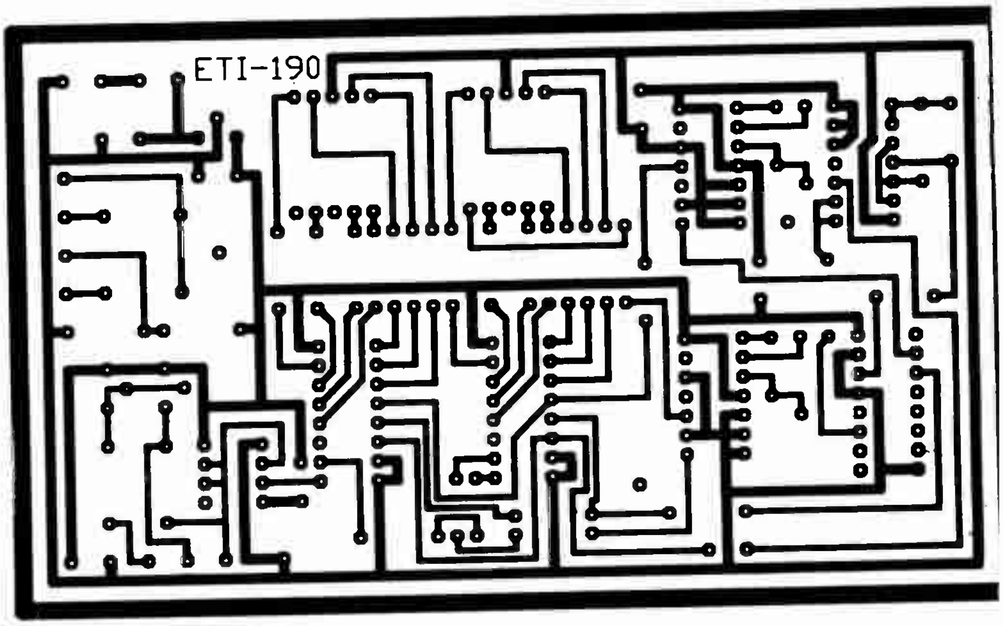


Figure 5: component overlay and inter-wiring diagrams for the digital transistor tester.



accommodate all the components. This assumes a reasonably small battery is used. The current consumption of the unit is around 65mA and if a small battery is used it must be a *high power* or nickel-cadmium rechargeable type.

The four controls and the test sockets are mounted on the front panel but the case is used vertically so that this effectively becomes the top panel. I used a 3-way DIN socket for SK1-3 and most small transistors will connect satisfactorily with one of these. An alternative would be to use three 1mm sockets mounted in the same triangular pattern. Either way, a set of test leads terminated in small crocodile clips will be needed in order to make connections to uncooperative devices.

The pc board is mounted on the rear panel (base) of the case, using 12 mm stand-offs so the fronts of the displays are brought suitably close to the front panel. A window for the displays must be cut in the front panel. This is not too difficult using a coping saw or fret saw to make a rough initial cutout and then carefully filing this out to precisely the required size. Some red display window material is then glued in place behind the cutout.

To complete the unit the hardwiring is added as detailed in the overlay. This is all pretty straightforward and should not give any difficulties.

In use

After giving the wiring the usual final check, switch the unit on and observe the display. This should be an initial random number, followed about half a second later by OO. If this does not happen, switch off at once and recheck the wiring.

Assuming all is well, try connecting a few test devices to the input sockets, remembering to select NPN or PNP, as appropriate. It is unlikely that any damage will occur to silicon devices if the wrong setting is inadvertently tried but greater care should be exercised when dealing with germanium devices. With silicon transistors the leakage currents are generally so low that a OO display should always be obtained when making leakage tests. Any other reading almost certainly indicates that the device under test is faulty.

The situation is less straightforward with germanium transistors, where quite high leakage currents are not unusual. Leakage readings of up to about 8 are quite normal but anything much higher than this would suggest the device under test is of dubious quality. Remember that gain readings must be adjusted downwards by the appropriate amount if a significant leakage level is detected. For example, if a transistor has a leakage level of 6 and a gain of 45 is measured on the x1 range then the true gain of the component is only 39.

The tester can be used to check diodes. With the cathode terminal connected to the collector socket and the anode connected to the emitter socket, there should be an overflow indication with the unit set to the PNP mode. With SW4 set to the NPN position, the display should read zero for silicon diodes and a very low reading should be obtained with germanium types.

Calibration

In common with many transistor testers, this one has no means of adjusting readings for calibration purposes. Provided C2, C7, R2, R21 and R22 all have tolerances of 5% or better, the unit should give good accuracy. The vagaries of transistor gain parameters are such that there is little point in getting too pedantic about the accuracy of a simple transistor tester. Also, to calibrate the unit it would be necessary to have a reliable transistor tester or reference devices having accurately known gains.

Beware of the transistor checkers built into some multimeters. The accuracy of these ranges often seem to be unspecified and the h_{fe} range of my analogue multimeter seems to under-read by about 50%!

The best calibration source is probably a few transistors which have had their gains accurately checked by feeding them with accurate base currents, measuring the resultant collector currents and then reaching for the calculator to work out the current gains. This is the method used to derive the optimum values for the prototype. If you wish to calibrate the finished unit, despite the difficulties involved, trim the value of R2 to give the unit the correct level of sensitivity. ETI

PARTS LIST — ETI-190

RESISTORS

R1	470k
R2	150k
R3	1M8
R4-R18	390R
R19, 20	3k9
R21	680k
R22	68k
R23, 24	220R

CAPACITORS

C1	100n greencap
C2	47n greencap
C3, 4	100n greencap
C5	100µ 25 V radial electrolytic
C6	330n greencap
C7	220n greencap

SEMICONDUCTORS

IC1	4001BE
IC2	4017BE
IC3, 5	4047BE
IC4	78L05
IC6, 7	40110BE
IC8	TLC555P
Q1, 2	BC559
LED 1, 2	0.5/0.56 inch common cathode 7-segment display

MISCELLANEOUS

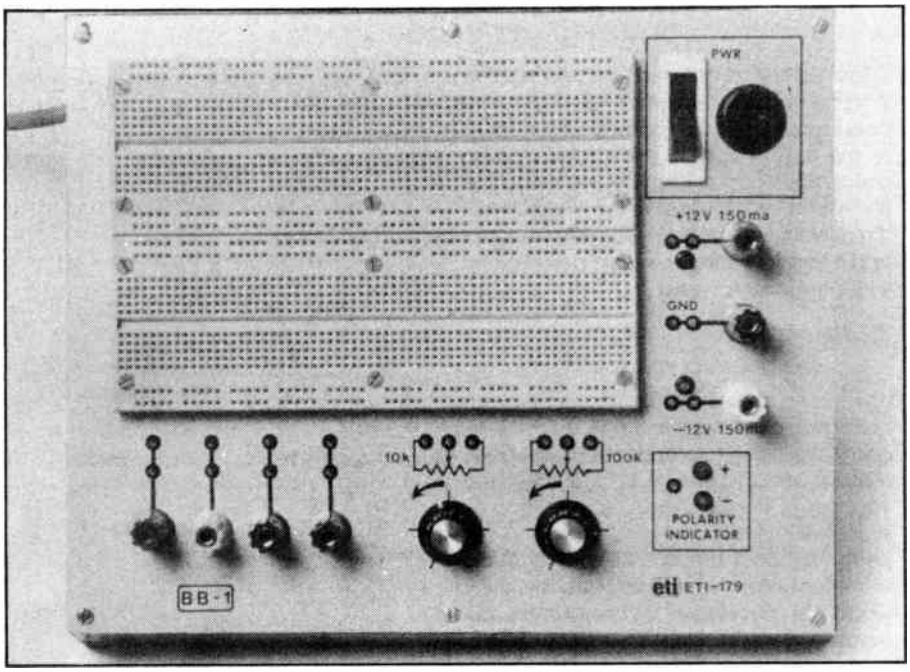
B1	9 V (PP3) battery
SK1-3	3-pin DIN socket
SW1, 3	ON/OFF toggle
SW2	SPDT toggle
SW4	DPDT toggle

PCB Case (180 x 120 x 39 mm). Battery clip ic sockets. Connecting wire. Nuts and bolts.

AN ANALOGUE BREADBOARD

A breadboarding socket on its own is useful, but it needs to be combined with an integral power supply, some hardware and support circuitry if a really versatile system is to be realised.

Peter Phillips



The top view. Front panel artwork for the ETI 179 was too big to fit in the magazine. If you require it, contact ETI readers services on (02) 693 6666.

THE BREADBOARD HAS traditionally been the developmental tool of circuit designers, as the ease of component replacement allows 'fine tuning' of the circuit before its eventual placement on a pcb. Educational institutions often use breadboarding systems for practical sessions of electronics, as component wastage is minimised. However, on its own, a breadboard is simply a means of interconnecting and supporting components. Any external attachments to the circuit, such as test equipment, potentionmeters, switches, the power supply, input signals etc, need to be arranged around the board and connected in whatever way possible. This can result in the traditional 'rats nest', resulting in confusion and circuit errors. More sophisticated circuit development systems offer a breadboard mounted on a box that contains external support for the board. It sound simple, and it is. The problem is; trying buying one that doesn't cost an arm and a leg. This article presents a design that includes all the pcb layouts and panel artwork to allow constructors to build their own, customised system. The design is intended for analogue applications, and features a dual polarity power supply and a sensitive voltage polarity indicator. A digital breadboarding system will be presented later and will include support circuitry peculiar to such a system.

The Circuit Principles

The circuitry of this project is relatively simple. One pcb design (PCB-1) contains the dual polarity power supply and the polarity indicator. PCB2 contains the hardware and the socket pins that enable interfacing between the breadboard and the power supply.PCB3 hold to two on-board potentionmeters and the connections to the outside world. The design is based

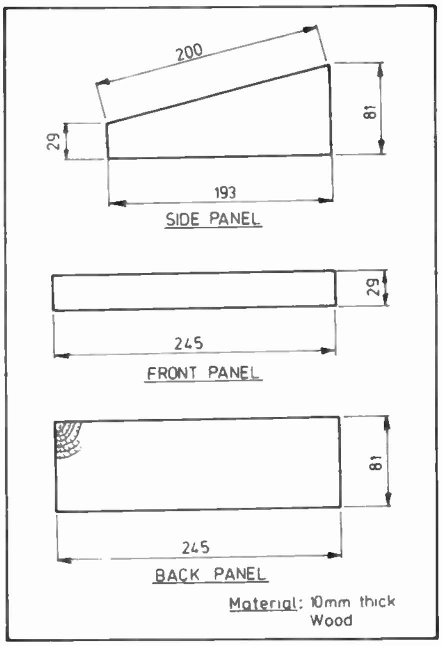


Figure 1: Cutting diagram for the box. This is only really necessary if you want to use a front panel like ours. Otherwise you can dimension the thing to please yourself.

upon my experience of the requirements for analogue circuit development. Cost has been minimised by using garden variety components, which perforce means a relatively basic, but still very versatile unit.

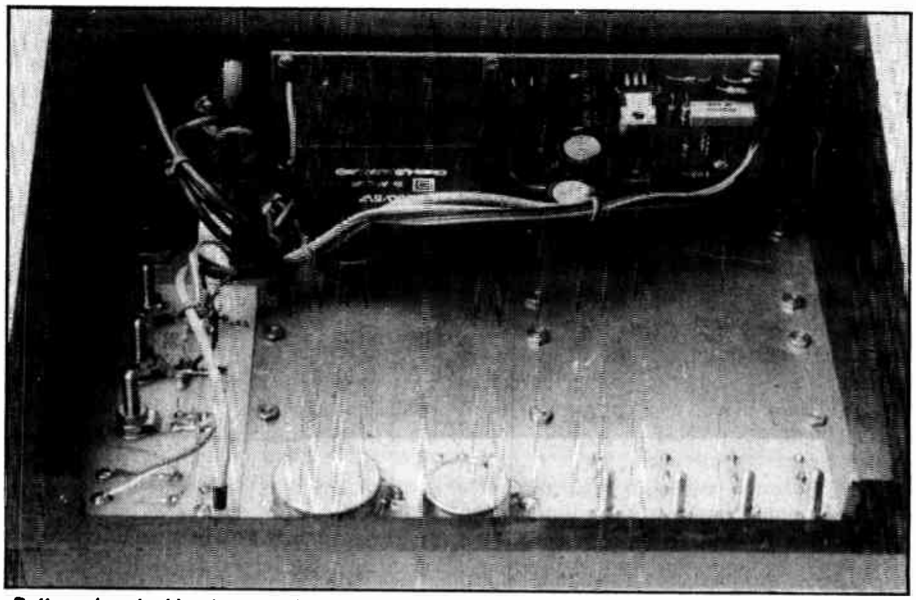
Naturally, readers can add, alter or delete sections as required. For example, PCB-3 could be used to mount switches instead of potentionmeters, or two of these boards could be used, one for the pots, the other with switches. The power supply will not handle currents greater than around 150 mA per side unless a higher VA rated transformer is used. However, in the unlikely event that higher currents are needed, an external power supply can be interfaced via the 4mm terminal posts provided. Heavy leads are connected to the terminals, which then connect to the breadboard via the associated socket pin(s) with 0.6mm wire.

Customising the Design

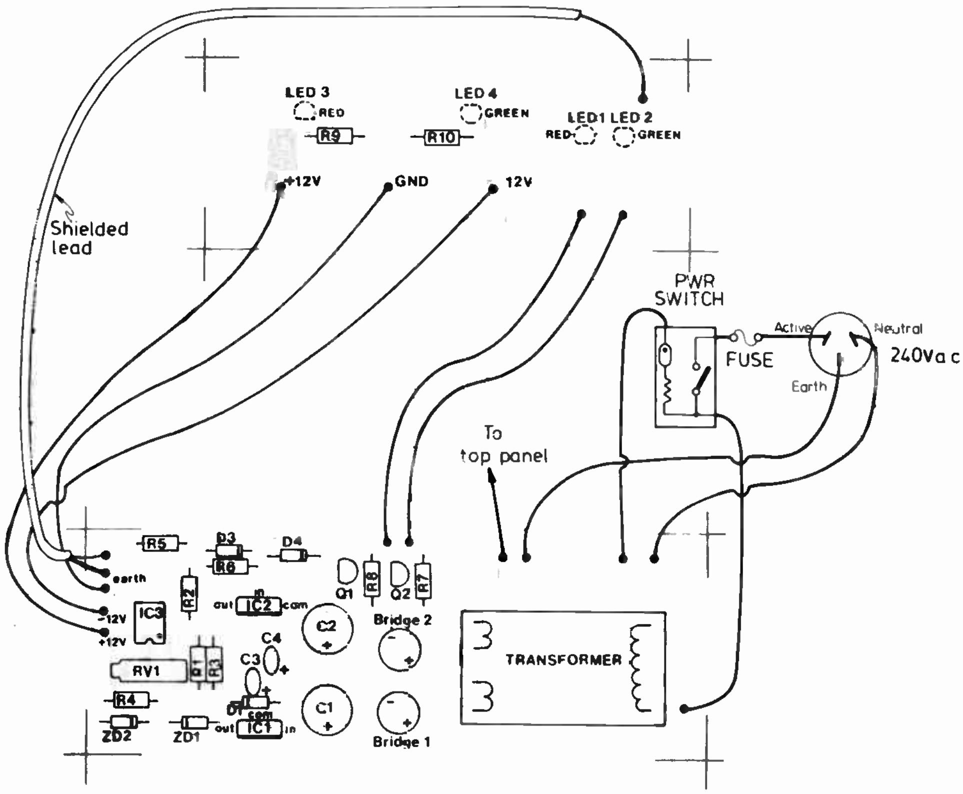
The mounting framework for the prototype, consisting of a wooden box fitted with an aluminium top may suit those with the necessary workshop facilities. However, any equivalent construction would do, although the metal top is recommended to act as a ground plane. A sloping top was incorporated to facilitate viewing the breadboards, but a simpler construction would result from a flat box. The power supply board (PCB-1) was mounted at the rear of the box in the prototype, but can be positioned anywhere it fits. I attached two breadboarding sockets (840 hole size), but one would probably be sufficient for most applications. A bottom cover of timber or metal should be added to the box, and rubber feet will provide stability. To further extend the versatility of the BB-1, mounting flanges for extra pots, switches and devices could also be attached to the unit.

If the supplied top panel artwork is to be used, the layout of the prototype is mandatory. Develop your own artwork and the sky's the limit. There are various methods of constructing the top panel. For example, the top could be made from one large pcb section (245mm x 200mm), etched using the supplied pcb layouts positioned accordingly. The component side could then be painted and screen printed using the supplied design. If screen printing facilities are not available, press-on lettering and hand drawn symbols and lines could be applied. This latter would require lacquering to protect the artwork.

Alternatively, an aluminium top with hand-painted artwork could be used. Treatment of the aluminium top should at least include coating with lacquer, perhaps preceded by polishing and dipping in caustic soda. This latter treatment should be done with some care, as irritating fumes are released during the process. Otherwise, the aluminium could be painted, after application of a suitable primer, and the artwork applied to the painted surface. The prototype was constructed using an aluminium top covered with a Scotchcal panel. The pcbs (2 and 3) were then laid to the underside of the top panel by drilling holes to allow mounting of the necessary hardware through the panel onto the pcbs. This method results in the pcbs being attached by the hardware, allowing trackside accessibility from underneath. An alternative approach is to make each pcb, and then mount the boards on top of the panel. This would require rectangular cutouts in the panel, and fixing with screws at each corner of the pcb. This 'modular' approach, with its inherent flex-



Bottom view, looking towards the back of the unit. PCBI holds the transformer, PCB 2 is to the left, and the pots carried by PCB 3 are almost obscured in the foreground. This view also shows how the box was built. Simple!



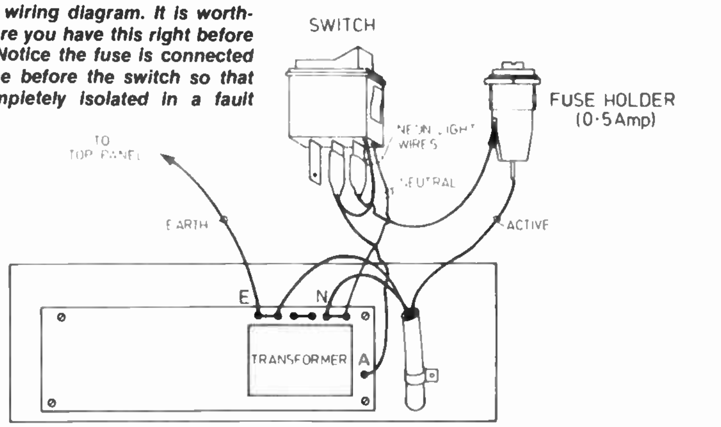
ibility would allow any arrangment sought, while still offering a good-looking end result.

Constructing the BB-1

Commence construction by building the box. Those with a flair for carpentry will probably build a box with mitred corners and a recess to hold the top panel. (The prototype wasn't; just a simple butt connection box, painted matt black, then sprayed with satin enamel lacquer).

The next step is to cut the top panel, trimmed in size to suit the box. The position of the holes and cutouts can be determined either from the artwork or by placing the pcbs and the breadboarding socket(s) into place, and cutting out accordingly. If a Scotchcal front panel or a screen print is being used, this could be applied first, and used as a reference for the various cutouts. Once the top panel has been completed, PCBs 2 and 3, the breadboarding socket(s) and the power switch (and fuse if used) can now be mounted in position. If the pcbs are mounted as in the prototype, that is, beneath the top panel, held by the hardware, it is likely hole alignment may not be exact. For this reason, you may prefer to drill the top panel according to the artwork, then drill the pcbs using the top plate as a template. Getting perfect alignment is impossible, but with care, good results can be obtained. This part of the construction is relatively time consuming and fiddly, as care must also be taken not to damage the top panel artwork, or to create any short-circuits between the panel and the components passing through it.

Figure 2: Mains wiring diagram. It is worthwhile making sure you have this right before you switch on. Notice the fuse is connected in the active line before the switch so that the unit is completely isolated in a fault condition.



PCB-1 should now be built, as per the layout diagram, but should be fitted only after all wiring, testing and adjustments have been completed. This pcb was fitted to the prototype using spacers and self-tapping screws into the timber back. If a bigger transformer than the specified one is used, it should be mounted directly on the case, earthed and connnected with leads to PCB-1. If a power supply with increased power characteristics is used, heatsinking the regulators will be necessary.

The choice of the power switch should be made with some care as it is likely this switch will get a lot of use, requiring a rugged switch if reliability is to be achieved. Use of a miniature switch is not recommended. The illuminated variety was employed as it provides a positive indication that the power is on.

Four LEDs are incorporated on PCB-2, two adjacent to the +12 and −12 terminals, two more for the polarity indicator. Pin-point LEDs are specified; green for both negative voltage indicators, reds for the positive indicators. They mount directly onto the pcb, through a 5mm hole drilled in the top panel. A piece of 4mm black plastic tubing over the bottom of the LEDs will enhance their light output and protect the leads from shorting to the panel. Current limiting resistors R9 and R10 for LEDs 3 and 4 should be soldered on the track side of PCB-2. The 4mm terminal posts on both PCBs 2 and 3 should be soldered directly to the pcb track with their connecting lugs. If VCU style potentionmeters are used for PCB-3, they can be soldered to the pcb lands directly by forming the lugs to suit. The flat section will be incorrect to suit a screwed dial knob, requiring another flat to be filed onto the shafts. Metric style pots overcome this problem, but will require connection with wire to the pcb lands.

The earth wire from the mains is terminated on PCB-1, and is subsequently connected to the ground terminal post of the unit via the power supply wiring. However, a separate 7 amp rated wire should be connected to the aluminium top panel. This not only provides protection, but serves to ground any noise pickup. The pcb mount transformer is not earthed, as it is double insulated. However, if a conventional mount transformer is used, it should also be earthed.

Construction of the pcbs is straight forward; just follow the layout diagrams and watch component orientation when building them. The IC socket pins associated with PCBs 2 and 3 can be obtained either by sacrificing IC sockets, or by purchasing insulated IC socket strips. The gold insert, machined variety are recommended to stand up to the type of use envisaged. (Those used in the prototype were supplied by George Brown). Molex pins would work, but are unlikely to give reli-

PARTS LIST

RESISTORS — All ¼ watt 10% unless otherwise specified.
— all values in ohms.

R1, R2, R5, R6 .... 10k
R3 .... 220k
R4 .... 68k
R5, R6, R7 ....
R8, R9, R10 .... 1k

POTENTIONMETERS RV1 ... 10k, 10 turn trimpot
RV2 .... 10k linear, panel mount
RV3 .... 100k linear, panel mount

CAPACITORS
C1, C2 1000
C3, C4 .... 2.2 tantalum.

SEMICONDUCTORS
Q1 .... BC547 or similar
Q2 .... BC557 or similar
D1, D2 .... IN4004 or similar
D3, D4 .... IN914 or similar
ZD1, ZD2 .... 5V6 400mW Zener diode
IC1 .... uA7812, T0220 voltage regulator
IC2 .... uA7912, T0220 voltage regulator
IC3 .... uA714
Bridge 1 and 2 .... W04 or equiv.
LED 1 and 2 .... Pinpoint red LED
LED 2 and 4 .... Pinpoint green LED

SWITCHES
S1 .... 240 volt mains switch (illuminated)

TRANSFORMER
T1 .... PCB mount, PL30/5 VA or equiv.

MISCELLANEOUS
PCB or vero board; Scotchcal front panel; timber for case; light gauge aluminium sheet 220mm x 245mm; 7 x 4mm terminal posts; 20 insulated IC socket pins, gold insert, machined variety; 2 control knobs; 4 pcb supports, rainbow cable hook-up wire, 240 lead and plug; cable clamp; lugs, 840 pin breadboard socket(s); fuse holder; 0.5 amp fuse.

able long term service in this application. The socket pins solder directly to the pcbs, through a 3mm hole in the top panel. The interconnections between PCBs 1 and 2 are shown in figure 3. Note particularly that a shielded lead is required to connect the polarity indicator input on PCB-2 to its circuitry on PCB-1.

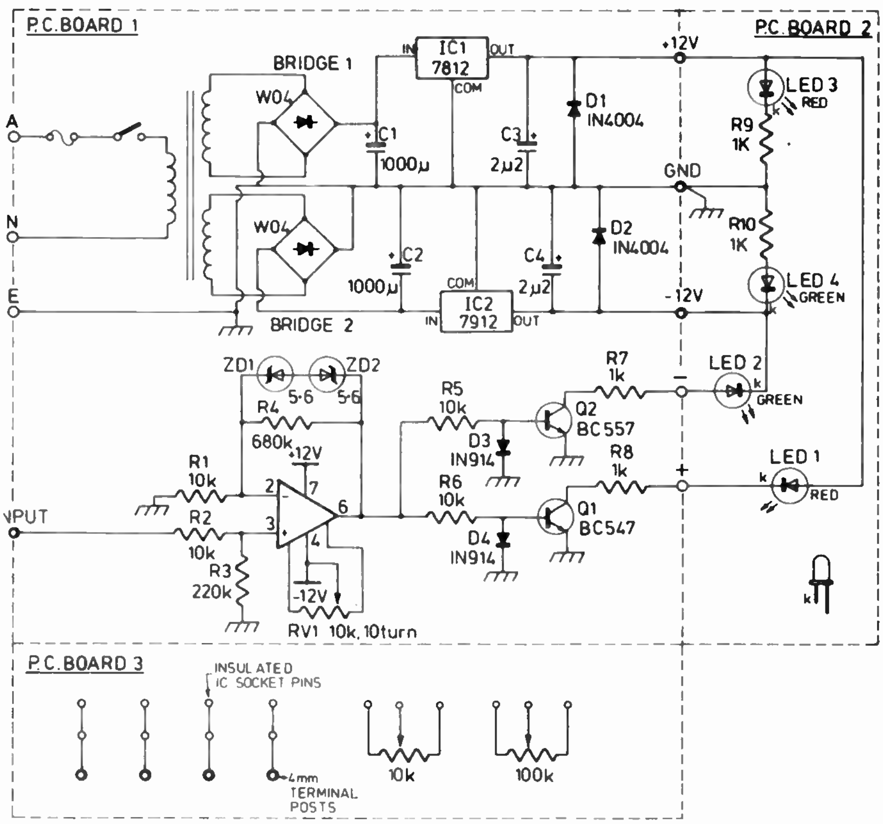
Commissioning

Once all wiring is complete, and final checks confirm that PCBs 1 and 2 have all components correctly polarised, apply power to the unit. The two voltage indicator LEDs should light to show the presence of the +12 and −12 volt rails. If not, determine if these voltages are in fact present. A likely problem is incorrect orientation of diodes D1 and D2, or of the regulators. Note that the 7800 series has different connections to the 7900 series, but that orientation on the pcb is the same for both devices. If voltage is present, check the LED orientation. If all is well, it remains to adjust the offset for the polarity indicator. This should ideally be done with a voltmeter connected to pin 6 of IC3 to measure the dc output voltage. Connect the polarity indicator's input terminal to ground, and adjust RV[7] until the dc voltage at pin 6 is exactly 0 volts. Allow a few minutes for everything to stablise before performing this adjustment. Once this is complete, both LEDs associated with the indicator should be out. Applying a positive input voltage greater than 10mV should cause the red LED to turn on; similarly, the green LED should light for an equivalent negative input voltage. The maximum input voltage should not exceed 24 volts. Finally, install PCB-1 into position and tie the wiring neatly into looms.

Using the BB-1

The BB-1 is designed to facilitate the development of analogue circuitry, by providing the necessary support to the central object of the unit; the breadboarding area. Next month will provide some interesting practical op amp circuits for you to experiment with, but some important do's and don'ts are worth mentioning now. The breadboard sockets are the most expensive item of the unit, and require special treatment if they are to last. It is important to only ever insert leads and wires into the sockets that do not exceed around 8.7mm. Telephone wire is useful as interconnecting wire, and a range of colours is recommended in various lengths. Try to use wires that are the right length to keep the circuit neat and to prevent random noise pickup. Develop a colour code standard, e.g., red for positive rail, white for negative, black for ground, and only use wires with clean bared ends. Broken pieces of wire lodge in the breadboard, caused by re-using old wires, will quickly render the socket useless. Ideally, a pair of wire strippers should be on hand to bare the wire without causing a nick in the wire when it is stripped. Bare the wire to expose around 5 to 6 mm only; too

ETI-179 How It Works

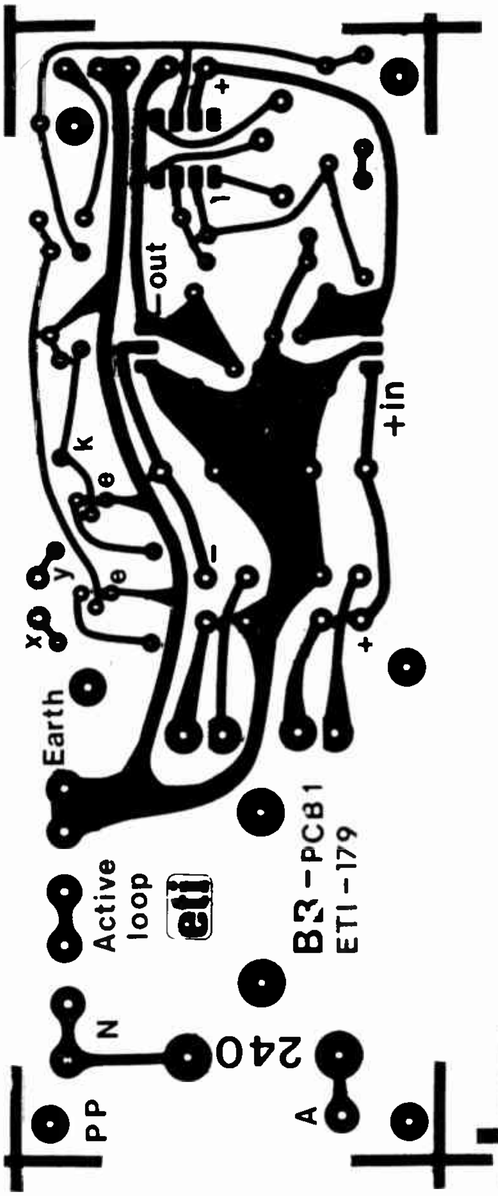


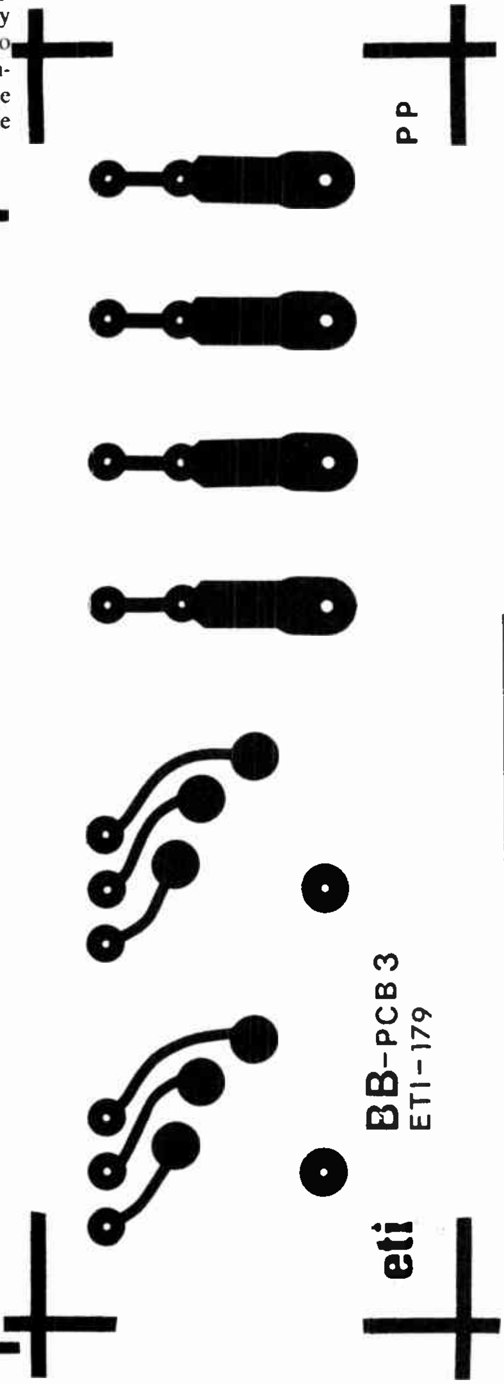
The electronics for this unit are mainly contained on PCB-1, although indicator LEDs and outputs/inputs are on PCB-2. The dual polarity power supply uses two 12V, three terminal regulators, supplied by two diode bridges in turn connected to the isolated secondary windings on the transformer. Capicators C7 and C2 filter the bridge outputs. Capacitors C3 and C4 improve the transient response of both regulators, and should be tantalums. The diodes D1 and D2 allow reliable start up of the regulators by eliminating any reverse voltages that may be present at the regulator outputs. The output voltages areestablished at +12 V and −12 V (within 5%) with respect to ground, and are applied to PCB-2 for subsequent connnection to the breadboard. The transformer secondary voltages are 15V ac each, and the suggested transformer has a rating of 5VA, limiting the available current per side to around 160mA. At this current, no heatsinking is needed for the regulators. Varying the output voltages by using different regulators would require a suitable transformer matched to the required voltages.

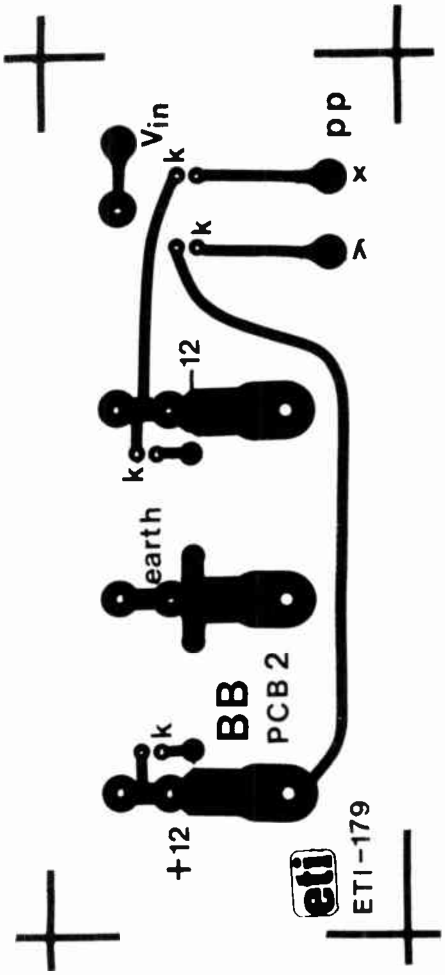
The polarity indicator is based on a 741 op amp, connected as a high gain, non-inverting amplifier. R1 and R4 set the gain at around 68, and ZD2 and ZD2 limit the output voltage of the 741 to plus or minus 6V. This prevents saturation problems, and minimises the effect of skew thereby allowing a quick response. The indicator LEDs 1 and 2 are driven by transistors Q1 and Q2 which operate when the output voltage of the op amp increases above 0.6 V. Q1 requires a positive voltage to operate, supplied when the input to the op amp exceeds +10 mV. Similarly, applying −10 mV or more to the input will cause Q2 to switch on. Diodes D3 and D4 protect the transistors against reverse cónduction. The input resistance of the circuit is around 200k ohms, allowing the circuit to be used to monitor voltages in most applications. R3 is used to limit random noise pickup when the input is left open-circuit. The maximum input voltage is 24 V. Offset adjustment is provided by RV1 to ensure the output voltage of the 741 is zero volts when the input is connected to ground.

long may cause short-circuits with adjacent wiring, too short may cause the insulation to enter the socket, preventing contact. Dirty or oxidised wire should not be used as it will have a deposit on the internal sockets.

Use only ¼ watt resistors and capacitors that have leads not exceeding 0.7 mm diameter. Also ensure the leads are clean and straight, as bad connections causing noise can otherwise result. Do not let components get hot, as this will eventually cause the internal sockets to loose their spring tension. ICs can be inserted by 'rolling' them into the board, but they should be removed with an extractor to protect your fingers and to minimise damage to the IC pins. Finally, switch off the power when modifications are being made to the circuit, as, although the regulators are current limited, excessive current may cause rapid overheating of the transformer. Note also that the 4mm terminal posts associated with the power supply are for **output** only; do not apply an external voltage as this could damage the regulators. Interfacing any external signal, voltage or device should be done via the four terminal posts associated with PCB-3. However, the ground terminal post can serve as a connection point to other instruments and circuits. ●

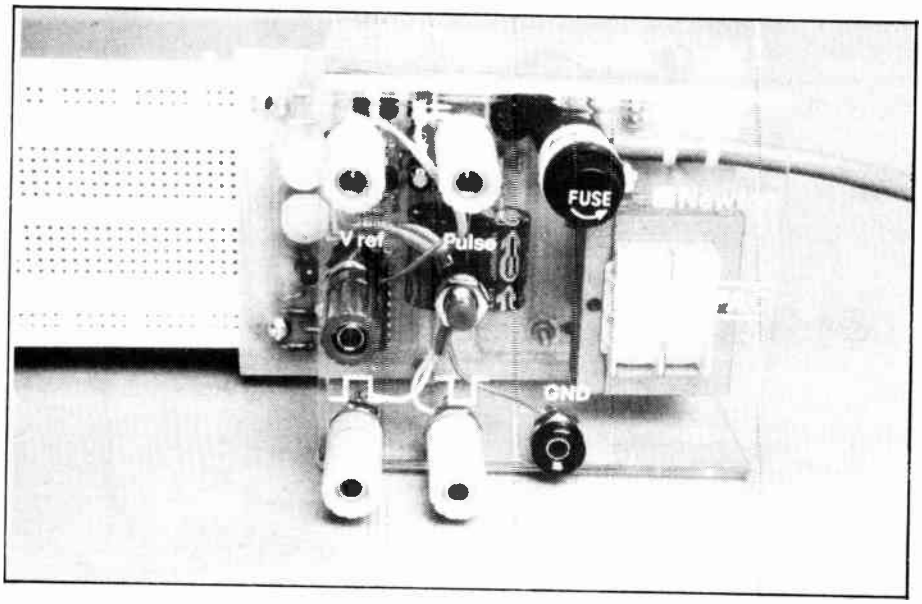






ETI-187 PROTOPAC

— A Low Cost Analogue/Digital Protoboard System



THE DEVELOPMENT AND testing of electronic circuits is usually a costly and time consuming business. Firstly, the system needs to be prototyped in one way or another, power supplies connected and test signals applied. Wires dangle around the test bench and connections to the circuit are often far from ideal. In order to simplify the prototyping stage of the design process this low cost protoboard system was developed.

Primarily the project was conceived as a training aid that could help those new to electronics develop the skills required to design and implement both analogue and digital circuits. Since the cost of a development lab is so high it was thought that more may be convinced to try their hand if an economic alternative was available. What better way to do this than constructing your own system, learning in the process, and getting a versatile piece of lab equipment in the end.

Initially the system specifications were developed in consultation with staff members of the electronic engineering section of the School of Information Sciences and Engineering, Canberra College of Advanced Education, and the Advanced Technology and Engineering Centre (ATEC). Their aim was to provide final year engineering students with a low cost system which could be taken home to allow them to develop circuits in their own time.

This meant that the system had to be very versatile and provide facilities for both analogue and digital design. In the end the following features were decided upon:

— mains operation
— +5 Volt power supply
— +12 Volt power supply
— −12 Volt power supply
— Variable reference voltage −11 to +11 Volts
— variable frequency clock with two phase output (180° apart) variable over 2.5 decades
— debounced pulser providing
  a) normally high output
  b) normally low output
— +5 or +12 volt operating levels for the clock and pulser, link selectable.
— power on LED indication

Design Philosophy

The aim of this project was to provide the greatest number of features for the lowest cost. In order to comply with this the construction shown in Figure 1 was chosen. This is a departure from the normal prototyping system in that the electronics is not housed in a box underneath the protoboard. The greatest advantage of this arrangement is that the +5, +12, −12 and ground connections can be made directly from the pc board to the protoboard. This saves the user having to wire power to the board every time it is used, and saves the cost of four extra connectors. A clear perspex cover was developed to house the electronics and hold the connectors. This is available as part of the kit associated with this project.

Although this project was developed for the student or hobbyist, it is such a convenient way to prototype that it has come into general use in our development labs over and above the more expensive "boxed" systems.

Construction

Using the component overlay given, begin with the five links. You will have to decide which level will be needed for the clock and pulser. If you use TTL (5 V), then configure the system for +5 volts by placing link 1 as shown. The long link beside the IC should be insulated to prevent any chance of shorting to the connector wires. Next insert the resistors, then trim-

PARTS LIST — PROTOPAC

Resistors	All ¼ Watt, 2%
R1	680
R2	2.2K
R3,R4	1K
R5,R6	510
R7,R8	10K
RV1	5L trimpot
RV2	200K trimpot
Capacitors	
C1, C2	470µF 25 Volt electrolitic
C3, C4, C8, C10	0.1µF monolithic chip
C5,C6,C7	10µ 16 Volt electrolitic
C9	1nF greencap
Semiconductors	
D1	Red LED
DB1	Bridge rectifier
IC1	µA7812
IC2	µA7912
IC3	µA7805
IC4	4049, Hex CMOS inverter
Q1, Q3, Q4	BC549 or equivalent
Q2	BC557 or equivalent

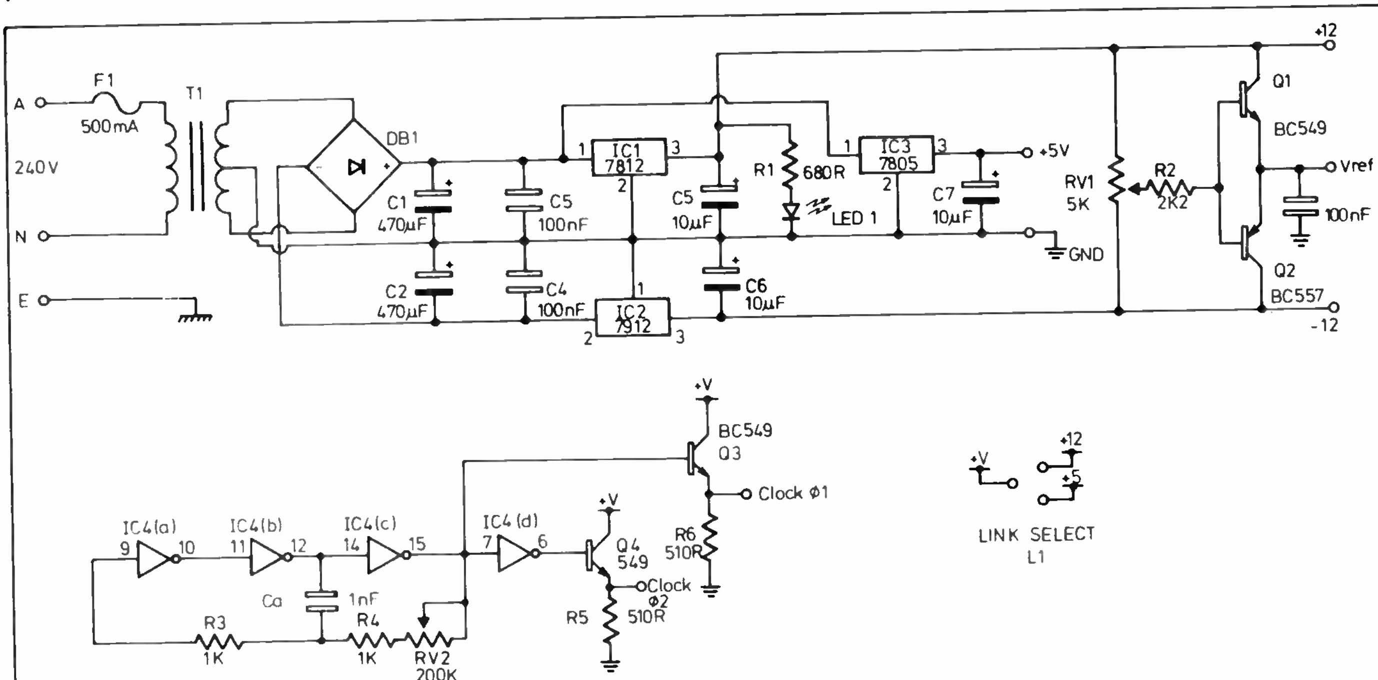
Miscellaneous
Perspex cover, 500mA fuse, fuse holder, pushbutton switch, power cable, printed circuit board, 240VAC protective cover, nuts and bolts, 5 banana socket terminal posts, heat shrink tubing or fuse boot, breadboard, cable ties, transformer, 4mm socket, piece of ribbon cable, solder 1µg.

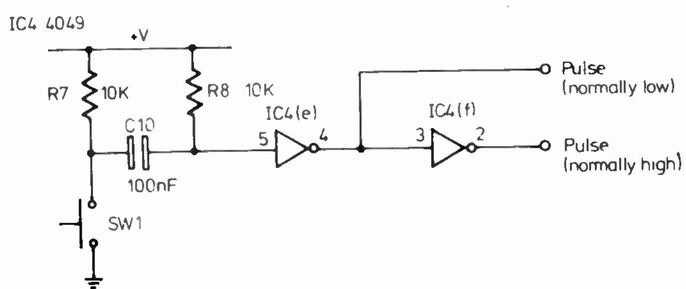
pots, IC, bridge, capacitors, regulators and transistors. This method works on component height above board, allowing each level to be held in place whilst soldering. The IC is CMOS, so take the usual precautions of not charging yourself up before handling it. If you use different transistors to those specified, ensure the E, B and C are as shown. Peel back and strip both ends of the ribbon cable. Insert one end into the eight points on the board. The board is labelled, with P for pulse normally LOW, P for pulse normally HIGH and d01, 02 are the two clock phases. Use four pieces of stiff wire for the power rail connections, and trim them off to about 6mm. These will push into the four power busses on the breadboard. Solder the transformer in and trim the 240 volt pins back.

The next section deals with the 240 volt wiring, so TAKE CARE.

Make sure there are no loose ends or exposed wires. Solder the neutral (blue) to the transformer connection point as shown and trim back. The power cable can be attached to the board using two cable ties. Solder a length of brown power cable to the other connection point. Now liberally coat the entire 240 V section with Silastic Adhesive Sealant (available at hardware stores). Place the protective cover over the sealant and screw down. If sealant does not ooze out everywhere, you have not used enough! The earth lug can be attached to one of the screws, and the earth wire (green) connected. Put the board aside and assemble the cover.

The red binding post should be used for Vref. Insert the fuseholder, switch and other connectors. Now back to the board. Wire each of the eight ribbon wires to their respective connectors, and switch, following the diagram. Slide a piece of heat shrink tubing, or a rubber fuse boot





HOW IT WORKS — PROTOPAC

The mains 240 volts is connected to the transformer with the active through fuse, F1. The output of the transformer is fed to a standard bridge rectifier circuit and capacitors C1 and C2 provide ripple filtering. C3 and C4 are high frequency bypass capacitors (chip monolithic types) to reduce high frequency components entering the regulators. IC1 is a three pin 12 volt regulator, IC2 negative 12 volt and IC3 5 volts. C5, C6, and C7 give additional filtering. The power on LED is driven via the 12 volt supply through current limiting resistor R1.

The variable power supply is made up of RV1, R2, Q1, Q2 and C8. RV1 provides the initial reference voltage and is then buffered through the simple push-pull arrangement which follows. C8 gives additional high frequency filtering.

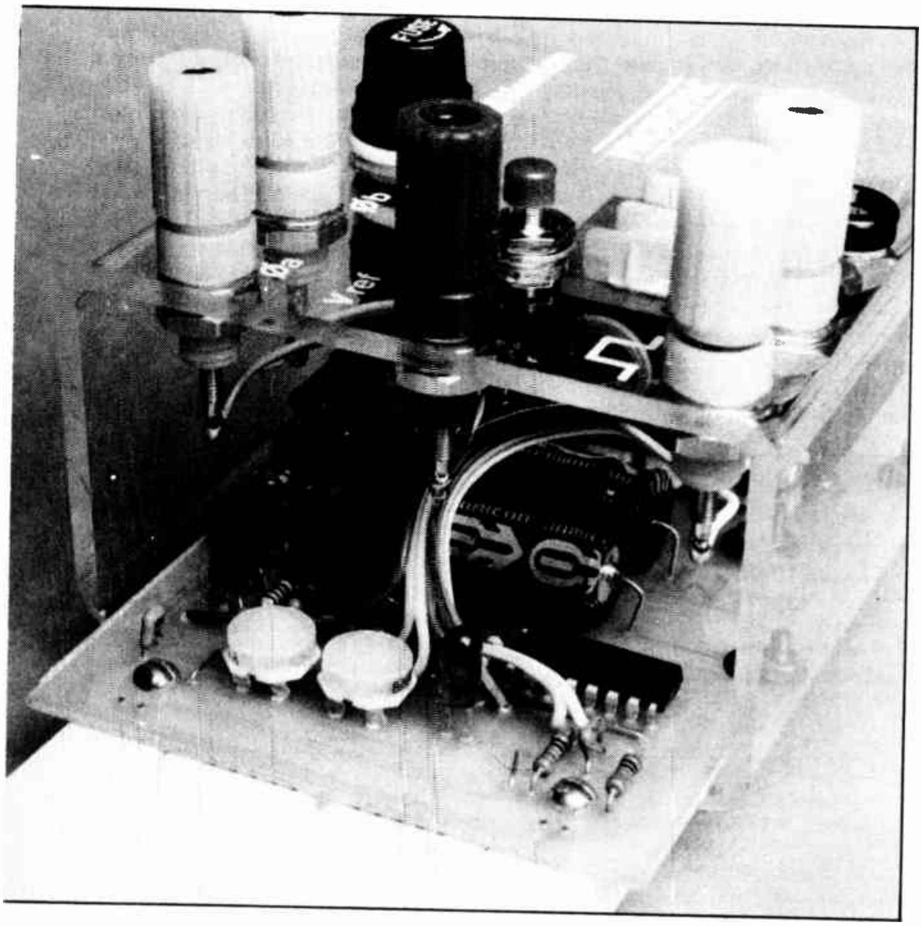
The clock has been constructed from three cross coupled inverters of IC4. The frequency of oscillation is approximately given by the equation:

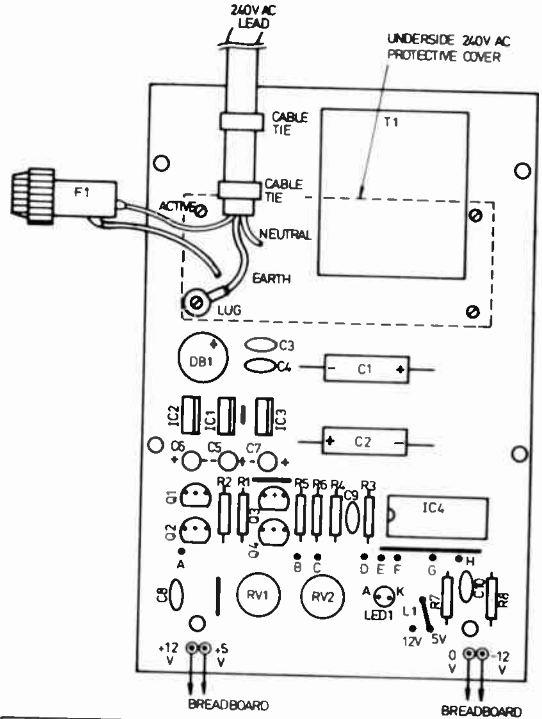
$$f = \frac{1}{2 \times Ra \times C9 \left(\frac{0.405\,R3 + 0.693}{R3 + Ra} \right)}$$

where Ra = R4 + RV2

By slotting in various values of Ra, R3 and C9 a frequency range of your own choice may be found. Q3, Q4, R5 and R6 are emitter follower buffers to provide greater output sourcing current.

The pulser is formed with R7, C10, R8, IC4(e) and SW1. The time constant formed by R7-C10, and R8-C10 give charge and discharge times that smooth the voltage applied to the input of OIC4(e) as the switch bounces until the switching threshold of the inverter has been reached. The time constants give an output pulse width of approximately 1ms. IC4(f) inverts the output of IC4(e) to provide a normally high output.





TERMINAL KEY
A — V REFERENCE TERMINAL
B — CLOCK ΦA "
C — CLOCK ΦB "
D — GND "
E — PULSER SWITCH TERMINALS
G — PULSE NORMALLY HIGH TERMINAL
H — PULSE NORMALLY LOW TERMINAL

over the two brown active wires and solder them to the fuse holder. Push the insulator up over the connections and heat or cable-tie in place. Position the board inside the cover and screw the two together. Finally push the Protopac into the breadboard and screw in place.

Testing

Once the circuit has been constructed, give it a careful review. Look for solder bridges and bad joints on the non component side, and loose wires or suspect connection on the component side. Pay strict and careful attention to ALL associated mains wiring, which is potentially lethal.

Examine the orientation of all the semiconductors making sure that the regulators, CMOS IC, transistors, LED and bridge rectifier are all inserted the correct way (as per the overlay diagram).

Make sure the fuse has been inserted and then turn the power on. The LED should then light up. If it doesn't — don't worry, turn the power off and get your multi-meter ready. Measure the voltage across the +12 rail and the ground rail. If it reads +12 volts and the LED still does not light then it is probably placed the wrong way around, so reverse it. If there is no voltage present check that the positive terminal of the bridge rectifier is in the correct position.

Once the power rails have been checked the variable reference voltage can be tested. Simply connect a multimeter to the output terminal and vary the trim potentiometer with a screwdriver. If the output does not vary then check that the transistors are inserted the correct way around.

The clock and pulser sections all run off the one IC. If they are both not operating then IC4 is probably to blame, so check its orientation. If the output from the pulser is OK but the clock output is not running look into the circuitry associated with the output buffers (Q4, Q3, R5 and R6).

Using the system

Using the system is easy. Plug it into the mains socket and power is applied to the board. Note that the power rails need to have wire links placed halfway down the board if you wish to distribute power to the second half.

Electronic components such as integrated circuits, resistors, capacitors, etc can now be plugged directly into the board. Connections between components are made with small pieces of solid core copper insulated wire sometimes called Bell wire to Telephone wire). The best place to get this is, you guessed it, multicore telephone cable. The rest is up to you. Happy developing! ●

WHERE TO BUY THE PROTOBOARD SYSTEM

A complete kit of parts for this project is available only through:

Applied Audio Consultants,
GPO Box 733
CANBERRA CITY 2601
Ph: (062) 43 3345
Fax: (062) 47 0985
KIT PRICE $79
(plus $5 postage and packaging expenses)

The kit comes complete with tinned printed circuit board, breadboard, pre-drilled and screen printed perspex cover, and all components necessary to construct the project. A full set of instructions is also supplied.

NOTE: The printed circuit board and design are copyright and the property of Applied Audio Consultants and may not be copied or used without permission.

ELECTRONICS
ETI — 1419

Like many projects, this one grew from a need, created by a recent re-organisation of my overcrowded workshop. An essential part of any workshop is a sound system, both as a means of entertainment and as part of the test gear. For nearly 25 years (is it that long??) a valve amplifier has faithfully fulfilled this role for my workshop, but, apart from a deteriorating performance, it was taking up too much room. So the search began for a simple, small equivalent. Research showed that a plethora of amplifiers are available — either as kits or as ready built modules. But, I wasn't sure if these amplifiers would have the same sound as the old valve unit. Also, there was no challenge.

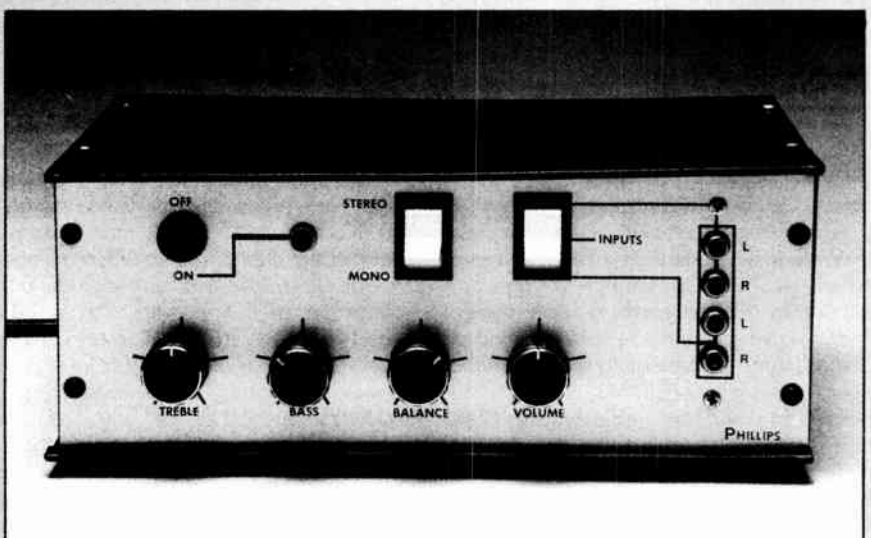
Whatever I came up with, costing and performance had to be competitive. After all, re-inventing the wheel is not viewed favourably by either editors or readers, and therein lay the challenge. Slowly some thought crystallised, and further research confirmed that there was a way by using ideas that (as far as I can tell) have been out of fasion for many years. Basically, the design uses three ideas that collectively do not appear to have been integrated before, but which individually are fully tried and tested. The first idea was to use Sanken Power modules as the outputs. These modules have been around for years, and have proven their worth as being both reliable and rugged. The question was, can you still get them? At the time of writing, George Brown confirmed that the type I wanted is currently available for around $20, (10 watt module) and that supply was no problem.

The next two ideas came as a result of the circuit development process. Like most projects, this one had its genesis on a breadboard. Basically, all I had to do was develop a tone control stage, as the Sanken modules had everything else, including sufficient gain to operate directly from a typical signal source. I researched the field to find that contemporary circuits all use an active tone control configuration. This method places the tone control circuitry as part of the feedback around an amplifier, with component values chosen to give the required boost or cut for both treble and bass controls. So, I commenced using a design that borrowed from various ideas, and soon had a circuit up and running. But, no matter what I did, I could not get the sound I wanted. I was able to perform an A — B test with the old valve amplifier, and it always won. Then, with an uncharacteristic burst of insight, I figured that if I wanted the same sound as the valve amp, I should use the same circuit. And that's idea number two.

The third idea grew after all my concentration on valve circuitry. Older readers may recall that in the 'good old days' magazine projects always built the circuit on tag strips or terminal strips — that is, using point to point wiring. While this method is fussy, it meant that anyone could do it. Today, the mini-

A BLAST TO BEAT THE PAST

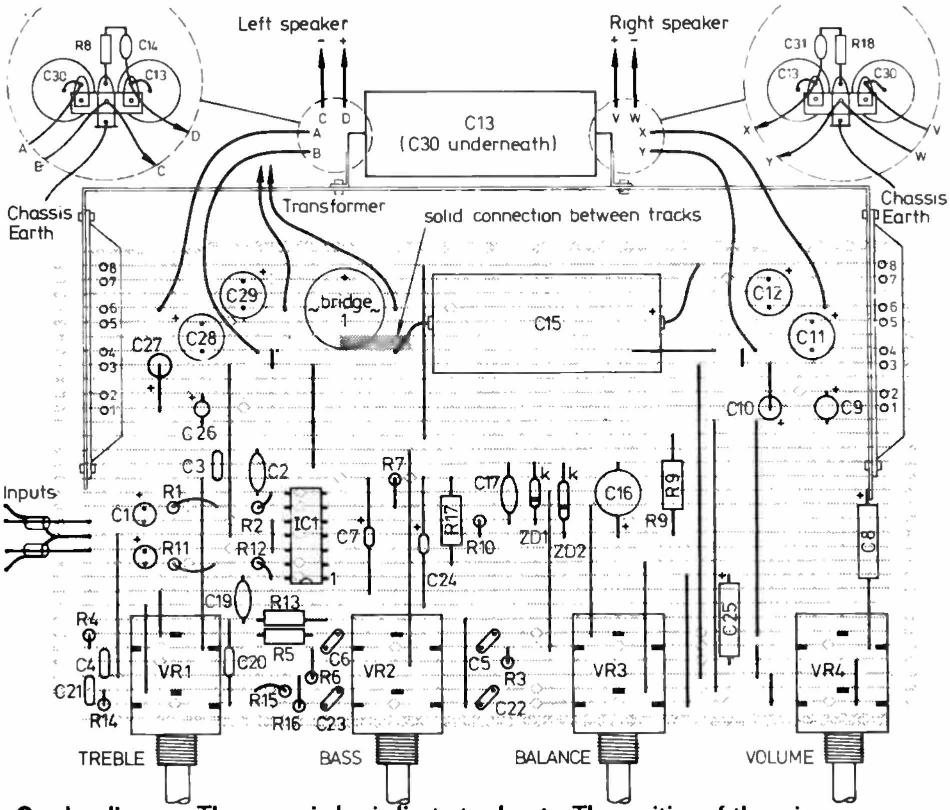
From the workbench of Peter Phillips comes a 10 watt per channel amplifier, costing around $75, which doesn't require a printed circuit board, but has a gut thumping performance that is reminiscent of valve amps.



Specifications:
Measured using 8 ohm resistive loads.
Output Power = 9.6 W per channel, 1 kHz, 8 ohm load, (both channels driven together).
Input for full output = 1 Vp-p. (Volume control at max)
Frequency Response = 20 Hz to 60 kHz (within 3 db) Tone controls to mid posn, output = 1 watt.
Tone Controls
Bass = +12 db and − 9 db at 40 Hz (boost-cut, compared to 1 kHz)
Treble = +15 db and − 18 db at 10 kHz (boost-cut, compared to 400 Hz)
Output impedance = 0.2 ohm, output = 1 watt
Harmonic distortion at full power = 0.5% max.
Approximate cost = $75.00, including transformer.

mum requirement for any but the most simple projects is a printed circuit board, which is great if you can make your own, or if it is available as a stock item. Because the circuit for this amplifier is essentially quite simple, I decided to build it all on strip-board (Vero board) and to present it in this form.

So, collectively, the final result is an amplifier that will 'knock your socks off' even though it uses the most basic components and a piece of strip board costing around $4.00. Demonstrations of the unit have resulted in high praise indeed — and I now have no reluctance to pension off the valve amplifier, as this one compares in every way. More importantly, I don't have to update to a more expensive set of speakers, as the boost frequencies from the tone controls compensate admirably.



Overlay diagram. The open circles indicate track cuts. The position of these is not critical provided the components all stay on the correct side.

About The Amplifier

These days, applications for a stereo amplifier that can operate into cheap speakers abound. A tuner, a video recorder, or a second sound system for Mum, (or whoever) are common applications. Other reasons include electronic keyboards, computers (such as the Apple 2GS with its sound synthesiser) and even fully blown hi-fi systems with a room equaliser that requires rear speakers. This amplifier has the right sort of bass and treble boosting compensate for a cheap set of speakers, so you win all round — a cheap amplifier with cheap speakers. Naturally, the amplifier will also perform well with good speakers, as distortion is extremely low, and frequency response very high.

But why does the tone control circuitry of this amplifier have a different sound (a better one, I believe) to other circuits? Certainly, sound quality is a subjective phenomenon, and has been the subject of much debate. The argument of solid-state versus valve amplification often involves reasoning more akin to a witch-doctor's explanation of bone pointing. One dedicated individual of my acquaintance even asserts that 20% carbon resistors are essential for true valve sound. It is probable that most people could not tell the difference between various amplifiers operating under 'flat' conditions, given controlled listening tests. But, fiddle the frequency response by rapping up the bass and treble controls, and each different amplifier will take on a characteristic as easily discernible as one set of speakers to another. The amount of boost, the boost frequencies and the shape of the response curve all contribute to the effect. Of course, purists will by now be shuddering at the very thought of listening to sound that is not 'flat', although it is arguable that most speaker systems exhibit humps and bumps in their response curves.

There are various reasons why boosting both ends of the audio spectrum is desirable. Compensating for speaker deficiencies is the most typical, as is low volume listening. The latter reason is to compensate for the ear, as low volume sound often sounds thin and lifeless. Another reason is to enhance certain effects, a liberty taken by many FM radio stations, particularly those with an emphasis on pop music. So, I reckon, if you are going to take musical license, then do it with the best effect. This amplifier uses high impedance voltage amplification with a passive tone control circuit, giving a close approximation of a solid-state equivalent to the traditional valve pre-amp circuit. As a result, the bass is big and the treble bright. There is a distinct lack of harshness, and distortion seems to be minimal, allowing hours of fatigue free listening. Convinced? Build it and see for yourself — it works!!

> ***"After all, re inventing the wheel is not viewed favourably by either editors or readers, and therein lay the challenge."***

The tone control circuit used in this amplifier is not my design, (I confess), but one that in various forms appeared with many valve amplifier circuits of the 1960s. Because of its losses, it needs amplification both before and after it. The circuit is intended to operate into an impedance of 220k, easily obtained with a valve, but difficult with a transistor amplifier. The solution here was to use operational amplifiers, although the spectre of a dual polarity power supply is consequently raised; a complexity I wanted to avoid. However, by using RC coupling and mid-supply biasing, single supply operation has been achieved.

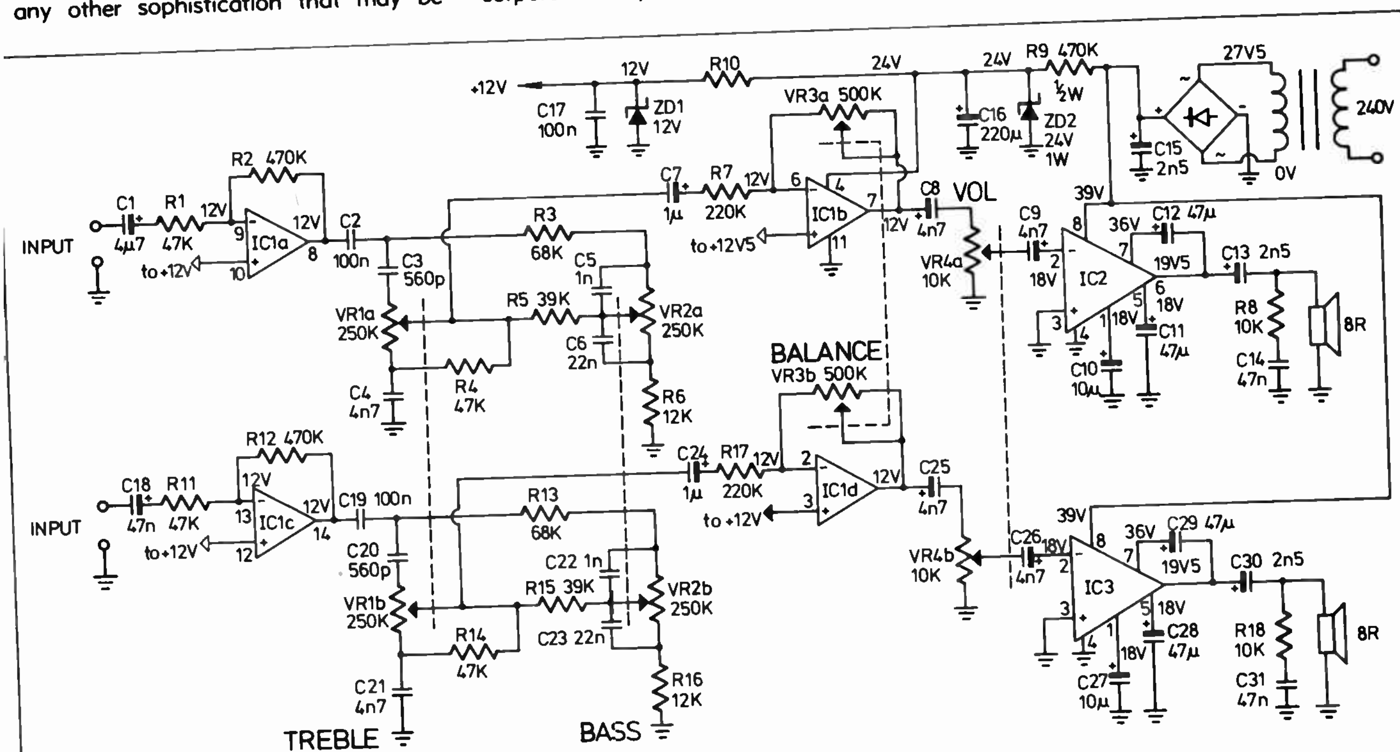
The Sanken power modules are the real 'guts' of the circuit, as virtually everything else is incorporated within them. Unlike other IC power amplifiers, the Sanken module is a hybrid unit, enclosed in an aluminium housing that allows direct mounting on a heat sink. They are physically large compared to an IC, and require only four externally connected capacitors to become operational. The specifications include 0.5 % distortion at full power, and a power bandwidth of 20 Hz to 20 kHz. However, I found that at a normal listening level, the bandwidth extended up to 60 kHz; impressive by any standards. These modules are available in various power ratings; 10 W, 20 W, 30 W and 50 W. I chose the 10 W size as this keeps things simpler and cheaper, and 10 watts per channel is more than enough for a domestic situation unless you have very inefficient speakers. Readers requiring more power could use the 20 W module, although a power transformer with a higher voltage and VA rating (37 V, 1.5 amp) would also be needed. The higher voltage would require the value of R9 to be raised to around 1k (1 watt minimum, as 0.9 W would be dissipated). Also, the heat sink should be increased if full power applications are envisaged. I have not researched the other higher power modules, but their cost effectiveness may make them less attractive than other currently available circuits.

The whole amplifier in its presented form is basic, but can be extended if required to include input selection and any other sophistication that may be needed. Under normal listening conditions, heat generation is low enough to permit the enclosure to be a plastic case or a timber cabinet, although any suitable size aluminium case is probably the easiest and cheapest. If you intend operating the amplifier at loud volumes for any length of time, the size of the heatsink should be increased or the case incorporated as part of the heatsink. The complete amplifier shown is an example of how I adapted the amplifier module to make a complete unit, and may give some ideas. There are all sorts of ways to make a suitable front panel, including application of press-on lettering direct to the selected front panel, or, as in the example, with Scotchcal aluminium. I suggest the use of a metal front panel, electrically connected to the circuit earth to minimuse hum pick-ups as it earths the casing of each potentiometer.

> ***"A no-frills workhorse, with all the facilities necessary, including the right sound"***

Construction

The design is based on the strip-board unit, catalog number H-5612, available from Dick Smith, and features tinned copper strips with an alpha-numeric grid to uniquely identify each location. Any similar board is suitable, and the alpha-



How It Works

IC1 is a quad, FET input op amp, and all four non-inverting inputs are tied to a fixed 12V supply, filtered by C17, and regulated by ZD1 and R10. This sets the quiscent output voltage of all four amplifiers to 12V, in turn making all the inverting inputs equal to 12V. The op amp is supplied by a 24V supply, regulated by ZD2 and R9, filtered by C16. Because of the quiescent dc voltages, RC coupling is used throughout the amplifier. IC1A and IC1C are set to give a gain of 10, with an input impedance of 47K, established by R1 (and R11). The tone control circuit follows, in which RV1 gives treble boost or cut, and RV2 bass boost or cut. If both controls are set to mid position, the frequency response is essentially flat, although a square wave will exhibit slight variations from the ideal at frequencies around 1kHz. The output of the tone control circuit is coupled to the next stage by C7 (and C24), which, because of the high impedances used, need to be tantalum for low leakage. RV3 is the balance control, and operates by varying the gain of IC1B and IC1D from zero to approximately unity in mid position, and 2 for either extreme. If more gain is required from the circuit is intended to operate into an resistance of 220k, and lowering either of these resistor values may affect the operation of the tone control circuit. RV4 is the volume control and supplies signals to the output modules.

The output modules have a gain of approximately 40, with bootstrapping applied via C12 (and C29). The remaining components connect sections of the internal circuit to ground. The network R8 and C14 (R18 and C31) are suggested by Sanken, presumably for stability and transient suppression. The output coupling capacitors, C13 (and C30) can be 1000uF values, as recommended by Sanken, but I used 250uF to get the best low frequency response.

numeric grid can be written on the board as required. If the chosen board is not tinned, clean it thoroughly with steel wool to ensure easy soldering. Commence by marking on the track side each of the 42 track cuts required using the track-cut diagram/listing. Use a 3mm drill bit in a slow speed drill to cut each track and remove any swarf on completion.

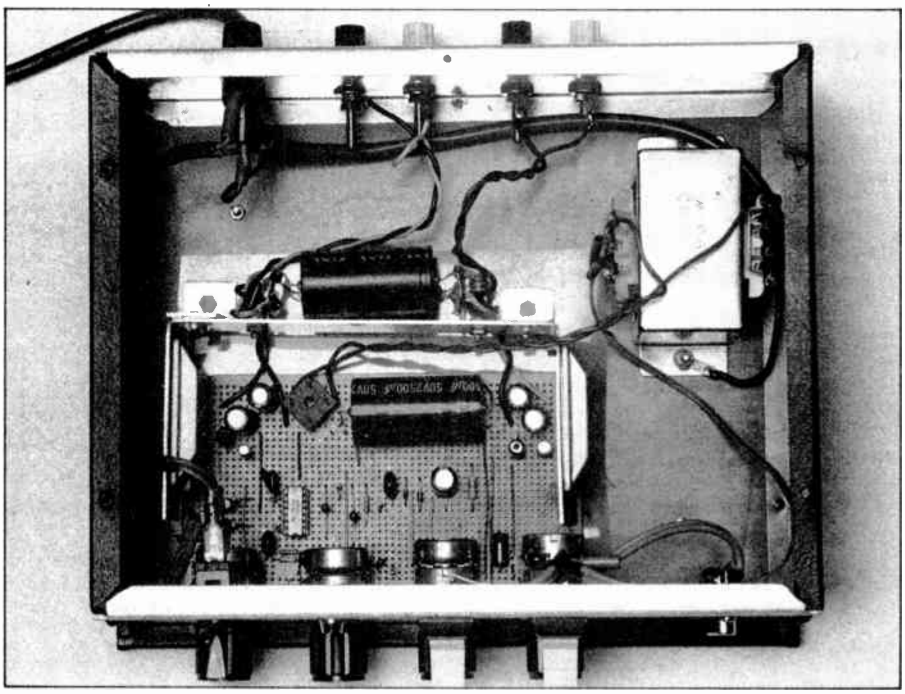
Insert the 23 track links next, using insulated tinned copper wire. It will be useful to write the alpha-numeric grid on the component side of the board to help identifying each point from this side. When fitting each link, bend the excess along the track, rather than sideways before soldering, to minimise adjacent track short-circuits. The diagram showing the links also shows the points for each track cut, this time as viewed from the component side. Check carefully for any misplaced cuts or links, as one error can make life difficult.

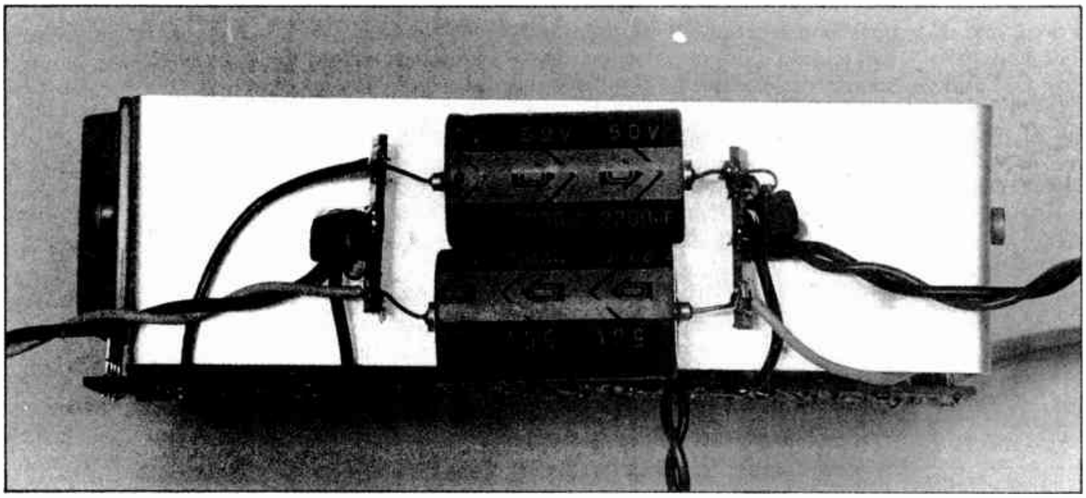
Then mount the resistors, carefully following the layout diagram. Mount them all before soldering, again bending the leads along the track. Once the resistors are soldered in, mount the IC socket and start inserting the capacitors. Fit several before soldering, as this helps to locate them and minimise mistakes. the next task is to fit the potentiometers. Attach 1mm diameter tinned leads to the lugs, and arrange them to fit into the required holes. Mark the holes to ensure all six leads will be correctly located then fit each pot in turn.

At this point, preliminary testing is recommended, as the possibility of error with this kind of construction is more likely than for a pcb. Attach the input shielded leads, then insert the IC. Apply a dc supply (25 V to 40 V) between earth and the top of R9 (end facing the rear of the board), and check that the supply current is around 15 mA to 20 mA. Connect an input signal, preferably from a signal generator, and confirm that signal is present at the volume control. If possible, check that all controls are working by observing the waveform with a CRO, or by listening to the sound with a signal tracer. Confirm that both channels are operating equally, that is the same amount of treble and bass control occurs, and the amplitude of both signals is the same when the balance control is set to its mid position. A square wave set to a frequency of around 400 Hz should exhibit considerable leading edge overshoot for treble boost, and the bass control should alter the slope of the top and bottom of the wave.

With everything working so far, mount the bridge rectifier. Note that tracks Y and Z are earth, and, although already joined at two other points, should now be connected together with a very solid connection between the negative terminal of the rectifier and the negative end of C15. Lay a short piece of wire between the tracks, and run solder to form a continuous connection. This connection minimises hum due to the charge current for C15 taking paths shared by the inputs to the power modules. Next connect the ac supply leads and the output leads checking carefully that there are no adjacent track shorts due to a single strand. It may be necessary to enlarge the holes for these leads with a 1mm drill.

The final task is to mount the power modules and the heatsink. Enlarge, with a 1mm drill, the holes in the strip board





Above: The top view of the amplifier mounted in its case. The transformer is at the bottom left, along with output terminals.

Left: A rear view of the module itself. The power modules are at either end of the aluminium strip, which serves as a heatsink, apart from carrying the power supply capacitors.

required to accept each module, and form the legs of the module to match the hole spacing by bending each odd numbered lead to form a dogleg towards the next even numbered lead. The even numbered leads do not need more than slight reforming. Note particularly that both modules face the same way and the left hand module faces in. Make the heat sink from a strip of aluminium measuring 45mm by 280mm. Bend the strip at right angles 68mm from each end, then drill it to match the holes in each module. Also drill two holes to attach the tag strips required to support the off-board components. Attach the heatsink, then solder the module terminals to their respective tracks, making very certain that no shorts occur between the tracks. The tag strips and their respective components can now be fitted, and the final wiring completed.

PARTS LIST — ETI-1419

Resistors:	all ¼ watt unless otherwise stated
R1, R11, R4, R14	47k
R2, R12	470k
R3, R13	68k
R5, R15	39k
R6, R16	12k
R7, R17	220k
R8, R18	10
R9	470 ohm ½ watt
R10	2k7
Potentiometers:	all duel ganged (available from Jaycar)
RV1, RV2	250k log (C curve)
RV3	500k linear (A curve)
RV4	10k log (C curve)
Capacitors:	All 50V electorlytic, pc mount unless otherwise stated
C1, C9,	
C18, C26	4.7uF
C8, C25	4.7uF axial
C2, C17, C19	40.1uF polyester
C3, C20	560pF Ceramic
C4, C21	0.0047uF polyester
C6, C23	0.022uF polyester
C5, C22	0.001uF polyester
C7, C24	1uf tantalum
C10, C27	10uF axial
C11, C12, C28, C29	47uF
C13, C30, C15	2500uF — axial
C14, C31	0.047uF polyester
C16	220uF
Semiconductors:	
IC1	TL074 (or uA774)
IC2, IC3	SANKEN SI-1010G (available George Brown)
ZD1	12 volt, 500mW or greater
ZD2	24 volt, 1W
Bridge 1	1.2A full wave bridge (WO4 or similar)
Transformer:	240 to 27.5V, 1A. Type 6672 or similar

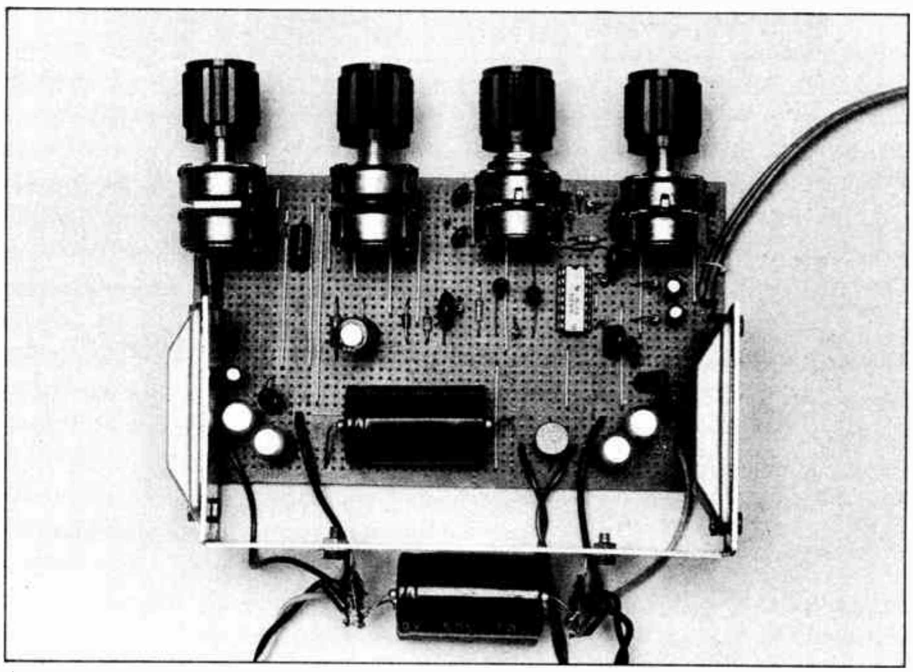
Miscellaneous:
Strip board, 90mm × 150mm (Dick Smith — catalog H-5612); thin guage aluminium, 45mm × 280mm; two × 3 lug tag strips; single strand insulated wire; hookup wire.

Before connecting the transformer, verify by measuring the resistance across C15 that there are no shorts between the power supply rail and earth. Also, do a final check that everything is as it should be, then attach the transformer and a pair of speakers and apply power to the unit. If all is well, and the fire extinguisher is not really needed after all, a thump at switch on should indicate both output coupling capacitors charging. Touching both input leads should also give the characteristic hum and the controls should all operate properly. There should be no hum or noise of any kind when the volume control is turned full anticlockwise. The power modules run slightly warm, as does R9. Obviously, if there is a significant rise in temperature of the heat sink, there is something wrong. The voltages shown on the circuit diagram may assist in any fault finding, although correct dc voltages still occur if the fault is an interruption in the signal path.

A Compete Unit

The example shown of a complete unit uses a home made aluminium box, dimensions 95mm × 260mm × 200mm (h × w × d) fitted with a Scotchcal front panel. Because the amplifier is for workshop use, the inputs are on the front panel, with selection of either pair accomplished with a 2 position, 2 pole piano-key toggle switch. A mains on-off switch, a LED power indicator and a stereo-mono switch complete the top row, all located above the amplifier controls. A 1 amp fuse in the mains lead (active), is fitted to the rear panel, along with the speaker output terminals. The box is earthed to the amplifier by having one speaker terminal (the earth of the pair) connected directly to the panel. Although probably unnecessary, I arranged everything so that this was the only earth point to the case to ensure no earth loop currents. Naturally, the box and transformer casing are earthed to the power point in the usual manner. The LED is operated by the transformer using the 27.5 V and 24 V tappings, (to give 3.5 V) with a diode and a 100 ohm resister in series with the LED. The stereo-mono switch simply connects both inputs together, which is somewhat rough, but useful in the planned application. Two brackets from the heat sink to the case bottom support the rear of the amplifier module whilst the front panel supports the controls. A no frills workhorse, with all the facilities necessary, including the right sound.

It is planned to present in a future article a pcb version of the tone control circuit of this amplifier, but integrated with a 4 input mixer. The power output section will be up to the reader, so if a more sophisticated unit is required, keep tuned-in to these poges. Otherwise, enjoy the sound of the 'sixties from this unit.

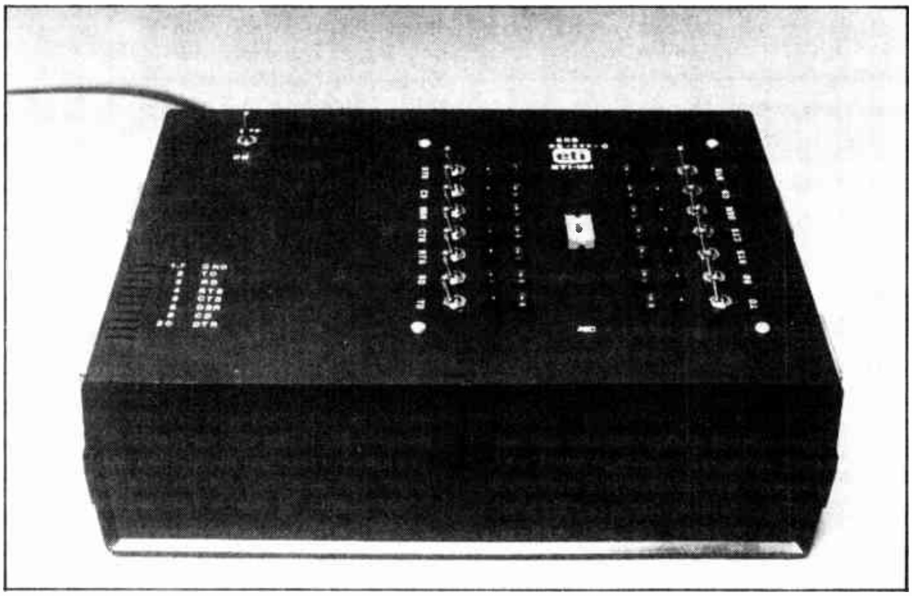


A view of the module from the top. The pots are conventional panel mounting ones, secured to the board by a bit of stiff wire, (off-cuts from resistor leads work just fine). You can use special circuit board mounting pots if you wish, but there are problems both of supply and price.

ETI 181: RS 232 BREAKOUT BOX

The BreakOut Box (BOB) is one of the fundamental pieces of test equipment for testing RS232 ports. This is a simple but flexible version of the idea.

This project was developed by the staff of All Electronic Components, Lonsdale St, Melbourne.



THE RS-232-C STANDARD greatly simplifies the interconnection of digital devices. When something goes wrong at an RS-232-C interface, the 25 signal lines and two equipment configurations — Data Terminal Equipment (DTE) and Data-circuit Terminating Equipment (DCE) — can easily confuse the trouble shooter. The RS-232-C BreakOut Box (BOB) is a piece of test equipment used to simplify RS-232-C trouble shooting.

BreakOut Boxes can be simple or very complex. In its simplest form a BreakOut Box displays the polarity of the voltage on an RS-232-C line. Fortunately, many of the problems generally encountered can be diagnosed with a relatively simple BreakOut Box.

Average BOB

This project describes the construction and use of BOB, an "average" BreakOut Box. It features:

— A male and female DB-25 connector at each end of the unit,

— A patch bay that any of the 7 most important lines at one end allow to be patched to any of the same lines at the other end,

— The ability to set any lines to +12 V or −12 V, and

— Two coloured LED's to indicate the polarity of the voltage on each line.

The circuit is simple. Each of the signal lines (2, 3, 4, 5, 6, 8, and 20) are taken from the DB-25 connectors to the DIP socket in the centre of the BOB. The DIP socket acts as a patch bay. Any line from one pair of connectors can be patched to any line on the other pair. Each line is attached to a SPDT Centre-off Toggle switch. One position of the switch sets the line to +12 V; the other position sets the line to −12 V. The centre position lets the line float at the voltage applied to it by the devices attached to the BOB. Polarity indicating LED's with current limiting resistors display the polarity of each line.

A ±12 V power supply is built in to this project, so that if set voltage levels ±12 V are required on RS-232-C lines, they are provided.

Construction

Begin construction by inserting all links on the pc board, follow with the resistors, and then the PC Stakes. DO NOT at this stage insert the LED's or Switches.

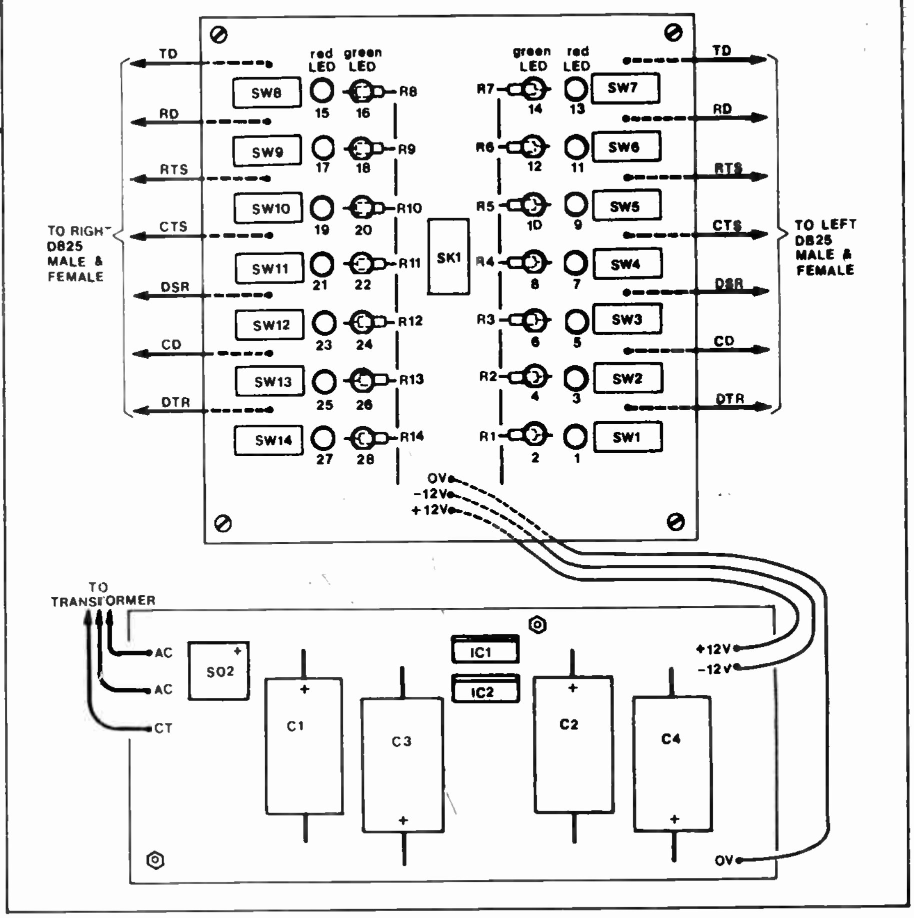
Mark off the four holes from the front panel and printed circuit board, making sure that they are aligned. The front panel is located in the centre of the case — leave a few millimetres clearance from the vents in the top of the case.

Now drill the holes for the switches and LED clips (6.2 mm), and cut out the rectangular section for the DIP Socket. Clip in the LED clips and insert the LED's, making sure the polarity is correct.

The switches may now be mounted, ensuring that they are all at the same height. At this stage the wires to the pc board should be connected, leaving enough length to allow them to go to the DB25 connector and power supply. Next, cut holes for the DB25's on the sides of the top half of the case, fit the DB25 and connect the wires from the pc board. One male and one female DB25 are wired in parallel. Watch the pin numbers.

Place the four ½" spacers between the pc board and lid, and at the same time, guide the LED leads, DIP Socket, and switches through the pc board. Insert the four bolts through the front panel, case, spacers and pc board and tighten with nuts supplied. Finally, solder the DIP Socket, LED's and switches on the back of the pc board.

The power supply is straight-forward. Assemble all the components on the pc board, and place the transformer on the left-hand-side base of the case. Bring in the mains cable through the back panel and keep it close to the bottom of the



case. Connect active and neutral wires to the transformer, and the earth wire to the case of the transformer.

Now place the power supply board next to the transformer, using 10 mm spacers to keep it clear of the case. Connect the secondary of the transformer to the appropriate terminals. Now the power rails from BOB may be connected to the power supply.

BOB is now ready for use.

Potential problems:

Before using BOB, you should understand two circuit shortcomings. Firstly, BOB draws current from the circuit under test, and secondly, unless care is taken BOB can create short circuits.

BOB draws current from the circuit under test to light the polarity indicating LED's. By using current from the circuit being monitored, the need for active buffers and transistor switches is eliminated. However, it means that BOB loads the RS-232-C device connected to it, and in extreme cases, this loading may interfere with circuit testing. If this is suspected, use an oscilloscope to check circuit voltages. If positive voltages are much below 12 V, or negative voltages are much above −12 V, the circuit is overloaded.

It is possible to switch a line to +12 V on one side of the patch bay, patch it across to the other side, and then switch it to −12 volt. This will short-circuit the power supply. It is also possible to cause a short-circuit if an attempt is made to set a line that the equipment under test is setting. This is even more serious, since it might damage the equipment. However, if care is taken, you will encounter no problems with BOB.

Using the Break-Out Box:

To begin, make yourself familiar with the names of the various lines, as this will give you an insight to these functions.

It is also valuable to know which type of device, DCE or DTE, generates each signal. When you are familiar with the RS-232-C interface, and familiar enough with BOB to understand its limitations, you will be able to begin using it. It is impossible to anticipate every configuration of BOB that will be useful; however, descriptions of a few common problems and how BOB is used to solve them can help you learn.

Activating DCE:

It is often necessary to test a modem or other piece of data-circuit-terminating equipment (DCE) without having it attached to a computer.

To activate a DCE, lines 20 and 4 must be set to +12 V. To do this, use BOB with no jumpers in the patch bay. Set all of BOB's switches in the centre (off) position. Connect the DCE to one of BOB's DB25 connectors, and set the switches for pins 4 and 20 to the +12 V position. Be sure to use the switches on the same side of the patch bay of the DB25 that you are using. Now, assuming that the DCE is a modem, you should see a carrier at its analogue output. By setting the switch for line 2 (TD) to +12 V it should be possible to change the tone coming from the modem.

LED's should indicate that the modem is setting pins 5 and 6 to +12 V. These lines are used to activate equipment attached to the modem. If they are inactive, or remain at −12 V, then the device attached to the modem might never send any data. If the modem does not respond correctly, then it is not operating as a standard DCE device. Further troubleshooting is in order. If the modem does not respond as predicted, you know that any problems you have are caused by some other piece of equipment.

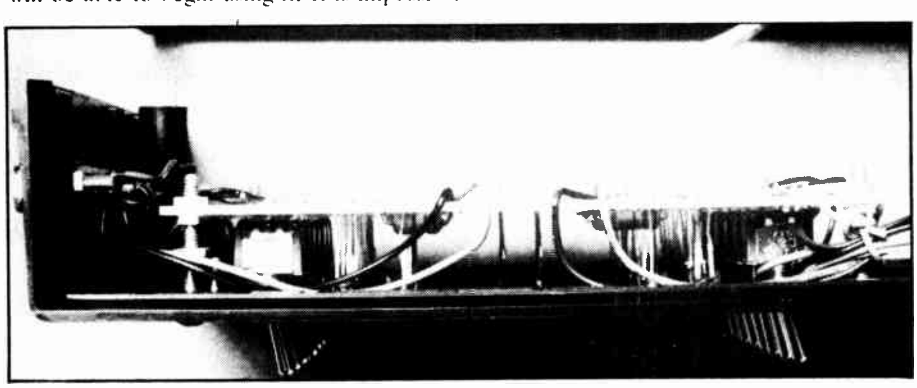
Activating DTE:

BOB can also be used to activate a Data Terminal Equipment device (DTE), such as a terminal, to see if it is operating correctly.

With CTS, DSR and CD switched to

RS232-C Connections:

PIN 1	Protective Ground
PIN 2	Transmit DATA (TD)
PIN 3	Received DATA (RD)
PIN 4	Request to send (RTS)
PIN 5	Clear to send (CTS)
PIN 6	DATA set Ready (DSR)
PIN 7	Signal Ground
PIN 8	Received line Signal Detector (CD)
PIN 20	DATA Terminal Ready (DTR)



+12 V, data sent by the DTE should appear as a flickering of the LED on line 2 (TD). If the data does appear, there are still some things you should use BOB to determine: e.g: does the DTE set line 20 (DTR) to +12 V? (many DCE devices must have this line set to +12 V); does the DTE under test, set line 4 (RTS) to +12 V before sending data?; how does the DTE react to changes on lines 5 (CTS), 6 (DSR) and 8 (CD)?

These questions can be answered quickly by changing switches and observing the LED's. The answers will come in handy when you wish to interface the DTE to another piece of equipment.

Connecting like devices:

It is sometimes necessary to connect two RS-232-C devices with the same configuration (DCE or DTE). If two terminals (DTE) are simply wired together, they would both attempt to transmit on line 2, and receive on line 3.

This means that the transmitted signals would collide, and no received signals would be heard. Both units would also be setting line 4 to +12 V, but lines 5, 6, and 8 would be set by neither. This would not work because there is no difference in potential.

By using BOB, two DTE devices can be connected without any problems. In this configuration, BOB is called a "null modem" since it replaces the modem's that would usually connect two terminals. Two DCE devices can be connected in a similar manner.

ETI 181 PARTS LIST

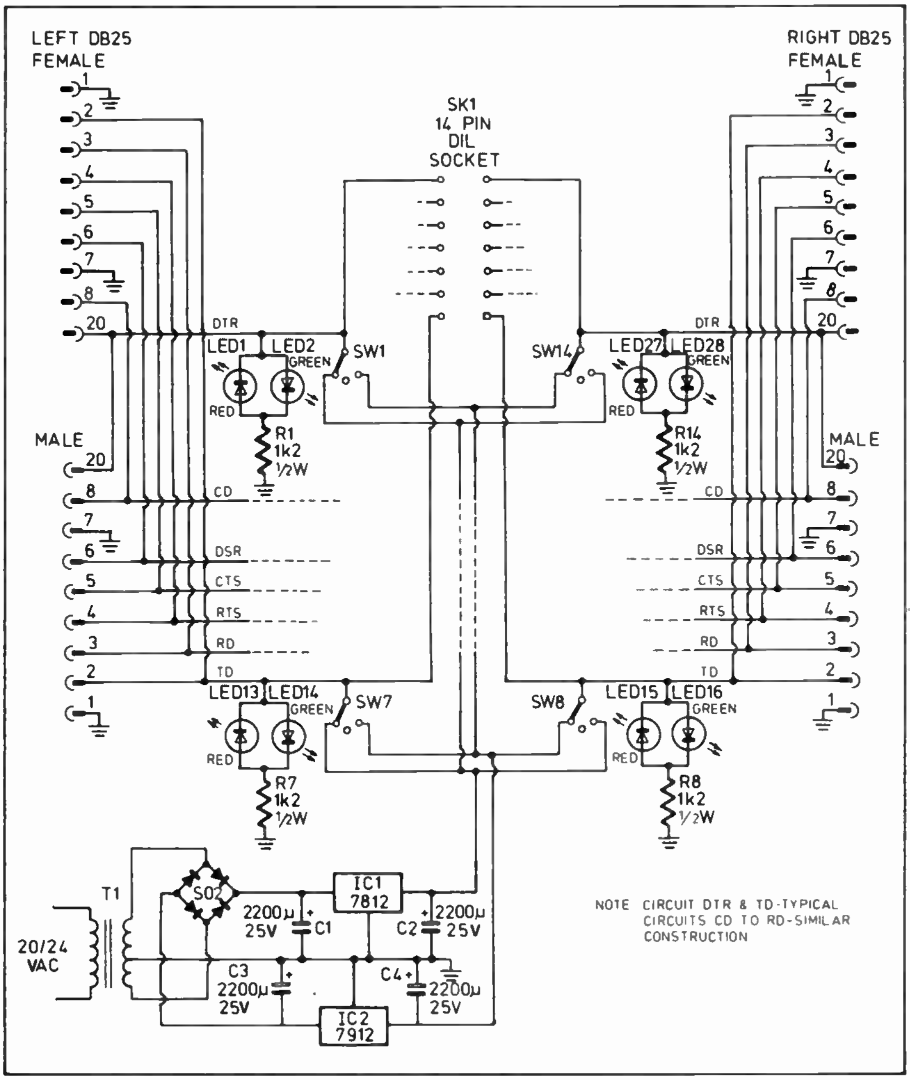
1 only Printed Circuit Board, 86-9-2
14 only SPDT Centre-off Toggle Switches
14 only Red 5 mm LED's
14 only Green 5 mm LED's
28 only LED clips
14 only 1K2 ½ watt Resistors
1 only 14 pin Wire-wrap I/C Socket
1 only Silk-screened, Aluminium, Front Panel
1 only Large Plastic Case
4 only ½" Plastic Spacers
4 only 1 x ⅛" Bolts
1 metre Tinned Copper Wire
2 only DB25 Male Connectors
2 only DB25 Female Connectors
14 only ½" x ⅛" Bolts
18 only ⅛" nuts

POWER SUPPLY

1 only Printed Circuit Board 86-9-3
1 only PL24/20 VA Transformer
4 only 2200uf 25 V RT Electrolytic Capacitors
1 only SO2 Bridge Rectifier
1 only 7812 TO-220 Voltage Regulator
1 only 7912 TO-220 Voltage Regulator
4 only ½" Spacers
4 only 1" x ⅛" Bolts
4 only ⅛" Nuts

GENERAL

1.5 metres Mains Cord
1 only 3 Pin Mains Plug
1 only Strain Relief Bush
10 metres Light duty Hook-up Wire
1 only DPDT Toggle Switch
19 only Printed Circuit Board Pins



For DTE devices set all lines in centre position, set CD line on both sides to +12 V, bridge TD to RD opposite side on both sides, bridge RTS to CTS opposite side on both sides, bridge DSR to DTR opposite side on both sides.

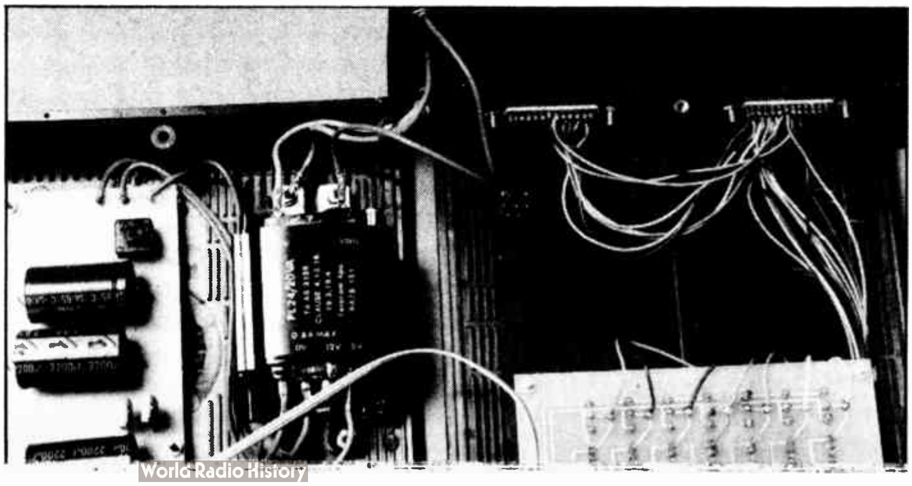
You may, if you wish, have this configuration permanently on a 14 pin header for future use.

These are only a few functions of BOB. BOB is a powerful unit for monitoring and changing signals at an RS-232-C interface. If care is exercised to avoid short-circuits, BOB can teach much about RS-232-C and help solve confusing problems.

Further reading:

"Beating the R232 Blues" G. Wideman, ETI, Aug. 1983, p84. ●

Kits of this project are available from All Electronic Components, 118-122 Lonsdale St, Melbourne Vic 3000. (03) 662 3506. Cost is $268.



PRECISION CRO CALIBRATOR

Anyone who has used a CRO will know what an incredibly useful instrument it can be — providing you can be sure of its calibrations. This handy low-cost calibrator will let you check and adjust all main settings easily and efficiently. Designed by the Electrical Engineering staff at Sydney University, it is easily built and can also be used to check out a CRO you're considering buying.

WHEN OSCILLOSCOPES or 'CROs' first started to be used in electronics, they were simply used to 'look at' what was going on in a circuit. Just being able to 'see' what was happening to the voltages and currents was a big step forward from what had been possible before, and for a while that was enough.

But nowadays people expect to be able to use a CRO in a much more quantitative fashion, to measure instantaneous voltages, voltage changes and timing details. In order to be used in this more serious way, a CRO naturally needs to be properly calibrated.

The small low-cost calibrator described here has been designed to make calibration of almost any reasonably modern CRO a fairly simple and straightforward job. It produces pulses of known accurate amplitude and timing, whose peak-to-peak voltage and repetition rate can be varied over a wide range to allow most of a CRO's normal ranges to be checked.

While the calibrator can be used to check and adjust both gain and linearity for both the vertical and horizontal channels of an instrument, in practice most CROs do not provide a means of adjusting the linearity of the vertical amplifier(s) — only the gain. Similarly although you can use the calibrator to check horizontal linearity, most instruments only allow you to adjust the horizontal gain and average timebase sweep speed.

A CRO which was made more than ten years ago (fairly rare in the commercial world, but still common in hobby workplaces) can easily drift a few percent in a period of months. Hence the need for a handy reference like this calibrator.

In addition to its main intended use in keeping your existing CRO or CROs calibrated, it is probably also worth taking along when you are looking at new CROs with an eye to purchase. It has surprised the authors to see how badly some otherwise impressive units fared, even brand new.

We will now discuss how the calibrator is used, for those who may not have been exposed to such procedures before. In addition to checking vertical gain accuracy and sweep speed, we will also explain how to check overshoot, timebase non-linearity, the presence of a satisfactory vertical delay line for viewing non-repetitive triggering events, and vertical non-linearity.

Note that we must assume you have access to the operating manual of your CRO, for details regarding the manufacturer's recommended procedures in adjusting the instrument. This varies from instrument to instrument, and it is often important to adjust things like the vertical attenuator and timebase ranges in a particular order.

Vertical calibration

This is initially fairly simple. Referring to the first CRO screen photograph (represented in Figure 1), the instrument's appropriate present potentiometer is adjusted to obtain precisely the correct deflection. The photograph was taken on a brand new CRO on a medium voltage deflection scale, and so the trace is narrow and sharp. The adjustment should be done on the range(s) specified by the manufacturer, but the checking of calibration should be done on several of the available ranges.

Figure 2 shows the same (brand new) CRO on its most sensitive range. There are two points to note. The first is that the trace was fuzzy, primarily on account of CRO amplifier noise. This requires that the operator effect some estimate of the difference between trace centres. It is best to take one edge (in Figure 2 the lower) and adjust the vertical position so as to set the reference edge precisely on a graticule line. Then the same edge of the trace on the other part of the squarewave cycle gives the deflection.

In Figure 2 the deflection is clearly about 5.2 divisions. The second point to note is that the CRO (correct to the limit of measurement on the 50 mV/div range) is 4 per cent high on the 1 mV/div range. Thus the need to check performance against specifications on *all* ranges is very clearly demonstrated.

Horizontal calibration

Initial sweep calibration should similarly be carried out at the sweep speed specified for the particular CRO, but a check is recommended over a wider range. Figure 3 is of a set of 10 timing pulses arrayed over a CRO screen. With the fine vertical edge of the first pulse set on the first vertical graticule line, the same part of the last pulse should align with the final graticule vertical. Even on the screen of the high quality and fast writing speed CRO used for this the intensity must be set quite high to clearly show the fast vertical edges.

Timebase Linearity

Regrettably for readers, but fortunately for ETI, we were unable to locate a CRO with bad linearity. Referring still to Figure 3, had there been a non-linearity in the sweep, it would have been evident as one or more vertical pulses not being aligned with their respective graticule lines after the horizontal calibration had set the end pulses on their respective markers. We have observed such problems even on new CROs. The calibrator prevented a poor purchase in that case!

Overshoot

This phenomenon is becoming rarer on CROs these days. The only CRO we could locate with significant overshoot proved to have neither graticule illumination for clear illustrations nor the ability to give a good look for single edge blow up.

We settled for using a 10 year old CRO — also with no graticule illumination. Figure 4 shows overshoot on the edges of the inverted timing waveforms, rather than the voltage waveforms. These were easier to trigger on than the relatively infrequent edges of the voltage calibration squarewave. The overshoot is the 'wiggle' seen at the beginning of the clear horizontal parts of the trace.

Delay Line

Figure 5 shows the rising edge of the voltage calibration waveform on a high quality oscilloscope. The trace was rather dim, because the sweep occurs only at 1 kHz, despite the sweep speed of 10 ns/div. The same edge that triggered the timebase can be seen, because the CRO contains a delay line, so that the signal is delayed for long enough (some tens of nanoseconds usually) to allow the sweep to commence before the signal is applied to the vertical deflection plates. A view such as this is impossible otherwise.

The edge is not perfect. The salient feature is a minor step-like appearance at the top of the edge. This is in fact not CRO

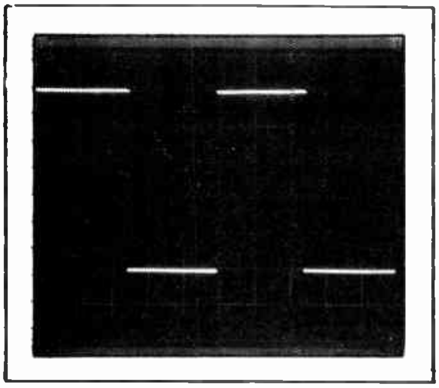


Figure 1. Vertical calibration. New Trio CS-1022 set to 0.2 ms/div, 50 mV/div.

induced overshoot, but the result of mismatch in the connection from the calibrator via a 50 ohms cable to the 1 m/50 pF input impedance of the CRO. Such is the typical situation which will be encountered.

Construction & setting up

The construction is basically a case of duplicating the prototype shown here. Initially drill the appropriate holes in the panel of the case being used. All the electronics is attached to the panel along with the switches and connectors, so the shape of the rear of the container is not important.

Once drilled, the various holes on the front label can be lined up with the holes in the panel and the label attached. After adhering the label, fit the components and their knobs where applicable. Next assemble the pc board, exercising the usual care to ensure that all the polarised components are orientated correctly. Also take care to leave the solder joints clean and free of blobs which might short tracks together. Remember that only resistors and capacitors without polarity markings can be put in either way around, because everything else is polarised in some way!

Next attach all the components which are mounted to the rear of the panel controls. These components are fixed to the panel controls to reduce the number of flying leads. Be careful to get all of them in their correct circuit locations, as reversal of components here can lead to 'satisfactory' but uncalibrated operation.

Finally connect the pc board to the panel with short lengths of hookup wire. The leads should be as short as is practicable while leaving the board accessible for alignment, servicing, etc. Long leads can give rise to unwanted coupling between the circuit sections, and may also permit the radiation of unwanted EMI (Electro-Magnetic Interference), even up to VHF frequencies, because of the sharp edges generated by the driver circuits.

With a battery connected the unit should now function. It remains only to adjust the supply to its correct voltage. This of course

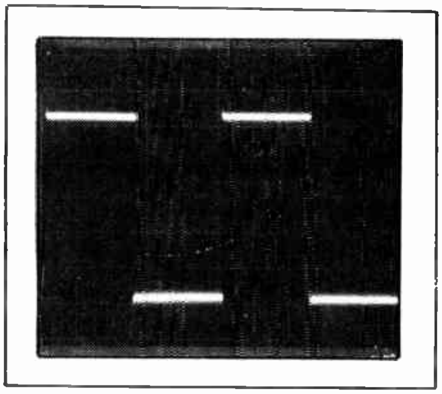


Figure 2. The same CRO as in Figure 1, at 1 mV/ div.

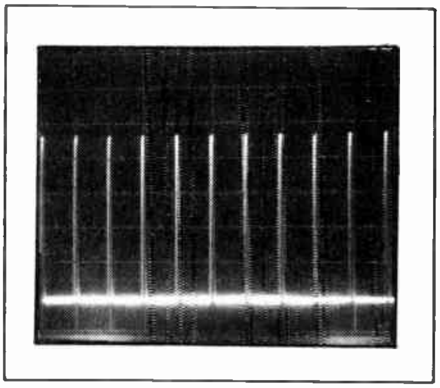


Figure 3. A seven year old Tektronix 465 is shown at 1 ms/div. Its last calibration date was not recorded, if it happened at all!

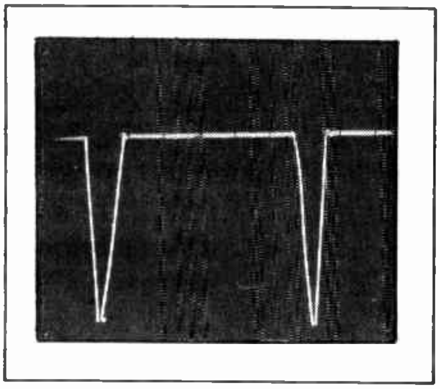


Figure 4. A BWD 511 of unguessable age. CRO calibrator set to 1 µs timing pulses, CRO at 1 V/div and 0.2 µs/div. No graticule illumination available.

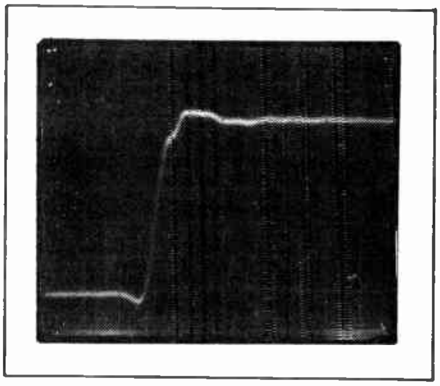


Figure 5. Tek 465 again, 20 mV/div and 10 ns/div, with the calibrator on 100 mV range.

The CRO calibrator circuit divides into six parts. These are a power supply section, a 1 MHz clock, a 10 MHz clock, a divider chain, a voltage output section and a timing pulse output section. Each will be discussed in its turn.

Power supply

The power supply consists of ICs 1, 2 and part of IC3, with their surrounding components. Q1 is used as a saturating pass element, to connect and disconnect the battery from those parts of the device which are not continuously powered. IC1 is the only IC connected all the time to the supply. Initially consider that C1 is discharged. IC1(b) and (c) have their inputs held low, so that their outputs are high. These gates are connected in parallel to increase the current delivering capacity. Because the outputs are high, Q1 is held off.

In this condition the current drain from the battery is so small that it will last its shelf life, and it may be considered unconnected.

When PB1 is depressed to turn on the instrument, C1 is charged up rapidly via R1. The inputs of IC1(b) and (c) and taken high and the outputs are thus driven low, turning on Q1. This applies almost the full battery potential to IC2, a precision supply regulator IC, via the external supply jack. Capacitors C3 and C4 decouple the rail to prevent instability problems arising from the source impedance of the battery. RV1 is used to trim the regulated supply voltage to 5.05 volts, while C5, C6 and C7 provide further decoupling and compensation.

C1 will slowly discharge via R3. When it reaches about half rail potential, IC1(b) and (c) will once again turn Q1 off, returning the instrument to the quiescent (off) condition. Thus automatic turnoff is effected. Should PB2 be fitted, it may be used to manually turn off the supply.

If the unregulated supply falls below a level where good regulation of the 5.05 volt supply by IC2 is not possible (when the battery goes flat, for example), IC3(a) and IC1(a) and (d) are wired to immediately turn the supply off. Note that IC3 is powered from the 5.05 volt rail. When the input of IC3(a) falls to below about half its supply rail, which is about 5.05 volts, it charges C2 via R5. This causes IC1(a) and (d) to discharge C1 via D1 and R2, which shuts off the supply.

Resistors R6 and R7 set the potential on the collector of Q1 which is required to initiate this automatic shutdown. This mechanism prevents any attempt to use the device when it is potentially operating incorrectly due to a low battery condition.

1 MHz Clock

The 1 MHz clock is crystal controlled and provides the timing reference for all (but the 10 MHz) timing signals and the voltage calibration squarewave. The oscillator consists of IC4(b) and surrounding components. It runs off the 5.05 volt supply, and operates continuously when the unit is in the on state.

Divider Chain

The divider chain consists of six decade counters. These provide signals divided by 10, 100, etc, up to 1 million, giving 10 per cent duty-cycle pulses of from 100 kHz to 1 Hz, all derived from the 1 MHz reference. The 'on' indicator is driven from one of these dividers, giving fast flashes with low duty cycle to give indication without wasting power.

Timing Pulse Output

IC3(b/d) and IC11 with surrounding components form the timing pulse output section. This section, when enabled, shapes the timing pulse train and delivers it to the output with 50 ohm impedance. With SW1(a) in the 'time' position, IC3(d) is enabled. The disabling system prevents the gates consuming power when not required. SW2(a) selects the timing train with the required frequency.

SW2(b) selects the capacitor which defines the shape of the output pulse. This mechanism is incorporated to provide pulses whose edges are conveniently visible on all frequency ranges.

On the 1μs range, the pulses are not modified by a capacitive load on IC11(c/d/e), but are shaped by the driving network formed by the sections of IC4 not used in the oscillator. Not only does this provide neat, visible pulses at this frequency, but it makes use of the gates in the package already handling 1 MHz signals. This reduces the chances of crosstalk between these signals, with components in very high frequency ranges, and the slower timing signals. Such crosstalk can upset triggering and interfere with the waveform being displayed.

10 MHz Clock

The 10 MHz clock circuit is built around IC12. It differs somewhat from the 1 MHz clock in order to allow clean operation at this high frequency. R39 and C18 with IC12(c/d) form the output wave of the oscillator into a pulse string for easy alignment against graticule timing marks. Note that the circuit is separately decoupled by C19, and is only powered by completing the earth return when required, as selected by the range switch SW2(b).

This circuit consumes considerable power in relation to the other parts of the calibrator, and so is not activated when not required. Its output is fed to a different connector. If it is not required, this whole section can be deleted, removing the 10 MHz 100 ns range with commensurate cost saving. If no fast CROs are likely to be encountered, this would not be a problem.

Output Section

The voltage output section, when enabled by SW1(a), produces calibrated output squarewaves at 1 kHz. The voltage level is selected by SW3. When not enabled Q2 remains off. When enabled it is switched into and out of saturation by IC11(b). Capacitor C16 is selected to precisely pull Q2 out of saturation as quickly as possible.

When fully saturated, Q2 has approximately 0.05 volts drop across it — hence the 5.05 volt supply. R16 through R36 provide a precise 1-2-5 sequence attenuator using only E12 series preferred resistor values. Signals of 1 mV p-p to 5 V p-p are developed. The output impedance is at most a few hundred ohms. Because no buffer is provided, the CRO input must not be terminated in 50 ohms.

WHILE WE'RE TALKING ABOUT CROs . . .

While we're thinking about CROs and their performance, let's consider for a moment a few general aspects of oscilloscope or 'CRO' performance which are often taken for granted.

Nowadays there are quite a few CROs available for around the same price as a humble (humble?) audio amplifier. But we expect rather different things from the two types of equipment. For example we only expect amplifiers to have a bandwidth of from 20 Hz to 20 kHz — yet for the same money we expect a CRO to have a bandwidth of dc to 20 MHz! This is a very big difference, yet some of the better CROs go a great deal further, to 200 MHz or even 500 MHz.

Aatute readers will guess at once that something is traded for this wide range. But perhaps few of you may have realised just what it is that is missing from the CRO's performance.

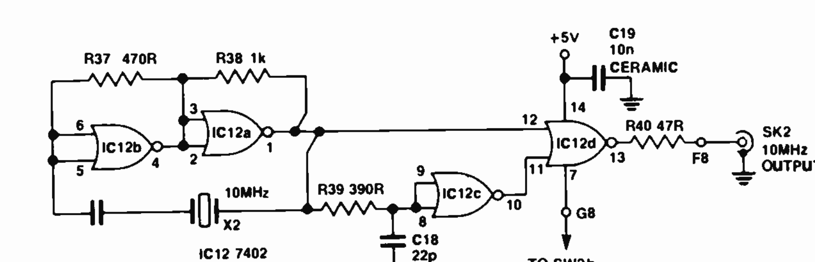
One of the things usually noted in the 'fine print' of CRO specifications is linearity. A vertical deflection linearity error of one or two per cent is typical, probably specified for a deflection of say six or seven major divisions.

In an audio amplifier this would be called distortion, and if it rose much above say .01%, today's audio enthusiast would be most unhappy. By paying more money, you can get an amplifier with ten times lower distortion again. In addition, the audio amp is expected to maintain this low level of distortion even at power output levels of 100 watts or so — a far cry from the few tens of volts, at low currents, expected from the CRO deflection amps.

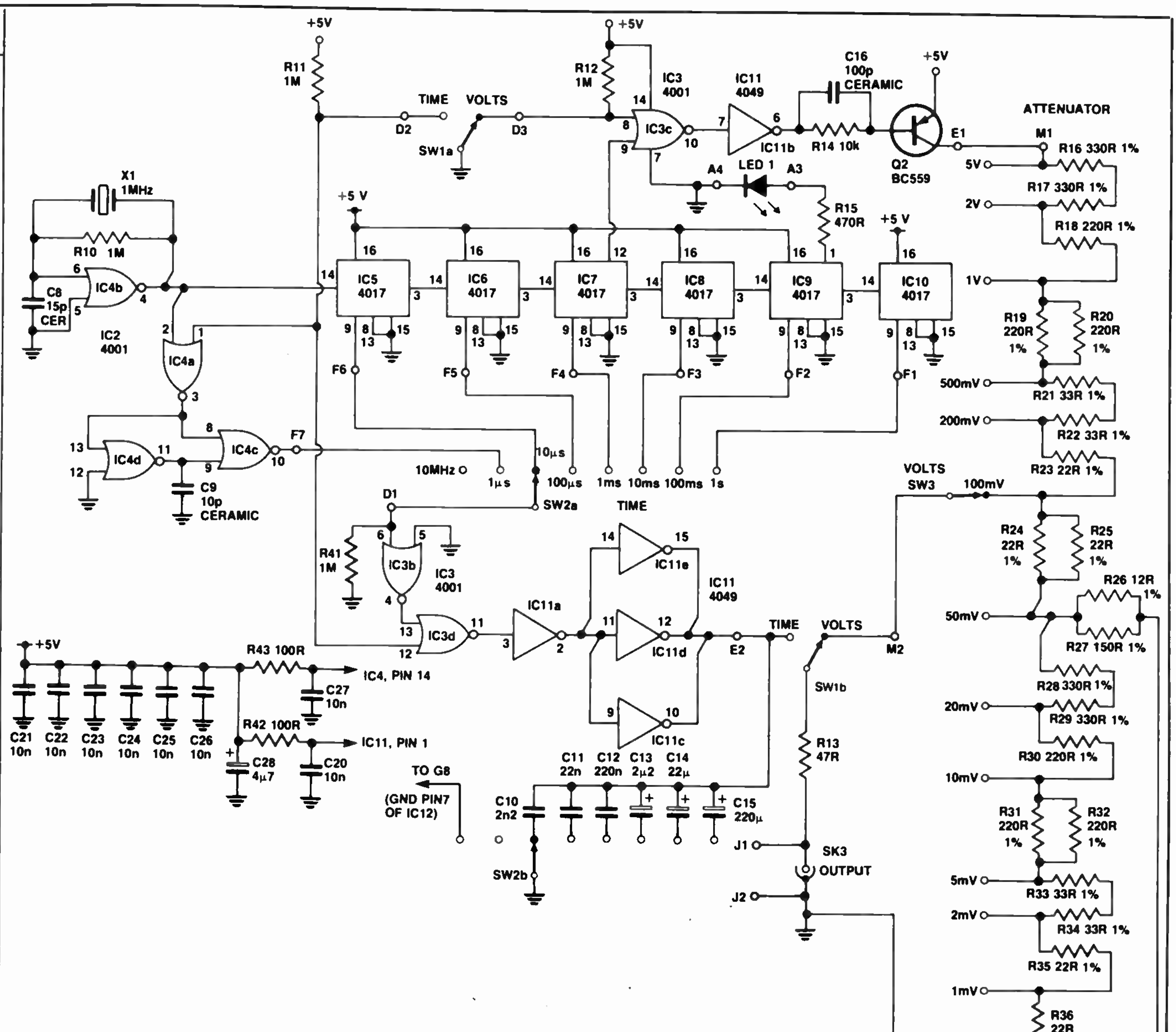
The CRO can of course get away with this relatively poor linearity because the human eye can't perceive less than about one per cent distortion, whereas our ears most certainly can — particularly on the right kind of programme material. Round two to the audio amplifier.

In most applications where a CRO is used, the user expects to be able to resolve the position of the trace to within 0.1 divisions, on a screen with eight vertical and 10 horizontal divisions. This represents a dynamic range of about 38 dB in the vertical direction. Compare this with the 50 dB or more you get from your cassette player, or the 96 dB from a compact disc.

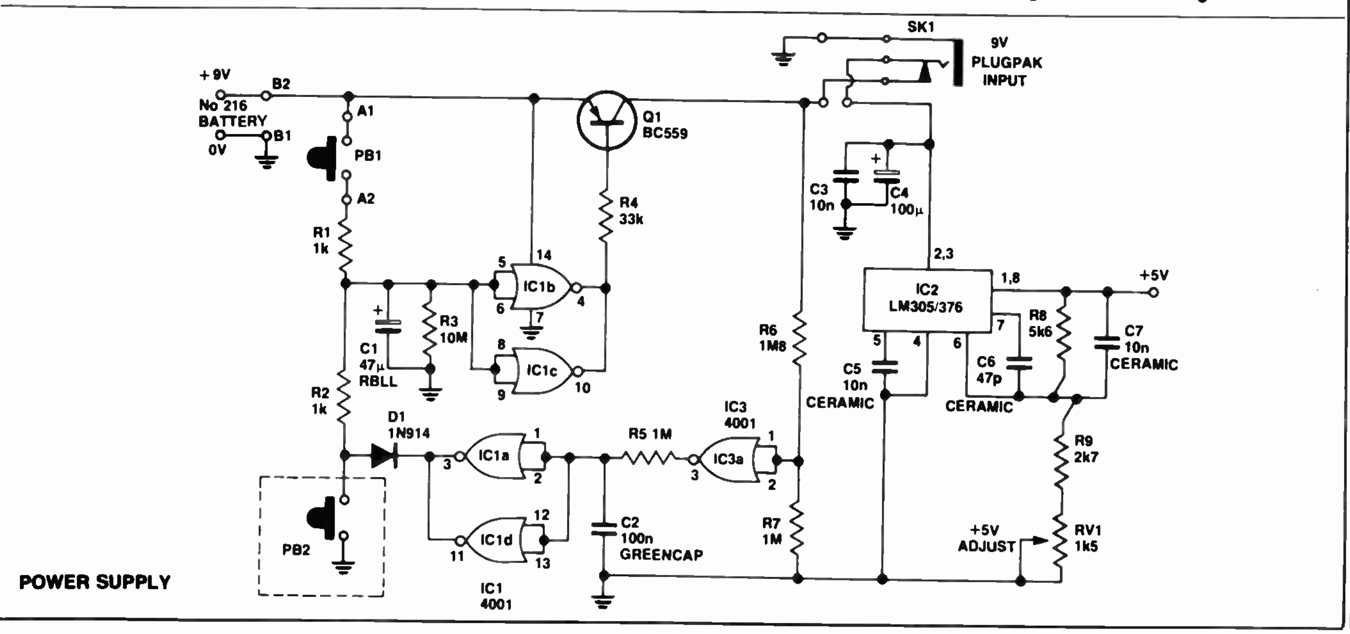
The upshot of this is that the signal-to-noise ratio of a CRO can be relatively poor, and the spot position uncertainty fairly large. Round three to the audio amplifier.



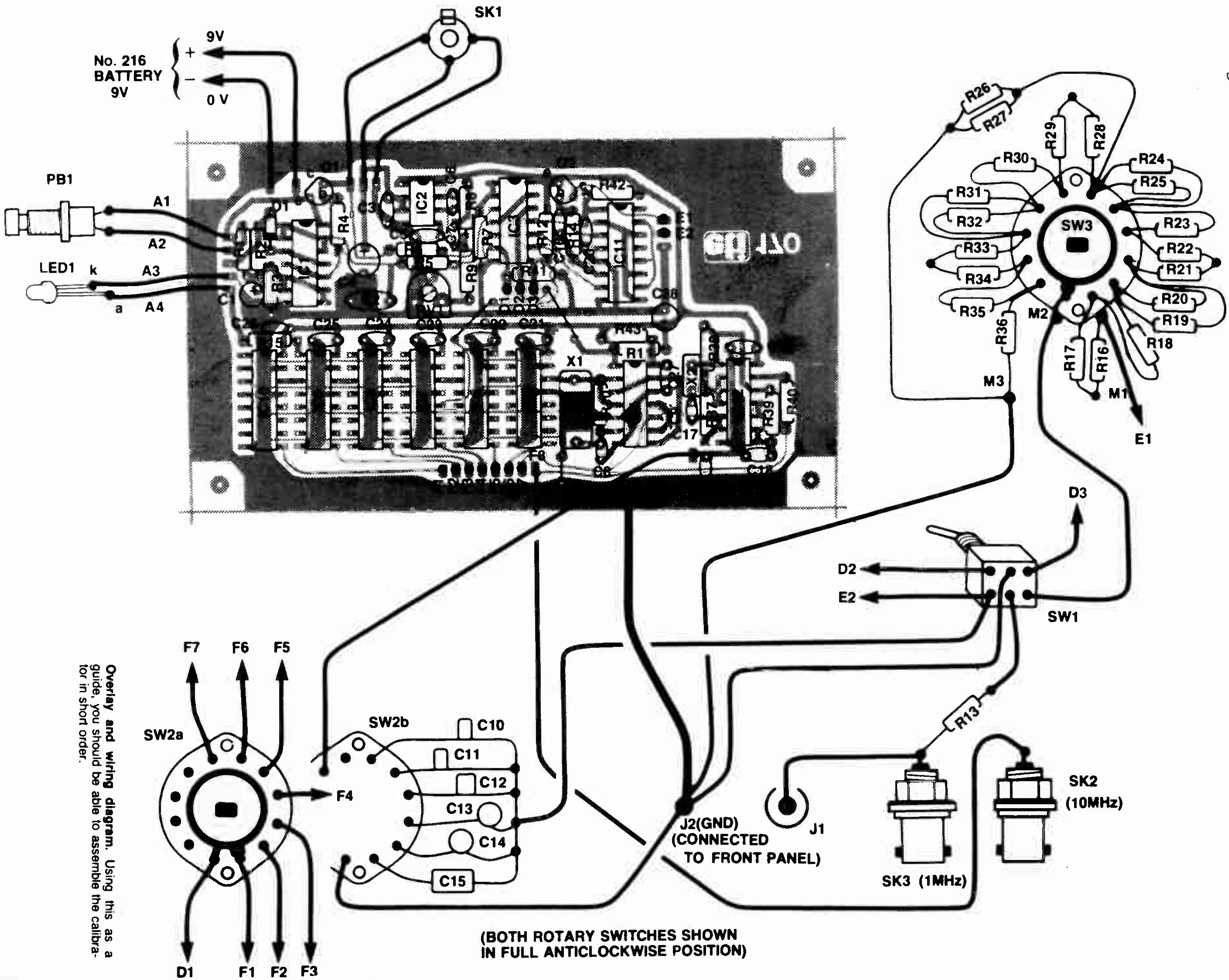
10 MHz timer.



MAIN CIRCUIT



POWER SUPPLY



Overlay and wiring diagram. Using this as a guide, you should be able to assemble the calibrator in short order.

PARTS LIST — ETI-170 CRO CALIBRATOR

Resistors.................all ¼W, 2% unless noted
R1, R2, R38..........1k
R3..........................10M
R4..........................33k
R5, 7, 10, 11, 12, 41..........................1M
R6..........................1M8
R8..........................5k6
R9..........................2k7
R13, R40...............47R
R14........................10k
R15, R37...............470R
R16, 17, 28, 29......330R 1%
R18, 19, 20, 30, 31, 32..........................220R 1%
R21, 22, 33, 34......33R 1%
R23, 24, 25, 35, 36..........................22R 1%
R26........................12R 1%
R27........................150R 1%
R37........................470R
R39........................390R
R42, R43...............100R

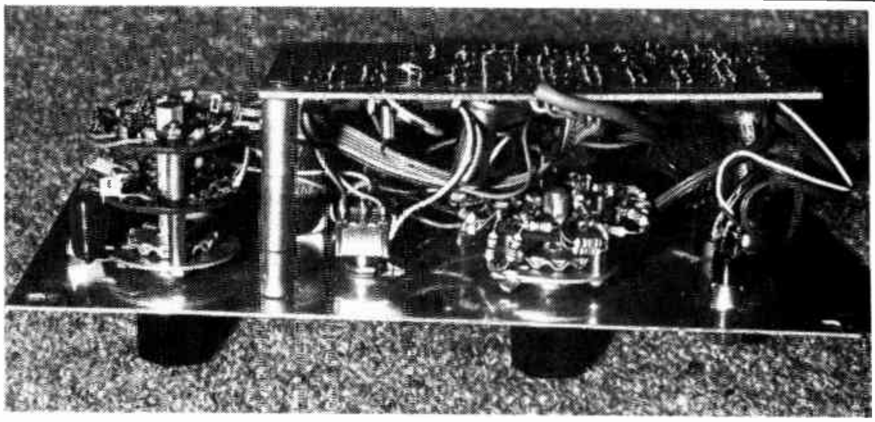
Capacitors
C1..........................47µ/16VW RB electro.
C2..........................100n/100V met. poly.
C3, 5, 7, 19-27, 29...............10n ceramic monolithic
C4..........................100µ/25VW RB electro.
C6..........................47p ceramic
C8..........................15p NPO ceramic
C9..........................10p NPO ceramic
C10........................2n2 ceramic monolithic
C11........................22n ceramic monolithic
C12........................220n met. poly.
C13........................2µ2/35VW solid tant.
C14........................22µ/6VW solid tant.
C15........................220µ/16VW electro.
C16........................100p ceramic
C17, C18...............22p NPO ceramic
C28........................4µ7/35VW solid tant.

Semiconductors
D1..........................1N914 diode or similar
LED1.....................5 mm red LED with bezel
IC1, 3, 4.................4001 CMOS quad NOR
IC2..........................LM305/376 regulator
IC5, 6, 7, 8, 9, 10 ...4017 CMOS decade divider
IC11........................4049 CMOS hex inverter
IC12........................7402 TTL quad NOR
Q1, Q2...................BC559 or similar

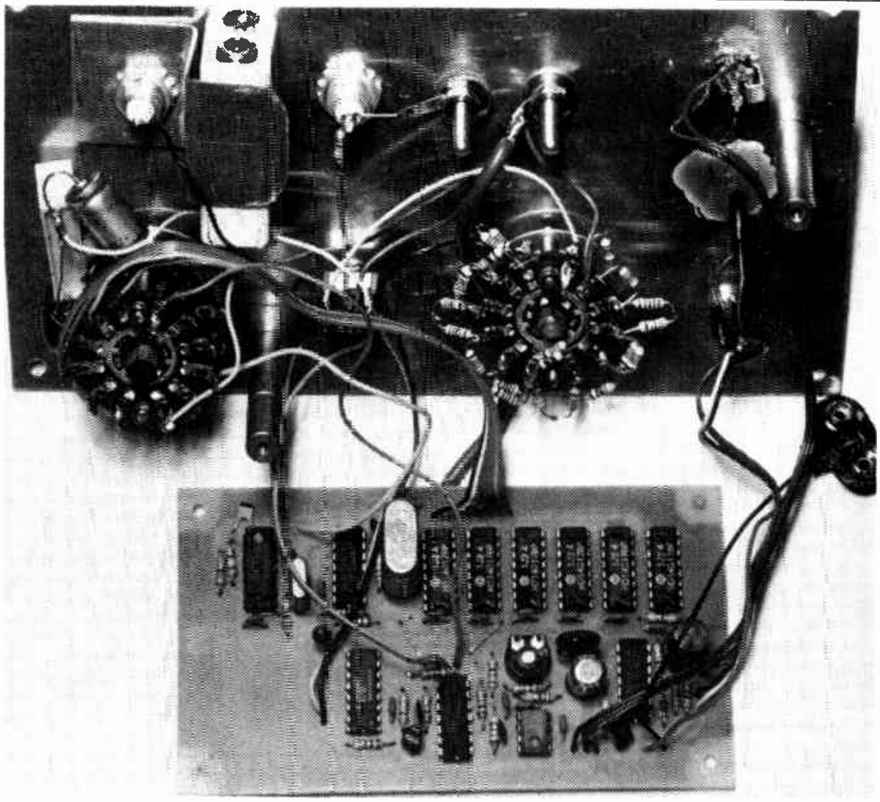
Miscellaneous
SW1 ....DPDT sub-min. toggle
SW2 ....2-pole 8-pos. rotary
SW3 ....1-pole 12-pos. rotary
PB1 ....Mom. contact sub-min. pushbutton
SK1 ....3.5 mm sub-min jack socket with sw.
SK2, SK3...............Panel-type BNC socket
J1, J2.....................Tip jacks
X1..........................1 MHz crystal, par. res. with 30p
X2..........................10 MHz crystal, par. res. with 30p

ETI-270 pc board and Scotchcal label; 196 x 112 x 60 mm zippy box; 2 small knobs; 216-type 9 V battery; battery clip lead; 2 x 45 mm tapped spacers for pcb mounting; hookup wire, nuts, bolts, etc.

Estimated price: $60



Mounting the pc board to the panel. The leads should be as short as possible to avoid coupling between circuit sections and to prevent unwanted radiation.



Inside the case. Apart from the ICs and other components mounted on the PCB there are few other components required and these are mounted directly on the switches. Note the battery clamp bracket, attached to the front panel by the 10 MHz output socket.

requires a known accurate reference. We used a 3½-digit multimeter, which is adequate. It should be attached to the regulated supply rail, and RV1 adjusted to give the specified 5.05 volts. This gives 1% accuracy to the unit. As most CROs are good only to 2% or so, this is reasonable.

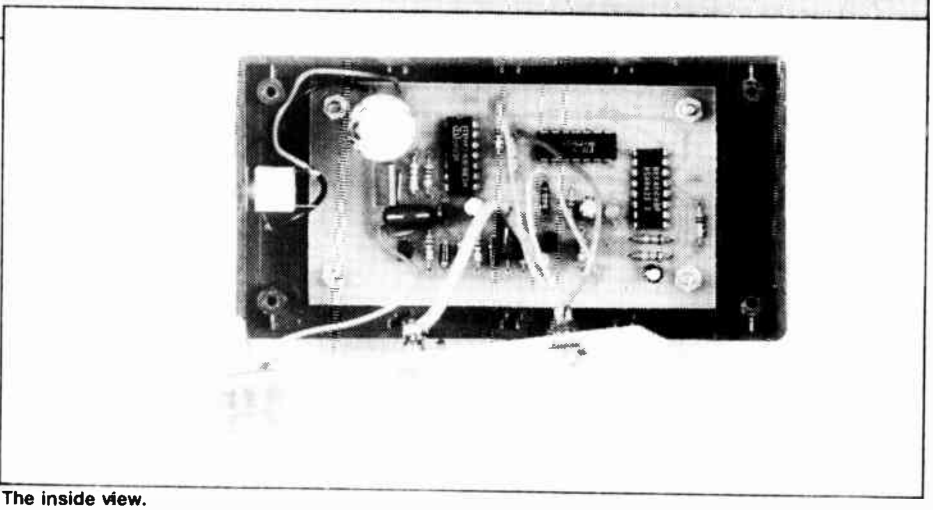
Further accuracy can be obtained if required by using the following alignment procedure. Switch the circuit to volts. Attach a 4½-digit meter to the volts output. Short IC3 pin 10 to rail momentarily, and set RV1 for 5 volts on the 5 V range. This gives accuracy limited only by the attenuator chain resistor tolerance.

Artwork: Front panel artwork and drilling diagram are available on request to ETI, PO Box 227, Waterloo, NSW 2017.

FREQUENCY STANDARD

Neale Hancock

Get the equivalent of a rubidium frequency standard by draping a piece of wire over the back of your TV set!!!



The inside view.

NO, THE ABOVE statement is not an ETI attempt at humour nor is it a figment of our imaginations. Believe it or not your humble television can provide an extremely stable and accurate reference frequency. You may ask "Oi, what's with the wire then?" (or words to that effect). Well that acts as a transducer to pick up electromagnetic radiation from the back of the set.

Normally you would need to spend thousands of dollars to achieve accuracy beyond the parts per hundred thousand you expect from ordinary meters. With this simple project, an extremely accurate 1MHz signal can be derived.

The horizontal deflection in the picture tube is achieved by applying voltage to the yoke at a frequency of 15.625kHz. These pulses are decoded by the receiver from the signal sent by the television station. The high voltage is provided by the extra high tension (high voltage) section of the TV set. Therefore, the eht section has a very strong electromagnetic field of 15.625kHz pulses around it.

The 15.625kHz pulses generated at the television station are derived from a rubidium standard (or will regularly check against one) thus they are very stable and very accurate. They provide the reference signal for our frequency standard.

Circuit overview

The electromagnetic radiation emitted by the eht section is transformed into electrical pulses by the loop of wire. These pulses are then conditioned and buffered providing a useful reference signal. When they are of an adequate level, the pulses go into a CMOS 4046 phase-locked loop (PLL). The internal oscillator of the PLL is set to run at 1MHz and is locked to the incoming 15.625kHz reference signal.

When the PLL is locked to the reference signal, its internal voltage controlled oscillator (VCO) has the same stability as the reference. Therefore the outgoing 1MHz signal will be as stable as the 15.625kHz reference except for some PLL jitter. The jitter is due to the VCO of the PLL drifting slightly then re-locking onto the reference. This jitter can be minimised by careful selection of components.

Construction

Before you commence construction, examine the tracks on the pc board for breaks and bridges. First of all, put in the wire link located next to IC4 then put in all the capacitors and resistors making sure that the electrolytic capacitors are polarised correctly.

Mount the diodes and transistors, but check their orientation against the overlay first. Except for the voltage regulator, all the ICs are CMOS so take care not to touch their pins when putting them into the board. Also check the orientation of all the ICs against the overlay before you mount them.

The case can now be drilled out, and then sockets, LEDs and pc board mounted in it. Finally connect the sockets and LEDs to the pc board using short lengths of hookup wire, but check the polarity of the LEDs. To reduce the chance of having stray signals bouncing around the case, shielded cable should be used to connect pc board to the output socket. Also make sure that the input socket is insulated from the case to prevent a short circuit between it and ground.

THE BROADCAST TIME BASE

Because broadcast television studios produce a great many hours of video programmes that may need to be edited or mixed with other programmes at any time, it is vital that they have a very accurate time base or synchronising signal.

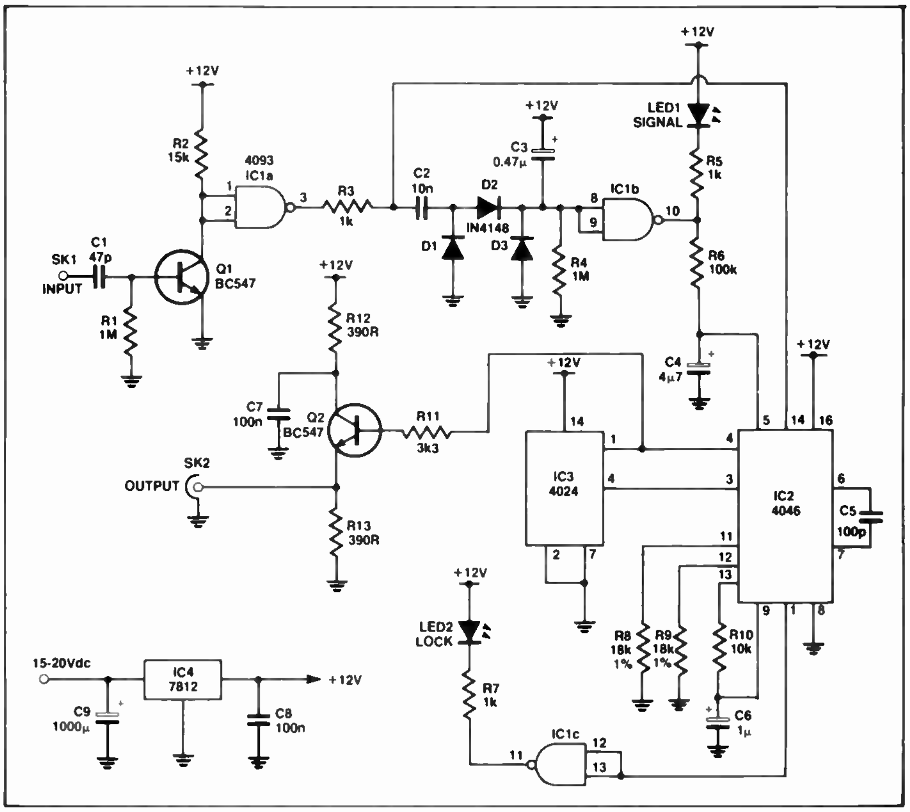
A few commercial stations and the ABC use a rubidium frequency standard with an accuracy of something like one part in one hundred billion. Most stations that don't have a rubidium standard will have an oven controlled crystal oscillator and will regularly check it against a station with a rubidium standard.

In Sydney the four VHF stations regularly compare their 4,433,618.75 hertz colour subcarrier against other stations and in virtually all instances all four are within one or two hertz of each other.

The only possible exception is that a bad video tape edit can sometimes cause a momentary phase error in the broadcast signal.

If you need the highest possible accuracy from this project then don't use it on something with a lot of video tape edits like advertising.

The system of using broadcast television signals as a frequency standard is so reliable it is used in the US by many registered laboratories. Each month the US National Bureau of Standards publishes a frequency offset for each of the networks.

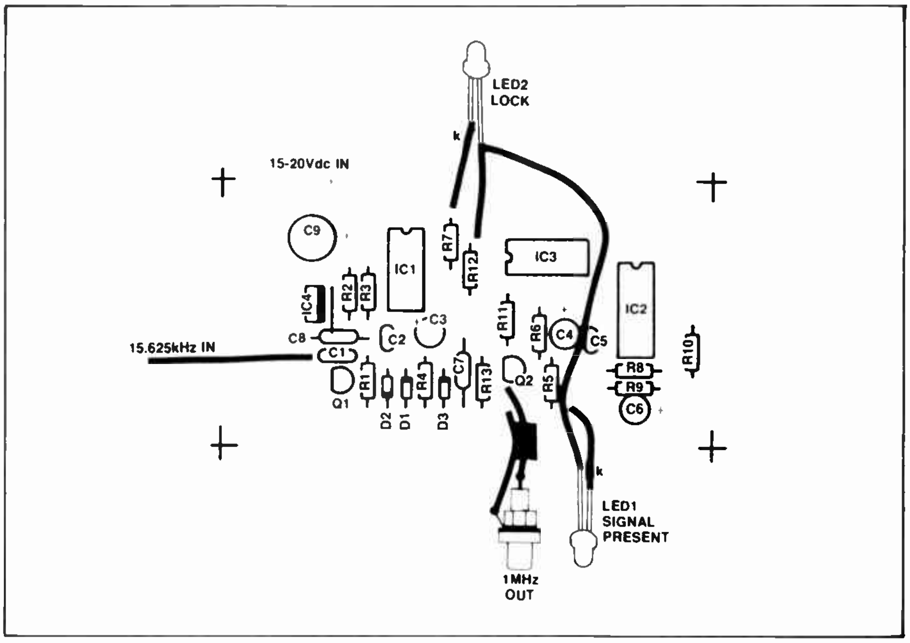


HOW IT WORKS — ETI-174

The incoming signal is filtered by C1, and R1 to remove any high frequency noise. The filtered signal then goes to the base of Q1. Providing this signal has enough level to turn Q1 on, the output from Q1 will be able to drive the inverting buffer IC1a. The signal is now at a high enough level to drive the input of the phase-locked loop, IC2. To show that this signal is present, LED1 is illuminated via IC1b.

The combination of R8, R9, and C5 sets the internal VCO of the phase-locked loop running at 1MHz. The ratio of R8 to R9 is optimised to minimise jitter. So that the internally generated 1MHz can be compared with the 15.625kHz reference, it is divided by 64. To do this, six stages of a 4024 CMOS counter are used.

To show that the VCO is locked to the reference, the lock detect output is used to illuminate LED2 via IC1c. The locked 1MHz signal goes to the output socket via the npn transistor, Q2. The voltage regulator IC4 and capacitors C8 and C9 supply the circuit with a stable 12 volts dc.



PARTS LIST — ETI-174

Resistors .... all 5%, ¼W except for R8 and R9 which are metal film ¼W

R1,4 .... 1M
R2 .... 15k
R3,5,7 .... 1k
R6 .... 100k
R8,9 .... 18k metal film
R10 .... 10k
R11 .... 3k3
R12,13 .... 390R

Capacitors

C1 .... 47p ceramic
C2 .... 10n greencap
C3 .... 0.47µ 25V electro
C4 .... 4µ7 25V electro
C5 .... 100p polystyrene
C6 .... 1µ 25V electro
C7,8 .... 100n greencap
C9 .... 1000µ 25V electro

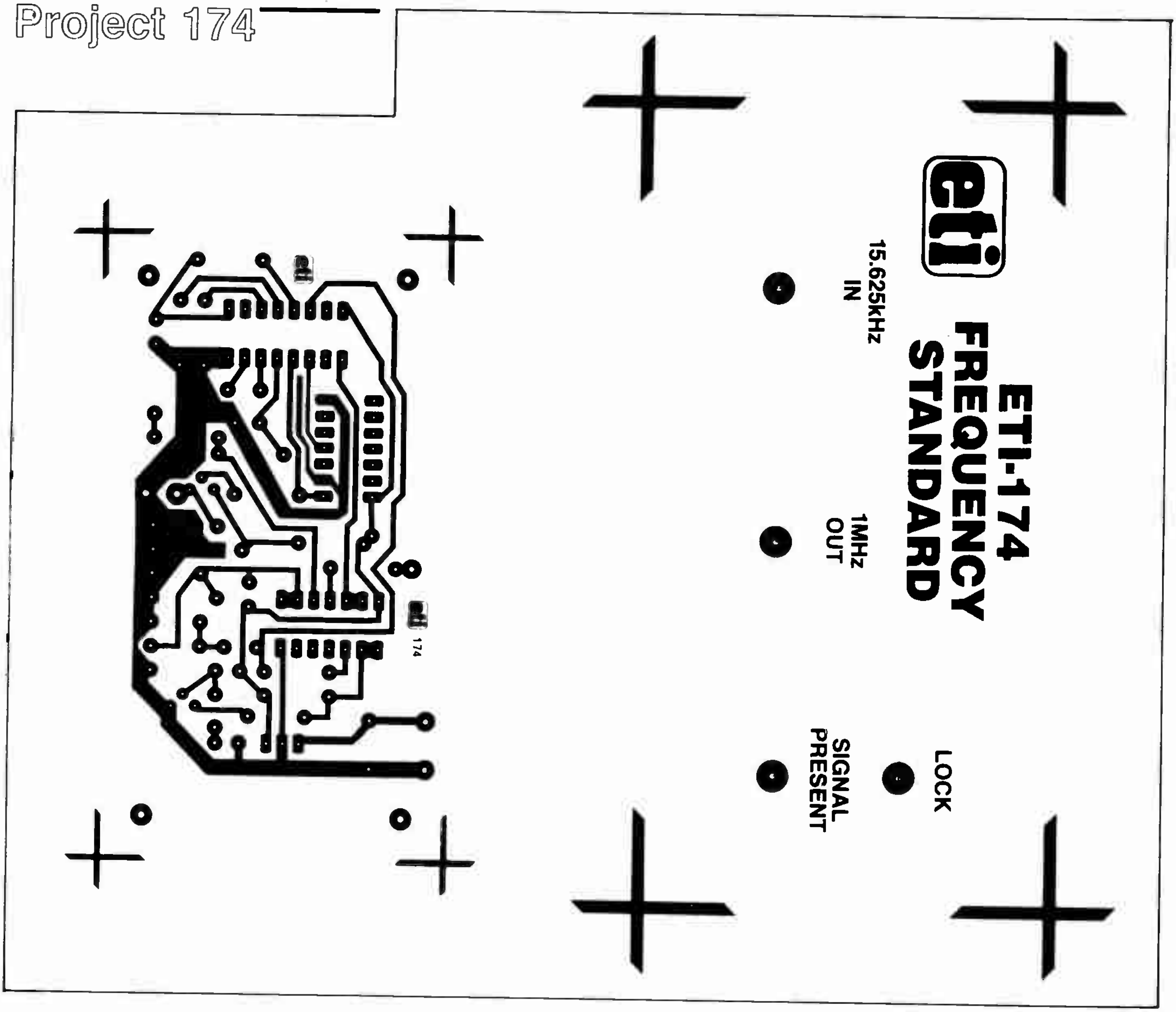
Semiconductors

D1,2,3 .... 1N4004
LED1,2 .... red 5mm
Q1,2 .... BC547
IC1 4093
IC2 4046
IC3 4024
IC4 7812

Miscellaneous

ETI-174 pc board; 41x68x130mm zippy box; female RCA socket; 4mm socket with binding post; shielded cable and hookup wire; Scotchcal or other front panel.

Price estimate: $21



Testing

After assembling the pc board, carefully examine it for solder bridges and misplaced components. Connect up a power supply to the frequency standard, this power supply can be between 15 and 20 volts and can be in the form of a battery, a plugpack or even a lab power supply. When the power is applied the output from the regulator should be around 12 volts. If this is not so look for short circuits around the supply rails.

If you want to perform an initial check on the PLL to see if it is operating correctly you can set it free running. This is done by shorting the VCO inhibit pin (pin 5 of IC2) to ground with a short (no pun intended) length of wire. The frequency at the output of the frequency standard should be between 700kHz and 1MHz (the one I tested oscillated at 850kHz).

Setting up and operation

When mounting the wire on the back of the TV set select the channel whose picture is clearest and most stable. This will provide the cleanest signal to put into the frequency standard. The vertical hold must be stable. If the picture 'rolls' then the reference frequency going into the frequency standard will be incorrect.

To pick up the strongest signal from the back of the TV, it is best to make three loops each about 12cm in diameter in a length of insulated wire. This should then be taped to the area at the back of the TV set which emits the strongest electromagnetic field. This area is near the eht section of the television and as I mentioned at the beginning of the article, this is where the field is strongest.

To find the place where the field is strongest you can use an ac reading analog multimeter with one end of the wire loop connected to the positive (or red) probe and the other end left free. As the wire loops are moved around the back of the set the needle on the meter will be at its maximum deflection where the field is strongest. If you do not possess an analog multimeter a digital multimeter will perform the same task as long as it has a 2Vac range. However, you will have to move the wire loops to a position then let the meter settle before you take the reading. This is more time consuming than using an analog multimeter, but it gives the same result.

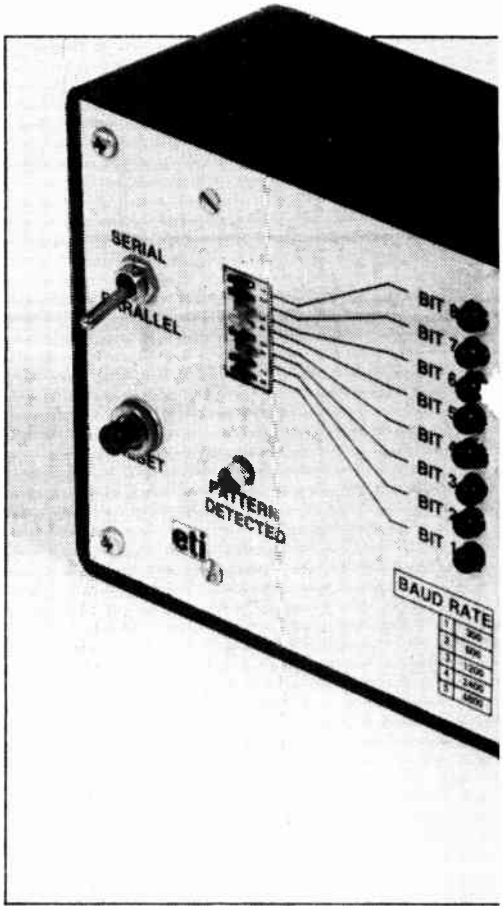
Once you have found the area where the field is strongest, and taped the loops there, connect the end of the wire to the frequency standard. The 'signal present' LED should illuminate followed shortly by the 'lock' LED. When the 'lock' LED comes on the frequency standard has stabilised and is ready to use.

If the 'signal present' LED does not illuminate then there is not sufficient field present. To remedy this, move the wire loops until the LED comes on. If this does not help add more loops to the wire or make the loops bigger. If the 'lock' LED does not come on, check that the PLL is operating correctly using the procedure mentioned in the "Testing" section. ●

Project 172

BIT PATTERN DETECTOR

Neale Hancock



Looking for a lost byte? Find it with the ETI-172! In applications where you are required to look for a particular byte of information in a serial or parallel data path, until now short of a logic analyser or a storage oscilloscope, there was not a lot to help you.

THIS PROJECT GIVES you a simple and economical way to detect and display specific bytes of data. It may be used on both parallel and serial data paths, or just on parallel data paths if you want to save yourself the cost of a UART.

Two methods used to indicate the presence of a specific data byte are possible, and shall be known affectionately as method 1 and method 2.

Method 1 is used to search a stream of data for a specific byte, then simultaneously illuminate an LED and send out a trigger pulse when the byte is detected. When this method is being used, the byte to be searched for is set on a bank of switches. Every byte which travels down the data path is compared with this byte and when the two are the same an LED illuminates.

Method 2 requires a single byte to be sent down the data path from a terminal or a bit pattern generator (the ETI-171 Arbitrary Waveform Generator can be used to perform this task). The bit pattern is set up on the switches and detected in the same way as in method 1; it is also displayed on the eight LEDs on the front panel.

When the bit pattern detector is used on serial data paths, either method 1 or 2 can be used to detect bytes of data. Method 1 is best used to determine whether a specific byte has reached the desired destination, and is convenient when troubleshooting RS232 links. In this case one can determine whether or not a peripheral is responding correctly to the data being sent to it.

Method 2 is best used to check whether or not the serial link is transmitting data without errors. If there is an error in the transmitted data you can identify which bits are incorrect.

Parallel data paths are a common way of routing data around a pc board. Therefore the bit pattern detector is ideally suited to checking inputs and outputs on data and address buses (providing that they are no greater than 8 bits wide). In parallel mode, method 1 is used to trigger an event when the byte set by the programming switches occurs. This is ideal for microprocessor circuits when you need to trigger an interrupt line. Method 2 is best used for checking data flow through a circuit.

Circuit synopsis

The basic operation of the circuit is as follows. The incoming serial data is converted to TTL levels to enable it to be received in a form acceptable to the General Instruments AY-3-1015D universal asynchronous receiver-transmitter (or UART for short).

The UART primarily converts serial data into parallel data, it also removes start, stop and parity bits from the serial signal. The speed at which the UART runs is set by the baud rate generator.

The buffer is used to select between data from the UART or the parallel input, to be sent to the comparator. The comparator is used to compare the incoming data with the settings on the programming switches. When the incoming data is the same as the settings on the switches the 'pattern detected' LED is illuminated.

The buffered display shows the byte present on the data path on a row of LEDs. These LEDs are driven by non-inverting buffers to reduce loading.

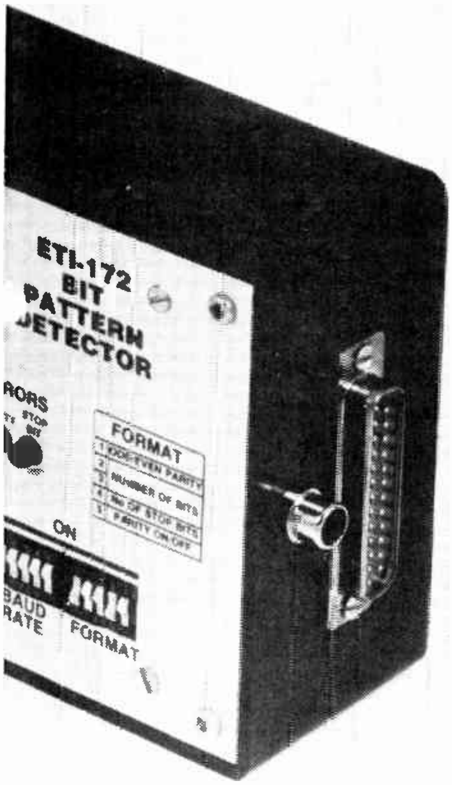
Construction

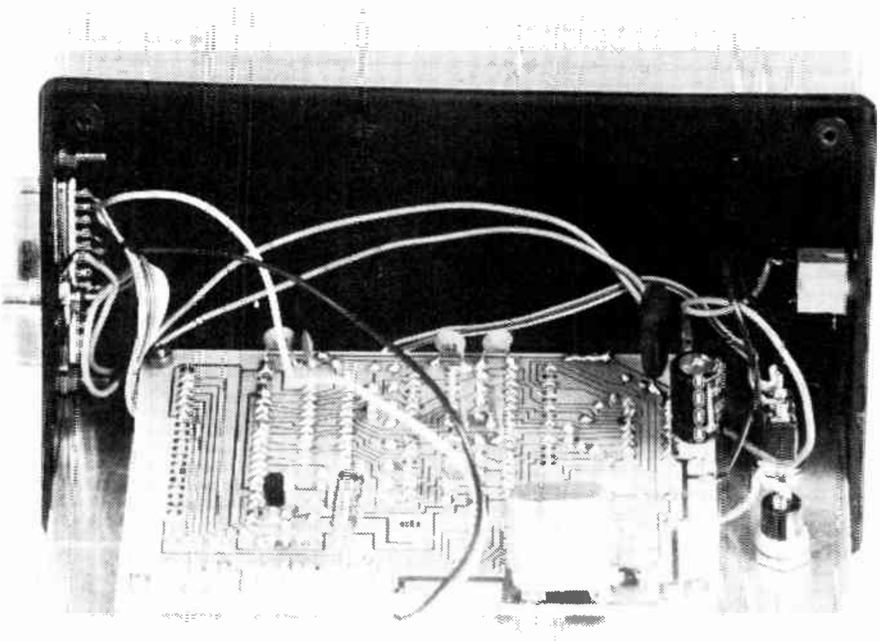
If you only require the bit pattern detector to operate in parallel mode you can omit a number of components. These components are, IC1, IC2, IC8, Q1, D1, R1, R2, R3, R4, C1, C2, C3, C8, C9, C10, C11, LED1, LED2, SW1, SW2, SW3 and the 2.457 MHz crystal.

Before you commence construction examine the pc board for defects such as bridges and broken tracks. Also take note of the components which should be mounted on the copper side of the pc board. These components are marked in colour on the overlay. The reason for components being mounted there is because there is not enough clearance for them between the pc board and the front panel.

The resistors and capacitors should be mounted first. Take note of the orientation of the electrolytic capacitors C2 and C12, and remember that all the capacitors are mounted on the copper side of the pc board.

Next mount the diodes, but first check their orientation. Try to keep the spacing between LEDs 3 to 10 as consistent as possible, keeping in mind that they have to be aligned with the front panel. Also mount the three banks of DIP switches. The crystal, the regulator and the transistor can then be mounted on the copper side of the pc board.





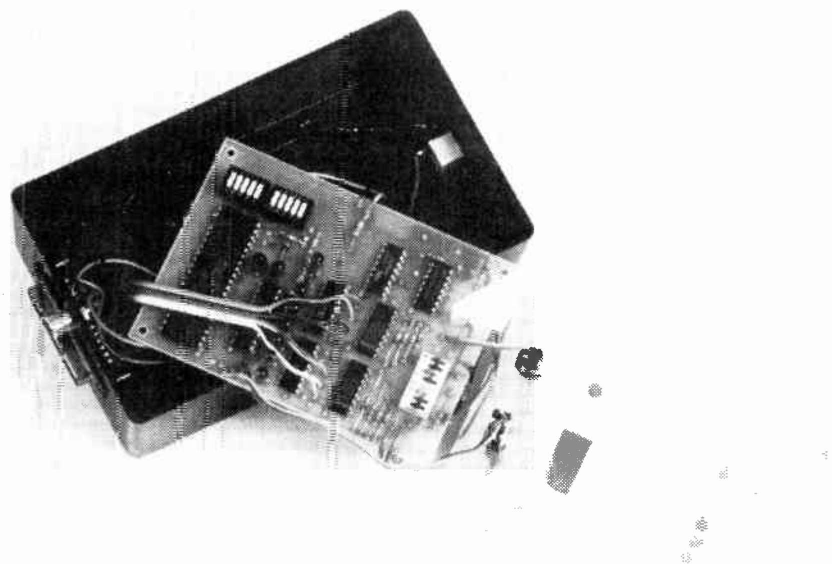
Rear view showing how the pc board is mounted behind the front panel.

Next mount the integrated circuits. All the ICs are CMOS except for IC1 which is NMOS so treat them all with respect. Remember NMOS and CMOS chips are static sensitive; they do not like having their legs touched, only pick them up by their ends.

When soldering in the integrated circuits, try not to overheat them by soldering rows of successive pins. This is especially important in the case of IC1 for which it is best to solder the pins from opposite ends, eg, pin 1 then pin 21, pin 2 then pin 22 etc. This prevents the chip from being excessively heated in one area and saves it from possible damage.

There are mercifully few flying leads to connect in this project, as the majority of the switches and all the LEDs are mounted on the pc board. Take note of the connections to the serial/parallel switch, the DB 25 socket and the 9 volt input socket. The DB 25 socket is also used to access the parallel data input. Table 1 shows the recommended connections. However, as RS232 connectors can be configured in different ways on different computers, check that these lines are not used in your particular case. If they are, you may get false data entering ICs 4 and 5. If some of the recommended lines are used, connect the parallel data input lines to unused pins on the connector.

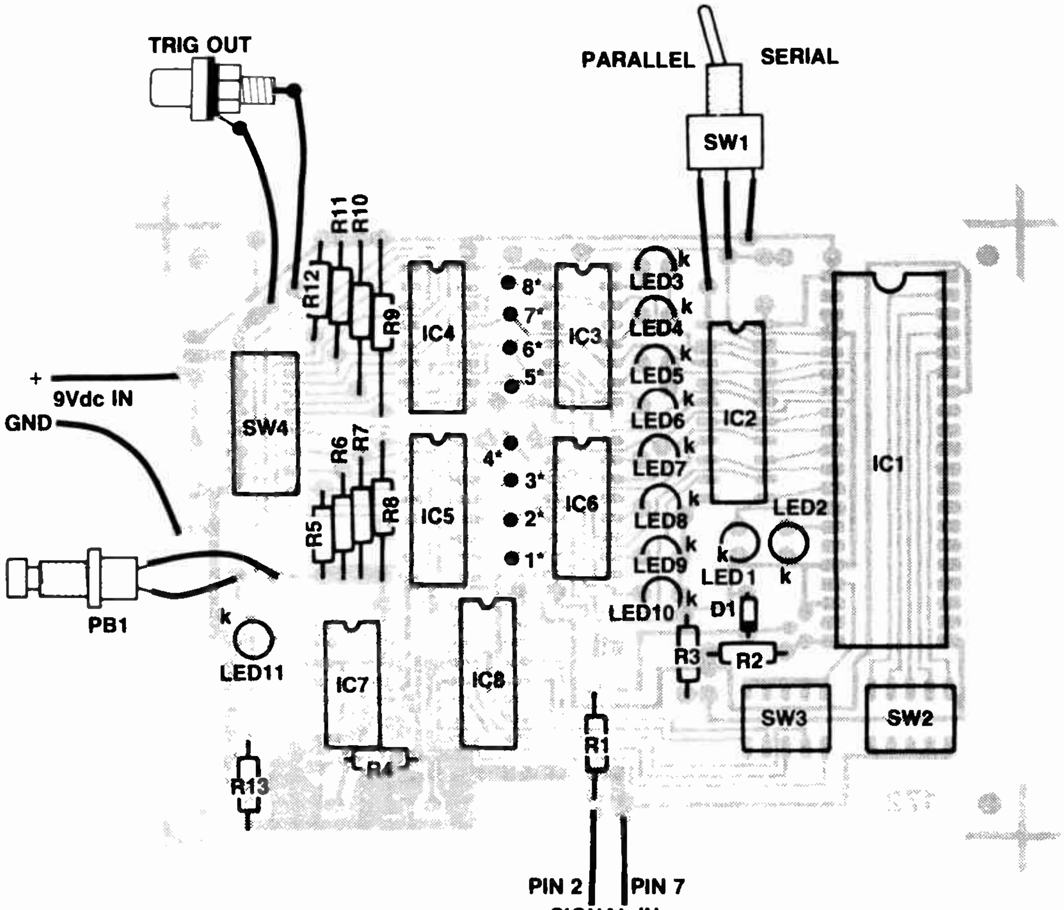
To allow all the LEDs and DIP switches to poke neatly through the front panel requires some careful drilling. To assist in this check the drilling diagram. By accurately



Placement of components on the pc board.

TABLE 1. DB 25 CONNECTIONS

Pin on DB 25 plug	14	15	16	17	18	19	21	22
Pad on pc board	1	2	3	4	5	6	7	8



NOTE:
* INDICATES PIN TO BE
CONNECTED TO ON PARALLEL INPUT

HOW IT WORKS — ETI-172

Some serial signals are 'non return to zero' (NRZ) meaning that a high state is represented by a positive voltage and a low state is represented by a negative voltage. The diode D1 is used to convert any NRZ signals to 'return to zero' signals and the transistor Q1 is used to set the signal level at 5 volts. The resultant signal output from Q1 will comply to TTL standards, that is, 5 volts as a high level and 0 volts as a low level. This signal conditioning is performed because the UART (IC1) requires TTL input signals.

The oscillator circuit consisting of the crystal, the NOR gate (IC7c), R4, C8 and C9 oscillates at a frequency of 2.457 MHz. This signal is divided down by the 7-bit ripple counter, IC8, to provide five different baud rates. These rates are switched to the UART by the bank of DIP switches, SW3.

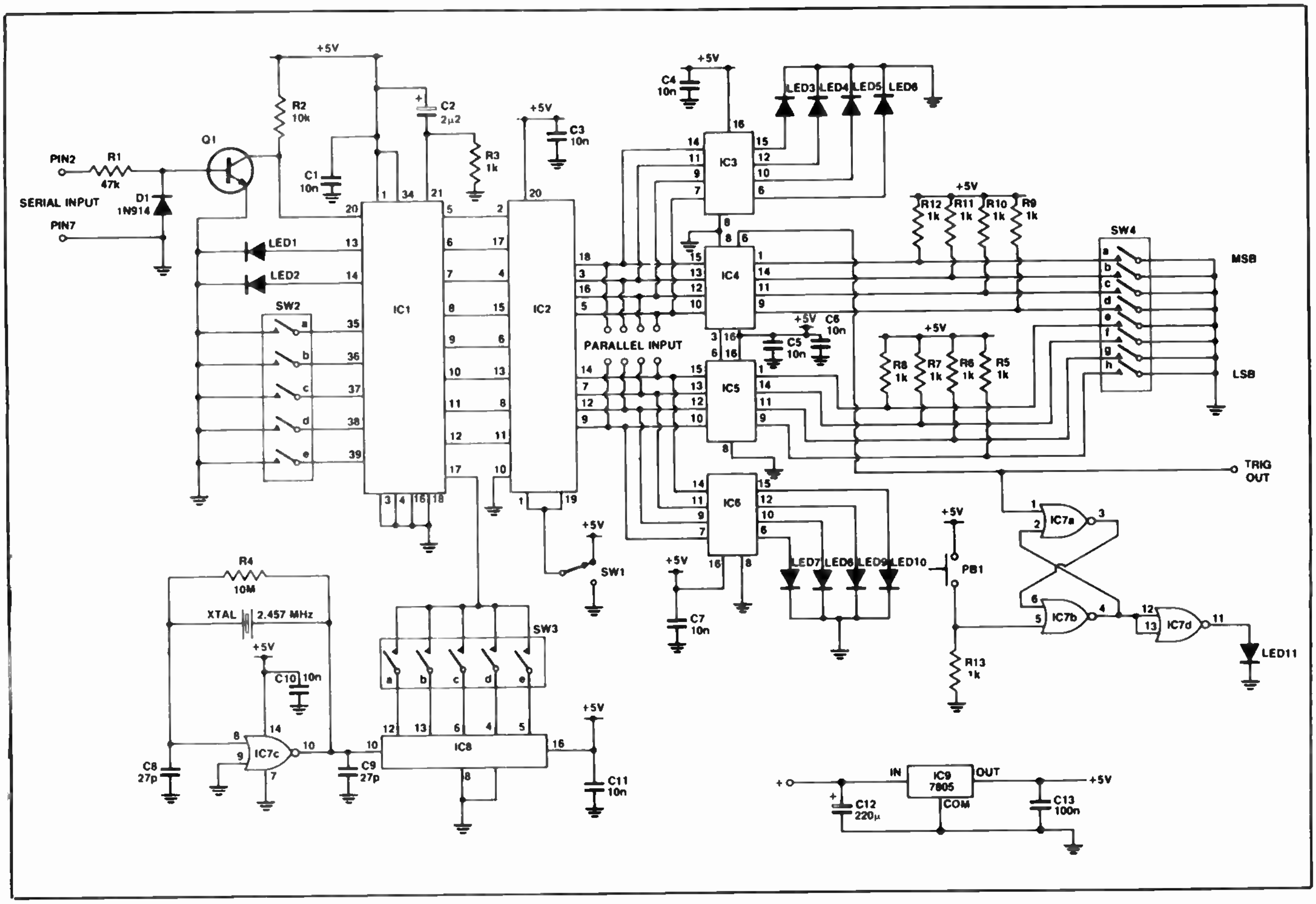
The UART removes start and stop bits from the incoming serial bit stream and converts the serial data into parallel data. SW2 is used to set up the UART for the correct parity, and correct number of data and stop bits. LEDs 1 and 2 indicate errors with parity and stop bits respectively.

After the serial data has been converted into parallel data, it is compared with the settings on the DIP switches (SW4) via ICs 4 and 5. When the bit pattern is the same as the switch settings, pin 6 of IC4 goes low. However, this output goes high again when a bit pattern different from the switch setting occurs. To latch this output an R-S flip-flop is used. This consists of NOR gates IC7a and IC7b. IC7d is used to invert the latched signal to a high level and thus switch on LED11 to indicate that the bit pattern has been detected.

IC2 is an octal tristate buffer which is used to select between serial and parallel incoming data by SW1. When SW1 is in the serial position, the data output from the UART is connected to the comparators. When this switch is in the parallel position the data output from the UART is disabled from the rest of the circuit, isolating the outputs of IC1 from incoming data. Otherwise this data would be fed into the outputs of the UART.

LEDs 3 to 10 display the incoming data in a binary form. They are driven by the non-inverting buffers IC3 and IC5. Thus the loading on IC2 is reduced when the circuit is examining serial data. The loading of the source of the parallel data is reduced when the circuit is examining parallel data.

The 10 nF capacitors connected between the Vcc pin of the integrated circuits and ground remove transients and any other undesirable ac signals which may be picked up by the Vcc rail. On the pc board, these components are located next to the Vcc pins of each integrated circuit. Capacitors C12 and C13 perform the above mentioned task for the incoming dc supply.



PARTS LIST — ETI-172

Resistors.................all ¼ W, 5%
- R1..........................47k
- R2..........................10k
- R3, R5-R13...........1k
- R4..........................10M

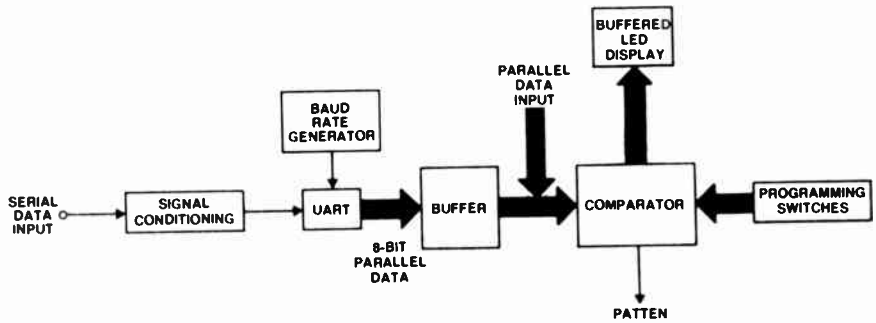
Capacitors
- C1,C3-C7,
- C10,C11................10n
- C2..........................2μ2 25 V electro
- C8, C9..................27p
- C12........................220μ 25 V electro
- C13........................100n greencap

Semiconductors
- D1..........................1N914
- LED 1,2,11............red 5 mm
- LED 3-10..............green 5 mm
- Q1..........................BC549
- IC1.........................AY-1015D UART
- IC2.........................74HC244
- IC3,6......................4050
- IC4,5......................74HC85
- IC7.........................4001
- IC8.........................4020
- IC9.........................7805

Miscellaneous
- SW1.......................SPDT
- SW2,3...................5-way DIP switch
- SW4.......................8-way DIP switch
- PB1.......................pushbutton switch

DB 25 plug; 2.457 MHz crystal; case 160 mm x 95 mm x 50 mm; RCA socket; 3.5 mm phono socket; ETI-172 pc board.

Price estimate: $55 serial/ parallel version
$30 parallel version



Block diagram of circuit.

aligning this diagram over your front panel, it can be used as a drilling template. When you have it in position, lightly punch the centres of all holes. To make the centres more accurate it may help to have a hard surface under the front panel when you are punching it. The outlines for the DIP switches can be marked by cutting through the diagram with a sharp blade or scalpel. This will transfer all the required markings on to the metal.

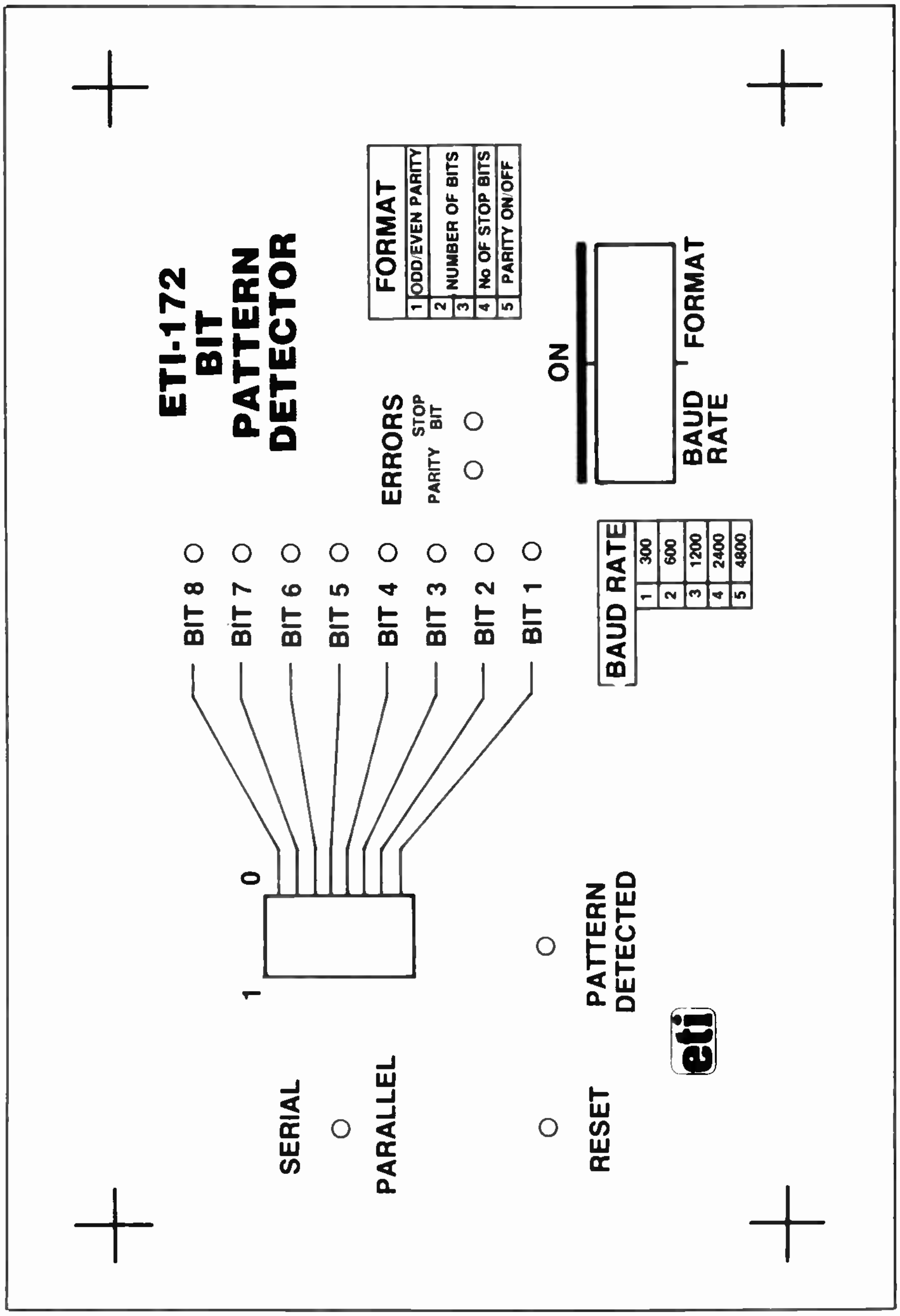
When you are drilling the 5 mm diameter holes it is advisable to drill a pilot hole first, using a 3 mm drill. Now use the 5 mm drill where required. The edges of these holes can be cleaned using a large drill bit. The rectangular holes for the DIP switches can be cut using a nibbling tool or by drilling a series of holes and filing between them.

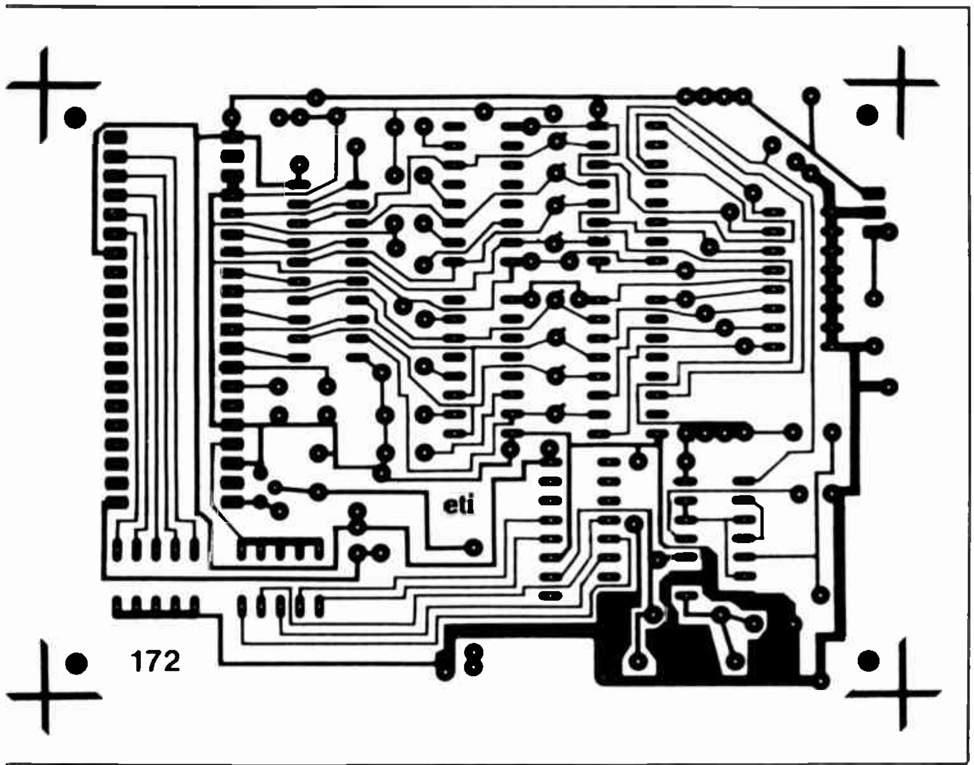
The circuit board can now be mounted behind the front panel using four 6 BA nuts and bolts. Some 5 mm spacers should provide adequate clearance between the front panel and the pc board. With a bit of wiggling the LEDs and the DIP switches should protrude through the front panel. If you have difficulty in getting the DIP switches through, file the holes gradually until they do. If the LEDs give you difficulty, twist them until they are evenly spaced so that they can be located in the holes.

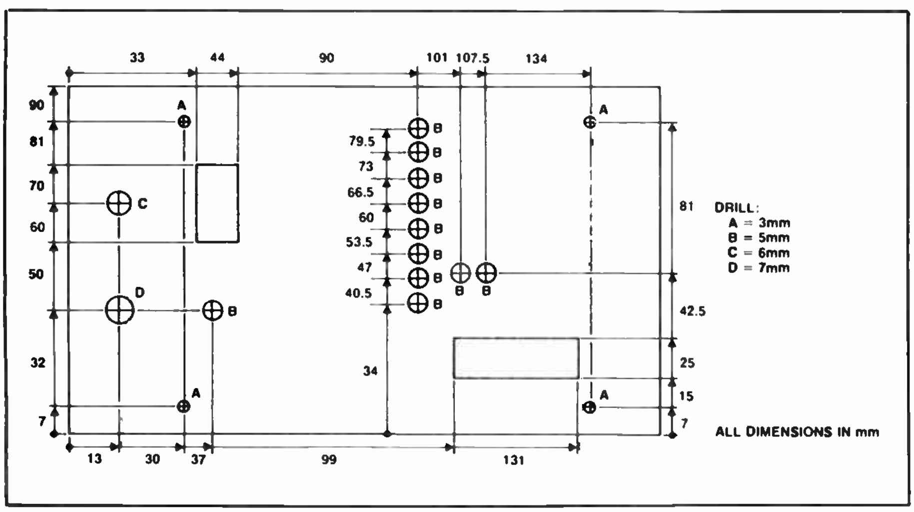
Before you connect in the plugpack carefully examine the pc board for solder bridges; also check that all the components are located and oriented correctly on the board. Now connect the 9 volt plugpack, ▶

TABLE 2. FORMAT SWITCH SETTINGS

Switch No.			
1	on = odd parity		off = even parity
2,3	2	3	no of bits
	on	on	5
	off	on	6
	on	off	7
	off	off	8
4	on = 1 stop bit		off = 2 stop bits
5	on =		off =







and check that the voltage rails are within 500 mV of 5 volts. If not, disconnect the plugpack and check for short circuits on the pc board in the vicinity of the 5 volt supply rails.

To test the bit pattern detector set the 8-bit programming DIP switches to '0' and the pattern detected LED should light up. Switch any of the DIP switches to the '1' position and press the RESET button. The PATTERN DETECTED LED should go out. If this does not occur, recheck the board for shorts, broken tracks or dry joints.

Using it

When detecting bytes on a serial line firstly set up the BAUD RATE and FORMAT DIP switches to suit the serial port. Table 2 shows the settings for these switches. Next set the binary value of the byte to be detected on the eight DIP switches next to the SERIAL/PARALLEL switch. The top switch (bit 8) is the most significant bit and the bottom one (bit 1) is the least significant bit. Now send the data stream or the specific byte to be detected from this port. If either of the ERROR LEDs light up, check that the FORMAT or BAUD RATE DIP switches are set correctly.

To detect bit patterns or bytes on a parallel data bus, firstly flip the SERIAL/PARALLEL switch to the PARALLEL position and connect the relevant pins on the DB 25 connector (listed in Table 1) to the relevant lines on the parallel data bus. The eight LEDs should indicate the state of these lines. If a particular pattern is to be detected set its binary value on the DIP switches and the PATTERN DETECTED LED should light up when it appears on the data bus.●

ESD HAZARD DETECTOR

Electric shock therapy is about the last thing you want for your vulnerable semiconductor components. But electrostatic discharge can be as hazardous as any pair of probes in a cuckoo's nest. This device gives warning of any moving electrostatic field which might zap your precious parts.

Ian Thomas

TO THOSE LESS aware than most, this title might appear to be about a device to detect some of the more brain disrupting chemicals that are currently around. However it isn't brains that this device protects, it's silicon.

Everyone's aware that on a cold dry winter's day you tend to build up an electrostatic charge on your body so that when you touch some metal object such as a tap or door knob you cop a most unpleasant ZAP! This zap is called an electrostatic discharge and if it happens to jump on to a lead of an IC or transistor then the poor device is never quite the same again. In the last few years it has become more and more commonly recognised that semiconductor devices are extremely sensitive to these stray high voltage transients, and MOS devices more than most. From a manufacturing point of view it isn't so much the outright broken bits that matter (they can be found during test) but the bits that are somewhat bent. They appear to work but will go toes-up after the equipment has been in use for a few days or weeks.

IC manufacturers are aware of this problem and go to great pains to build protection into the devices where pins are connected out. There is, however, a limit to what you can do without degrading the device performance. In general the distributors are pretty good and try to do the right thing; but you, dear reader, may not.

An electrostatic charge is generated when two insulating dissimilar materials are rubbed together or even touched together then pulled apart. The degree of dissimilarity and the amount of charge generated is a function of the position of the two materials in an order known as the Triboelectric Series, part of which is given in Table 1. It is

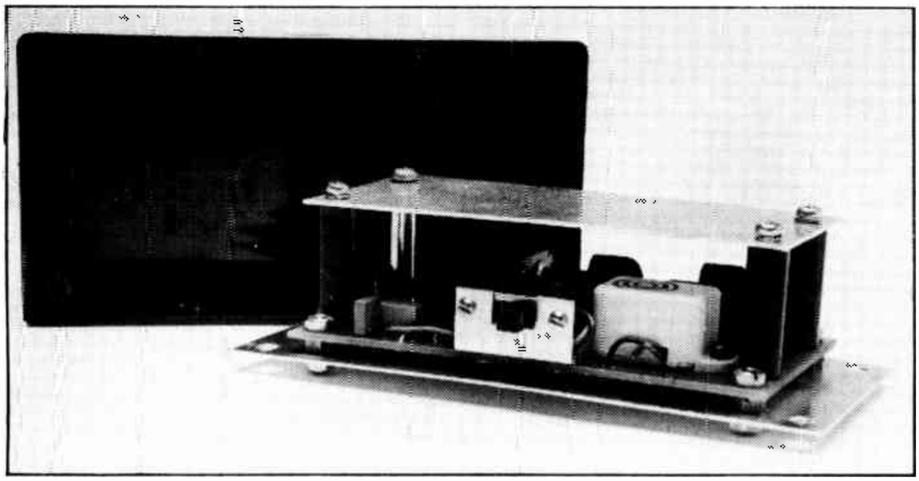


Table 1. THE TRIBOELECTRIC SERIES

Air	POSITIVE
Human skin	POSITIVE
Glass	POSITIVE
Human hair	POSITIVE
Wool	POSITIVE
Fur	POSITIVE
Paper	POSITIVE
Cotton	
Wood	NEGATIVE
Hard rubber	NEGATIVE
Acetate rayon	NEGATIVE
Polyester	NEGATIVE
Polyurethane	NEGATIVE
PVC (vinyl)	NEGATIVE
Teflon	NEGATIVE

also a function of the humidity of the air, intimacy of contact of the two materials and the rate of separation and is in general a pretty inexact sort of thing but there are clear trends. You can see from the table that cotton is pretty much neutral (and hence cotton shirts don't charge up and zap you). In general it's the materials that tend to reject moisture that create the worst charge problems as moisture tends to conduct and bleed charge away.

Table 2. TYPICAL ELECTROSTATIC VOLTAGES

Means of generation	Humidity 10-20%	Humidity 60-90%
Walking across carpet	35,000	1500
Walking on vinyl floor	12,000	250
Worker at bench	6,000	100
Vinyl envelopes as per work instructions	7,000	600
Common poly bag picked up from bench	20,000	1200
Work chair padded with polyurethane foam	18,000	1500

To give you an idea of the sorts of potentials that can be built up, Table 2 shows the effect of various actions on a dry day and a relatively humid one. It's easy to see why you can get a belt by walking across a carpet on a cold winter's day! A normal human body insulated by your average sort of running shoes has a self capacitance in the order of 100pF. A few quick prods at a calculator shows that the stored energy in your delicate body is about 60 millijoules. The energy per discharge in the ETI-342 Pulse Shaped CDI (Feb '85) was only 120 millijoules which explains why it hurts when you touch something. Obviously a piece of silicon, the dimensions of which are measured in micrometres, is blown to a thin haze!

In general MOS devices tend to be the most delicate as a fundamental part of their structure is a very thin layer of silicon dioxide which can be absolutely guaranteed to rupture at the order of a 100 volts or so. For this reason even power MOSFETs that can cope with tens of amps at hundreds of volts can be ruined by one little zap on the gate lead. This certainly doesn't mean that you can do what you want with bipolar devices. If sufficient energy is dumped into the input junction then the device will fail.

In industrial situations where it's absolutely essential that these terrible things don't happen, quite exotic precautions are taken. The top of the bench being worked on is covered with conductive material (usually a type of conductive rubber) which is grounded. The workers wear conductive wrist straps which are grounded (through a large resistor). Ionised air (which is slightly conductive) may be blown over the work area. On top of all this there is a whole bunch of specifications and procedures to cover all phases of assembly. About this stage you're probably thinking it's a miracle anything ever worked and you might as well give it all away. But ETI has a simple, precautionary device that should help.

Design

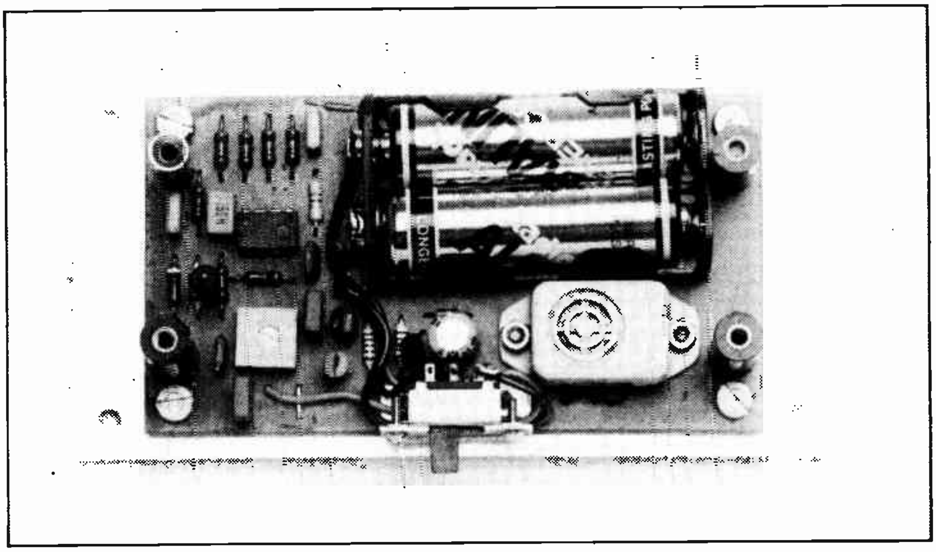
Whenever a body is charged up to a high potential then it is surrounded by an electrostatic field normally expressed in volts/metre, and if the body moves then the field moves with it. If you've just been dancing on your lovely new nylon carpet and charged yourself up to 10 zillion volts then the field from that charge spreads out from your body quite a long way.

It seemed to me that an instrument that detected the approach of this field and sounded an alarm would go a long way to avoiding blasted silicon. At least you would be warned to discharge yourself. It's extremely difficult (read expensive) to detect a completely static electrostatic field but a moving one is not such a great problem. A simple thought experiment will explain the problem and show how a detector could be built.

Suppose we have two parallel plates of a capacitor magically suspended in space 25mm apart and with an area of about 5000mm^2, a not inconvenient size for a detector. The capacitance between its plates works out to be very roughly 2pF. Suppose a field of 500 volts/metre is suddenly imposed across these capacitor plates (definitely ESD hazard conditions!). The potential between the plates will immediately rise to 12.5 volts (500 volts/metre times 2.5/100 metres). The problem is that any attempt to measure the voltage (cheaply) will immediately bleed the charge away. Even a 10 megohm resistor gives a time constant of only 20 microseconds.

This transient could possibly be detected except for another major problem. The whole world is filled with (it would seem) literally thousands of volts/metre of 50Hz mains hum and if a detector was built with a bandwidth sufficient to detect the spikes then it would be continually set off by mains. This means that the detector needed to operate on our parallel plate capacitor must have a filter that cuts off fast before 50Hz to keep mains out. If the detector doesn't respond to frequencies above 50Hz then the input RC time constant needs to be very long, about a second! Hence the problem in building such a device. It must have an incredibly high input impedance or alternatively huge capacitor plates to up the sensing capacitance. With 2pF input capacitance this gives a detector input resistance of 500 gigohms! Even though the input impedance of some MOS input op-amps approaches this figure, it isn't a lot of use as an actual resistor is needed to define reference voltages. As 500 gigohm resistors are just a little hard to buy this basic approach won't work.

It then occurred to me that we were trying to detect voltages in the order of tens of volts. This combined with the fact that these days gain is cheap gave me the clue. If one plate of the sensor capacitor plates was connected to the other through say a 5000pF capacitor then the input tens of volts would be attenuated to fours of millivolts, but then the detector capacitor forms a potential divider with the 5000pF and increases the size of the source capacitance. In this case the needed detector input resistance drops to around 100 megohms to be usable. This is still a bit gross but getting closer. From an engineering point of view ▶



you can get 100 megohm resistors but they are expensive and hard to procure. About the largest I could find at Geoff Woods' was 10M. He had 22 megs but I settled for 10M for reasons I'll go into later.

The next problem is how one can magically change a 10 meg resistor into a 100 meg resistor. The method goes by the delightful term "bootstrapping". Consider a resistor with a voltage applied to one end as shown in Figure 1. The current through the resistor due to the imposed voltage is I. If a buffer amplifier with a very high input impedance is connected to the top of the resistor with a gain of say 0.9 and its output is connected to the earth end of the resistor then the voltage drop across the resistor is no longer V as in Figure 1 but V-0.9V or only 0.1V. This means that even though the full V volts is being imposed on the resistor, the current flowing into it is only 0.1I so the effective resistance is multiplied by 10. This would seem to be a pretty good trick to use. It does have bad side effects like multiplying all offset problems by 10 but if these can be managed then a 10 meg resistor can be changed into a 100 meg resistor.

Given the ability to artificially create a very high input impedance amplifier, the idea of a detector of changing strong electrostatic fields seemed to be quite workable. It should be in a small box to one side of the work area and preferably run off batteries. As the detector does not need a large bandwidth (in the final device there is a lot of circuitry to reduce it) a very low power op-amp such as the National LM4250 can be used. The power drawn by this IC can be set to almost any value required by an external resistor. The op-amp bandwidth varies accordingly but even at microamp levels has more than enough. The LM4250 is specified down to 3 volts total supply voltage so the detector could be powered off two penlight cells. If the total IC power drain is kept below a few microamps then the life of the cells in the detector is effectively their shelf life.

Circuit details

The actual filter used to attenuate the 50Hz is straightforward and 10M resistors were used throughout. As there is effectively 30 megohms between the positive input and ground it is necessary to add in about 30 megohms between the negative input and the output to balance the effect of bias currents in the op-amp. For the LM4250 operating at a few microamps the offset current (that is the *difference* in bias currents of the op-amp inputs) is still up to 5 nanoamps. With 30 megohms this generates a differential offset voltage at the input in the worst case of 150 millivolts. This could be adjusted out with an offset adjust pot but the problem is further aggravated by the bootstrapping with R7 and R8. C7 had to be added to remove both the bootstrapping and the high gain for very low frequencies or the input simply floated all over the place.

However, for frequencies between 0.5Hz and 15Hz the detector gain is pretty much flat and set by the ratio of R10 to R7+R8. When I slogged through the algebra to calculate the transfer polynomial for the circuit, one very interesting fact emerged: the gain of the detector is proportional to the size of C8. You will recall that the input to the detector is actually a capacitive divider made up of the detector plates and C8 so the larger C8 is the higher the input capacitive divider attenuation *but then so is the amplifier gain!* They cancel out. Most serendipitous! Once again, there are always limits like offset problems that don't show up in the mathematics but basically it's whatever is convenient in the filter for C8.

Transistors Q1 and Q2 act to turn on the Darlington transistor Q3 when the op-amp output exceeds one V_{be}. I wanted a nice loud alarm when there was a hazard situation so I used one of the small 3 volt self contained alarms. They're nothing if not loud. A quick estimate of the board area needed showed it would fit nicely into a plastic jiffy box 130mm x 70mm x 40mm *with a metal bottom*. The metal case bottom serves as one plate of the sensing capacitor so plastic cases aren't any good.

Construction

Everything is mounted on the pc board including the batteries. You can copy the artwork provided or purchase transparencies from ETI, or do it the easy way and buy from the kit suppliers. However you do it, try to stick exactly to the layout given.

Mounted over the board on 25.4mm spacers is the second plate of the sensing capacitor. Three of the spacers are made of insulating material and the fourth is metal and carries the received voltage to the detector input. I even chose to put a tiny slide switch and the buzzer on the board (both of which I got from Tandy) just to make things complete. The switch is included in case a situation arises where high fields can't be avoided (say you're building a Van de Graaff generator and you don't want to have to throw it out the window).

Before the board components are assembled it's a smart idea to use it as a template to drill the holes for the base plate and the detector plate.

Hold the board *exactly* in the centre of the base plate for the four mounting holes. A dimensional drawing is given for the detector plate but it's a lot easier to simply hold the board and plate together and drill them (more accurate too!). *Note that the detector plate is offset if you wish to use the small slider switch mounted on the board.*

Make sure that the IC and the electrolytic capacitors are in the right way. The battery case I used was one which held four penlight cells. You can get battery holders that only take two cells but if you look at where the holder is mounted you'll see that it would be impossible to get one of the cells in. The four cell holders come in various brands, some with nickel plated steel contacts and some with aluminium. The aluminium ones aren't so good here as it's necessary to solder to the springs and you can't solder aluminium! It's difficult enough with the nickel.

Carefully cut away the side of the battery holder that doesn't have terminal clips. This will leave you with a spring protruding which should be straightened out. On the piece of holder that was cut away there is a second spring still riveted to the plastic. Drill, bash or bend the rivet out and retrieve the spring. Straighten out the end of the spring that was riveted so it can be inserted in a hole in the printed circuit board. Mix up some Araldite or similar and glue down the battery holder then insert the straightened out spring so as to make a neat two cell battery holder as pictured.

The next and nastiest bit is to solder the nickel plated springs on to the large ground area of the printed board. Nickel is a beast of a material to solder and I must confess I weakened and used el corodo Bakers soldering flux but it has to be washed off com- ▶

pletely. The buzzer is screwed down and both its leads soldered into the relevant pads. Be careful as the polarity must be right. Finally the board must be cleaned absolutely scrupulously of all fluk and residue. There are a lot of high impedances and leakage could be a nuisance.

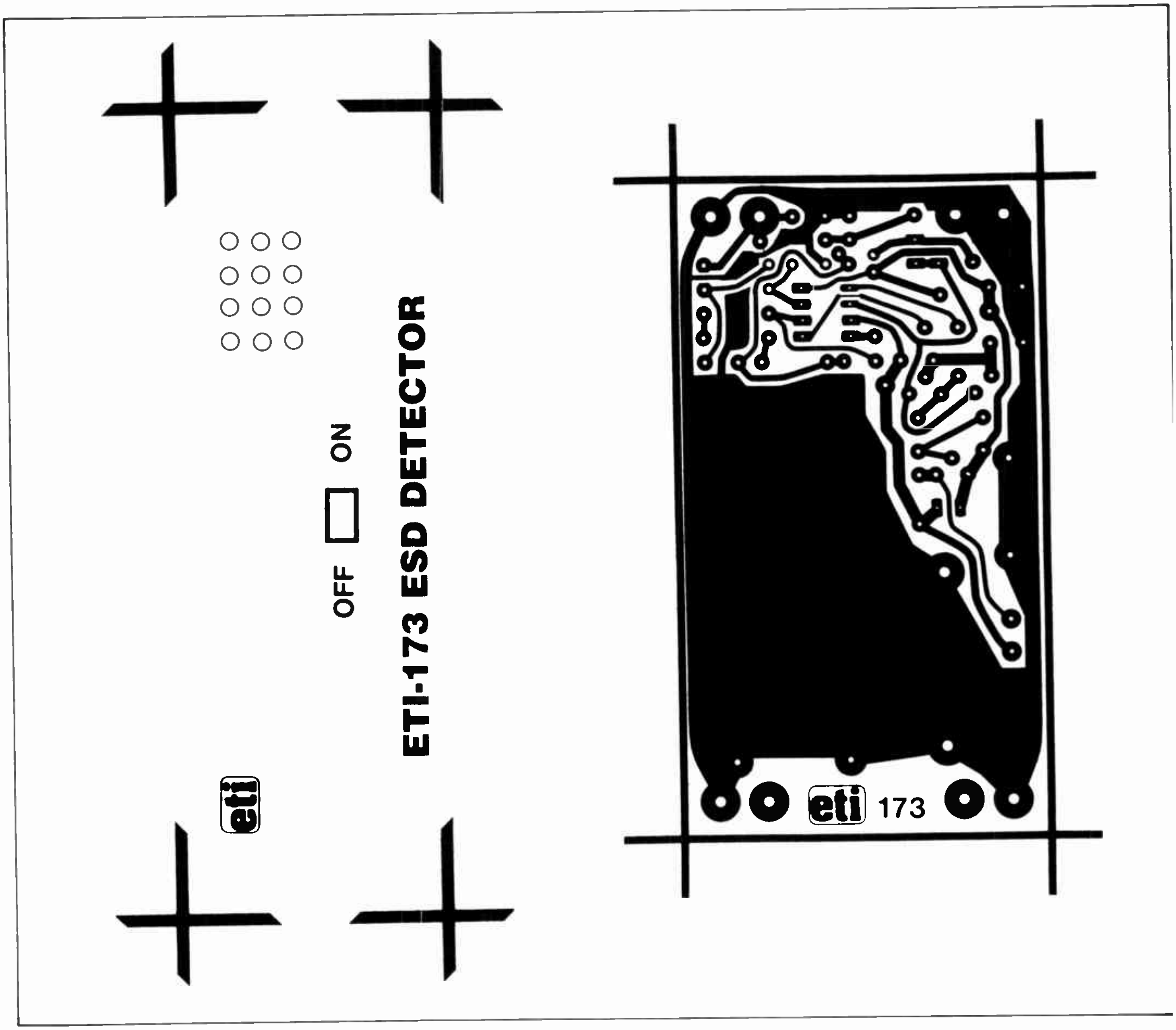
You need three insulating spacers and one metal one to support the pickup plate. The metal one connects to the pad that has C8, R1 and R2 connected to it. If you choose to use the little slide switch you'll find it necessary to trim one side of the pickup plate so the electronics can be assembled into the box. If you use another type remember that both the positive and negative supplies must be switched. I didn't bother with the clip leads for the battery holder but simply soldered pieces of hookup wire on to the terminals as they will never be disconnected.

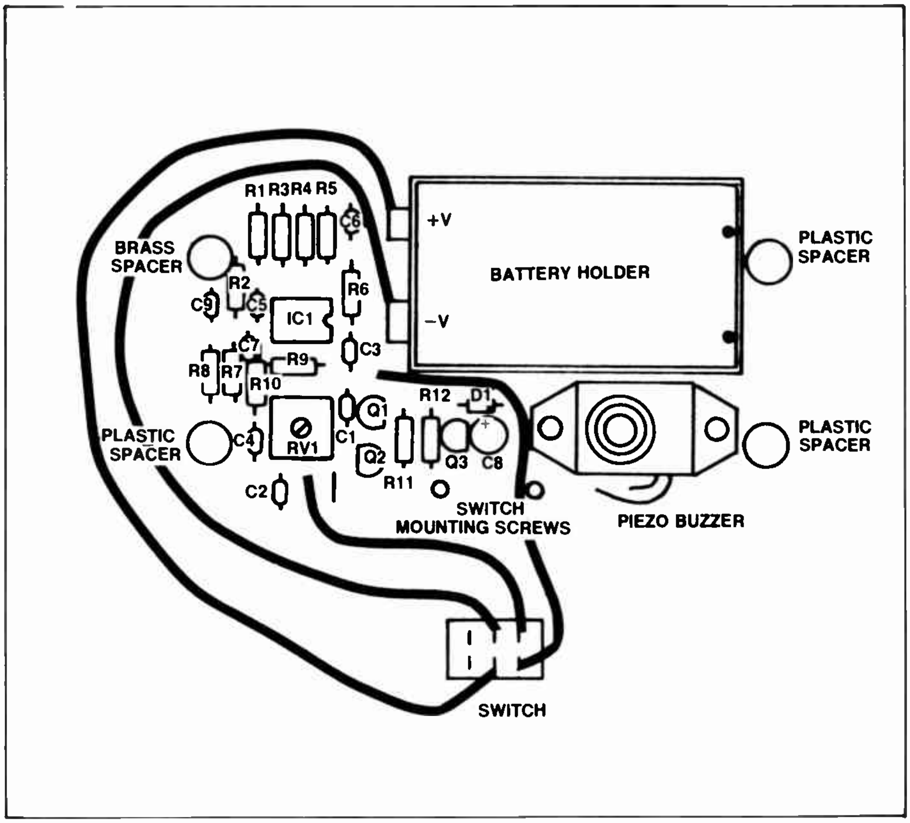
The board assembly is mounted on the base plate of the box with four 4BA 3/8 screws. The screws are passed through the board and one nut done up tight on each screw, then the protruding ends of the screws are passed through the four holes in the base plate and another four nuts done up to hold the board neatly in place.

Testing

Testing is really quite easy. Insert two batteries and turn it on. The alarm should sound for about 20 seconds or so as C7 charges to the offset voltage then it shuts off. Let it settle for a few minutes then measure the voltage between pin 6 of the IC and ground. Adjust RV1 until it averages out to zero. This is a fiddly and odious task as you only have to (literally!) wave your hand near the detector input and pin 6 slams up against the rails. Fortunately, it isn't all that critical. Once this is done, screw on the detector plate and the unit is ready to go into the box.

Finally before fitting the unit together you'll have to cut a hole for the power switch. It makes sense to drill a few holes to let the buzzer sound out. Slip the unit into the box and screw it in place with the four screws provided then turn it on. If you keep well away from it, it should sound for 10 seconds or so then go quiet. Just waving your hand close over it or touching the box will set it off. Then you may leave it on in a corner of your workbench and breathe a bit easier! ●





ETI-173 — PARTS LIST

Resistors................all ¼W, 2% metal film
- R1, 2, 3, 4, 5 .... 10 megohm
- R6 .... 4M7
- R7, 12 .... 1k0
- R8 .... 9k1
- R10, 9 .... 2M2
- R11 .... 100k
- RV1 .... 100k trimpot

Capacitors
- C1, 2 .... 100n 10% met poly*
- C3, 4 .... 1n 10% met poly*
- C5 .... 150p 10% met poly*
- C6 .... 6n8 10% met poly*
- C7 .... 100µ 10% 3V tag tantalum
- C8 .... 47µ Al electro
- C9 .... 4n7

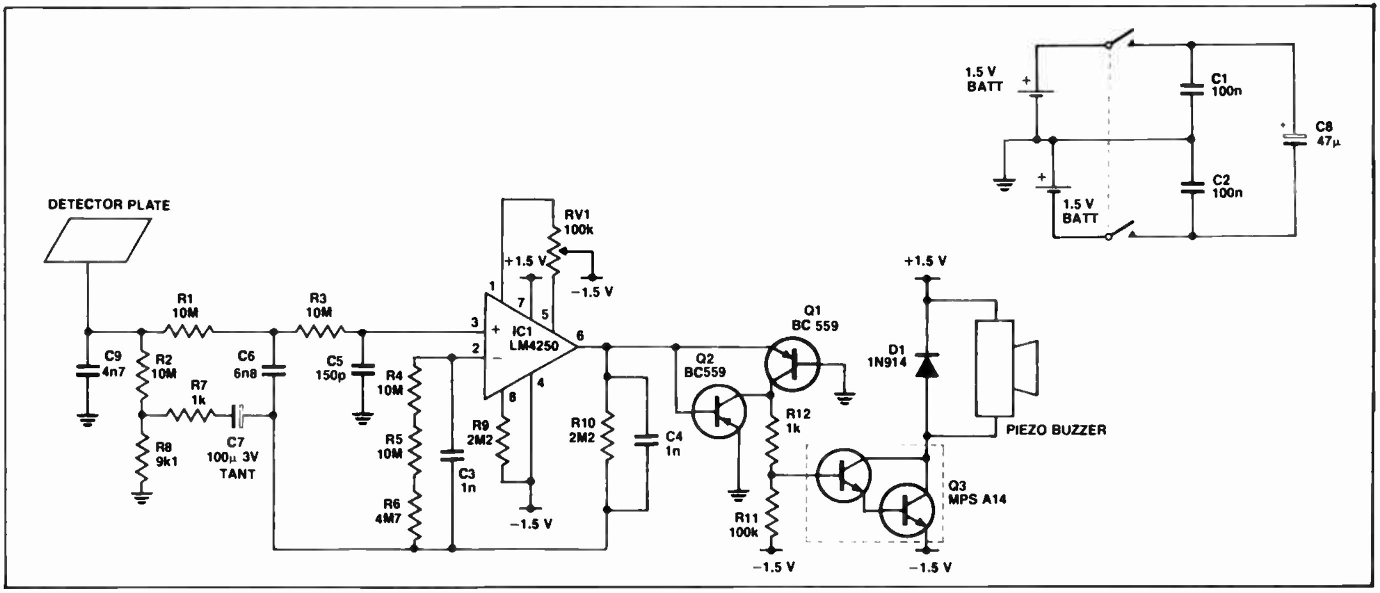
Semiconductors
- IC1 .... LM4250
- D1 .... 1N914
- Q1, 2 .... BC559
- Q3 .... Motorola MPS-A14

Miscellaneous

ETI-173 pc board; 2 x AA batteries; battery holder; Piezo buzzer; slide switch; 3 x 25.4mm insulated spacers; 1 x 25.4mm metal spacer; hookup wire; 130mm x 70mm x 40mm jiffy box with metal bottom; 5cm x 100cm metal plate; 4 x 4BA ⅜" screws and nuts; Scotchcal front panel.

Price estimate: $25

**The prototype used metallised polyester film caps such as ERO MKT1817 or MKT1826, or Wima PR21 or PS21 but greencaps could be substituted.*



ETI-173 — HOW IT WORKS

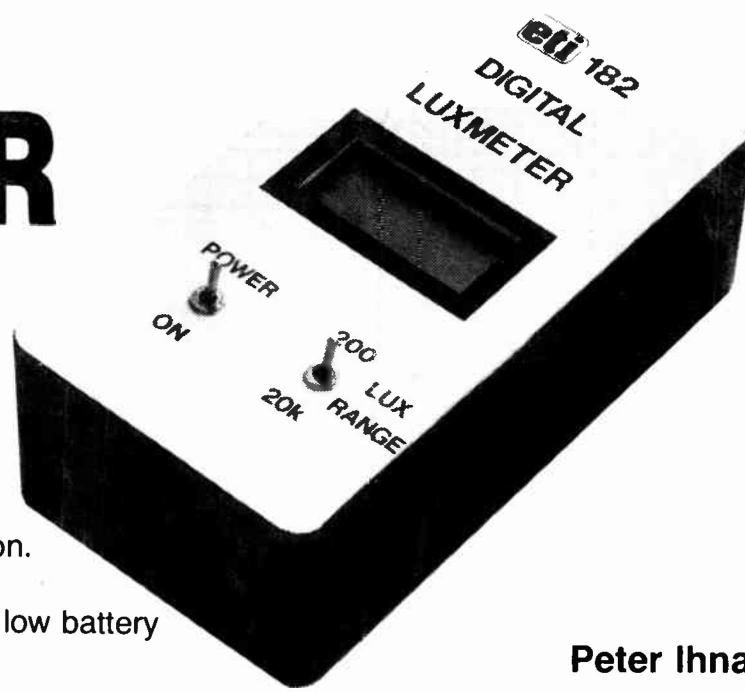
The alarm detects electrostatic fields by sensing the potential difference they generate on a parallel plate capacitor. One plate is the base plate of the case and the other is a plate held by spacers above the detector.

Resistors R1 and R3 form a low pass filter with C5 and C6 to attenuate mains hum. Further attenuation is provided by C4 which rolls off the operational amplifier IC1's response. R7 and R8 serve two purposes. The first is to act as a gain determining element which, together with R10 sets the gain of the op-amp at 220. Also the node of R7 and R8 has a voltage of 0.9 times the voltage on the positive input of the amplifier (and the negative input too — such is the nature of operational amplifiers). R2, the input bias setting resistor, is connected to this node so that any signals that appear at the input to the detector are also imposed, multiplied by 0.9, on the other end of R2. This is known as bootstrapping and effectively multiplies the value of R2 by 10.

The overall effect of this is that any voltage that would be induced on a capacitor with plates about 20mm apart appears with a low source impedance at the output of the op-amp, attenuated 10 times when everything is taken into account.

Transistor Q1 is turned on if the op-amp output goes more than 0.6 volts positive and Q2 is turned on if it goes negative. Both collectors drive the base of the Darlington transistor Q3 which in turn powers the alarm. Thus excursions of ±0.6 volts set off the alarm or, tracing back through the circuit, the detector will pick up fields of greater than 300 volts/metre.

DIGITAL LUXMETER



This instrument is a portable, battery operated device for measuring illumination. It covers light levels from below 1 lux up to 20k lux in two ranges and includes low battery indication.

Peter Ihnat

ETI, IN THE PAST, has described many instruments for measuring just about anything from heart rate through to passion (yes, we published a passion meter once long ago !!!). One quantity however which seems to have missed out is *light*.

In the last twelve months we described two devices which measure light, but not in absolute terms. They were both darkroom exposure (or light) meters. These 'measure' the amount of light in a particular area of the image produced by an enlarger by comparing the intensity with that of a value obtained when a test print was made. Unfortunately, outside the darkroom this technique is of limited use.

The current project can measure light in absolute terms, just as we measure current in amps and frequency in Hertz. This results in a versatile unit for anyone having anything to do with lighting systems. Some examples include, photographers, electricians who fit lights into classrooms, offices and factories or even the home video nut who insists on video taping under very low light levels . . . the list is endless.

Description	illumination
Full moon	0.4
Candle flame at 1 m	1
Highways	20-30
Living rooms and offices	300-400
Shops, workshops and classrooms	500
Area for fitting components to pc board	800
Precision engineering workshops, drawing offices	1000
High precision work eg, repairing watches	3000
Bright summer day	100 000

Table 1. Illumination levels in lux.

Why do these people need to know the level of illumination? The answer is rather obvious in the case of photographers, movie and video camera operators. They need to ensure that the illumination of a scene or subject is sufficient to produce images with the maximum amount of detail. Poor lighting setups can only produce poor images.

In the case of classroom, factory and office lighting, there are recommended illumination levels to suit the type of work performed in each area. To give the reader an idea of illumination available from some common sources and some recommended levels, refer to Table 1. As you may notice, the eye (like the ear) has a wide dynamic range.

One problem in the past with designing a luxmeter was that a suitable sensor was not available in this country. Only fairly recently has one of the few photodiodes which covers more than the red and infra-red regions of the spectrum become readily available in Australia. Ladies and gentlemen . . . introducing the BPW21 photodiode.

Figure 1 shows some of the photodiode's characteristics. Its peak spectral response is around 555 nm, corresponding to yellow/green light. Its spectral range is 350 to 775 nm which almost matches that of the human eye (see dotted line labelled as Vλ). For many applications this is quite adequate and makes it ideal for monitoring daylight or artificial light. One more important feature is that its short circuit current versus illumination is highly linear over a wide range (0.01 to 100k Lux) . . . truly a remarkable sensor.

Figure 2 shows the operation of the luxmeter in block diagram form. The BPW21 is connected in the standard way (reverse biased and into a short circuit) to produce a current in direct proportion to illumination. This is converted to a voltage which is fed into a voltmeter and displayed in digital form. Normally, a circuit such as this would be rather complex but the availability of the ICL7106 digital voltmeter IC reduces the parts count dramatically. Those interested in the digital voltmeter circuitry can refer to two previous ETI articles — ICL7106 data sheet ETI October 1977 and project 161 ETI August 1982.

Contruction

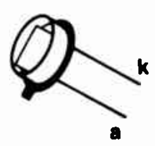
Construction should present few problems if the recommended pc board is used. The layout is not critical and other forms of construction such as Vero-board, may be employed. Use of the recommended board does result in a very compact unit and if correctly assembled will help to ensure that everything works first go.

Firstly, inspect the pc board for broken tracks or shorts — check carefully in the areas where tracks pass between IC pins. If all is OK, start by fitting the three wire links. Note that one link has a 90° bend in it.

Next, mount the resistors, capacitors and trimpots in that order — it may be necessary to bend the leads of capacitor C6 inwards slightly to fit in its correct position. The 40- ▶

Silicon photodiode
with incorporated Vλ filter

BPW21

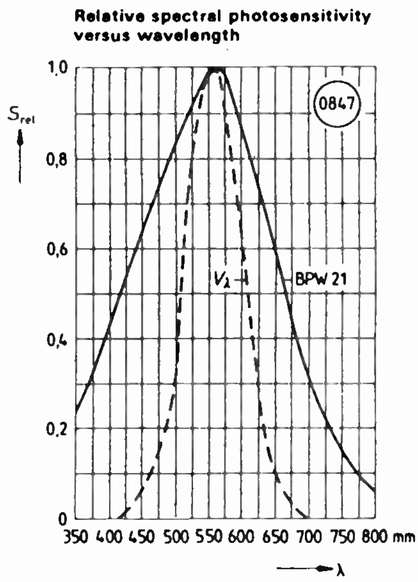


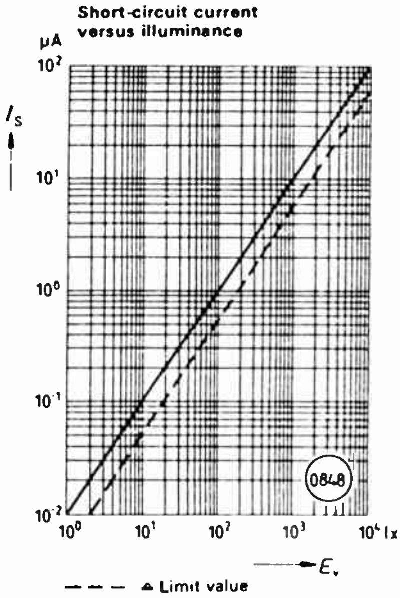
Special features:

- High reliability
- No testable degradation
- Low noise
- High open-circuit voltage as photovoltaic cells
- Detector for low illuminance
- Short switching time
- High photosensitivity
- Strong logarithmic relation between V_o or I_s and illuminance of 10^{-2} to 10^5 lx
- Wide temperature range
- Suitable in the range of visible light

Characteristics (T_{amb} = 25°C)

Photosensitivity (V_R = 5 V, standard light A, T = 2856 K)	S	9 (≥ 5,5)	nA/lx
Wavelength of max. photosensitivity	$\lambda_{S\,max}$	550	nm
Spectral range of photosensitivity (S = 10% of S_{max})	λ	350...775	nm
Radiant sensitive area	A	7.34	mm²
Dimension of radiant sensitive area	L x B	2.71 x 2.71	mm
Distance chip surface to case top edge	H	1.9...2.3	mm
Half angle	φ	60	degrees
Dark current (V_R = 5 V)	I_R	2 (≤ 30)	nA
(V_R = 10 mV)	I_R	8	pA
Spectral photosensitivity (λ = 550 nm)	$S\lambda$	0.21	A/W
Quantum yield (λ = 550 nm)	η	0.47	Electrons / Photon
Open-circuit voltage (E_v = 1000 lx, standard light A, T = 2856 K)	V_O	390 (≥ 320)	mV
Short-circuit current (E_v = 1000 lx, standard light A, T = 2856 K) (Deviation of I_s linearity in the range of $3 \cdot 10^{-2}$ to 10^4 lx: max. 12%)	I_S	9 (≥ 5,5)	µA
Rise and fall time of photocurrent from 10% to 90% and from 90% to 10% of final value (R_L = 1 kΩ, V_R = 10 V, λ = 550 nm, I_P = 9 µA)	t_r, t_f	1	µs
Forward voltage (I_F = 100 mA, E_e = 0, T_{amb} = 25°C)	V_F	1.2	V
Capacitance (V_R = 0 V, f = 1 MHz, E_v = 0 lx)	C_0	750	pF
(V_R = 10 V, f = 1 MHz, E_v = 0 lx)	C_{10}	220	pF
Temperature coefficient of V_O	TC	-2.6	mV/K
Temperature coefficient of I_S	TC	0.12	%/K





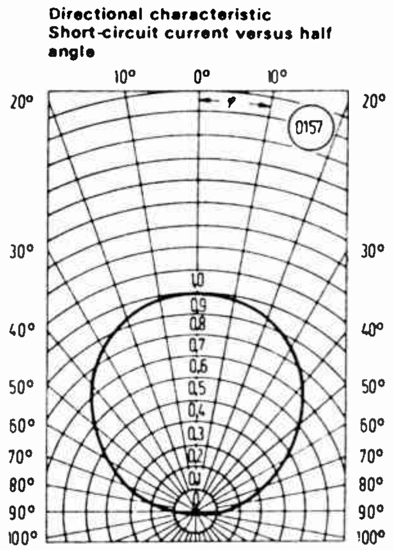


Figure 1. Silicon photodiode characteristics.

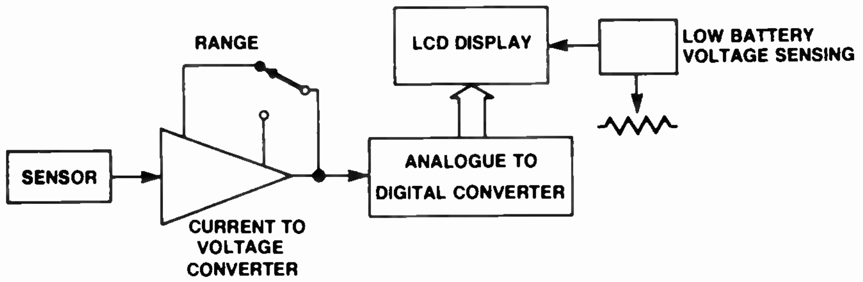


Figure 2. Block diagram.

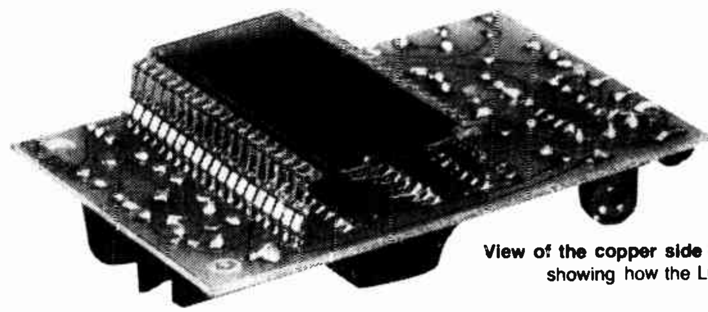
pin IC socket can now be inserted and soldered.

The next part of the construction is quite tricky and so work carefully. The LCD display mounts on the COPPER SIDE of the board using the Molex pins. Firstly, cut two 20-pin lengths of Molex pins. Mount one row in its correct place on the copper side of the board but don't push it all the way down.

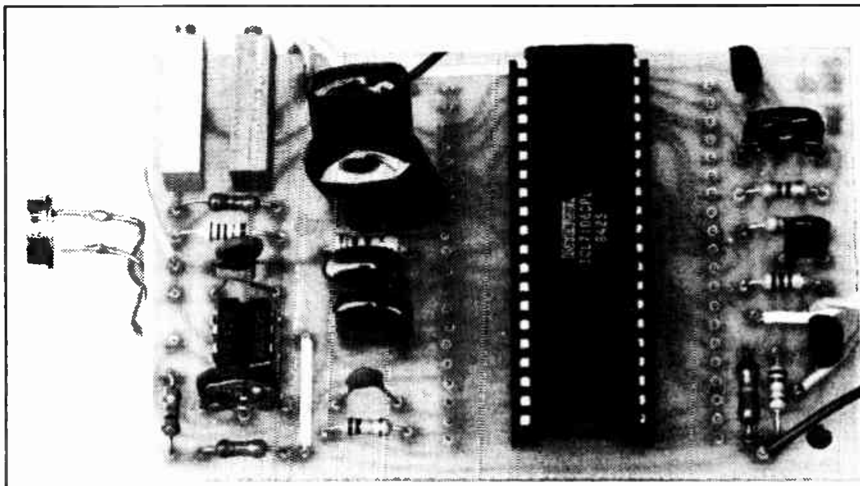
I found the pins easiest to solder if their ends sit flush with the component side of the board. This leaves a small part of the Molex pin's legs sitting above each copper pad ready to solder. Lean the strip inwards slightly, then with a fine tipped iron, solder each pin. Repeat for the other strip.

Before breaking off the Molex support strips inspect the pc board around all soldering. Due to the proximity of conductors, it's fairly likely that a solder bridge may have been formed. These must be removed before continuing.

If you are happy with the construction so far, break off the Molex support strips and mount the transistors and IC1 (check orientation). Don't use a socket for IC1 since this may introduce future problems (the input from the photodiode will be in the order of nanoamps so a direct connection from sensor to IC is preferred). Take normal CMOS precautions when handling the ICs. Plug in IC2 and then finally, plug in the LCD display. ▶



View of the copper side of the pc board showing how the LCD is mounted.



View of the component side of pc board.

PARTS LIST — ETI-182

Resistors.................all 1/4W, 5%
R1, R6, R8.............100k
R2, R4...................1k
R3..........................27k
R5, R9...................1M
R7..........................47k
R10, R11...............4M7
R12........................680k
RV1.......................100k ten-turn trimpot
RV2.......................1k ten-turn trimpot
RV3.......................100k trimpot
RV4.......................500k trimpot

Capacitors
C1..........................180p ceramic
C2..........................56p ceramic
C3, C5...................100n, 50 V mylar
C4..........................100p NPO ceramic
C6..........................470n, 50 V mylar
C7..........................220n, 50 V mylar
C8..........................10n mylar

Semiconductors
IC1.........................CA3130
IC2.........................ICL7106
Q1.........................BC547, 8 or 9
Q2.........................2N5458, MPF106
D1..........................BPW21 photodiode.

Miscellaneous
SW1,2...................DPDT miniature toggle switches

LAD204(or similar) liquid crystal display; ETI-182 pc board; Scotchcal front panel; 150 x 80 x 50 mm multi-purpose box; 9 volt battery clip; three 20 mm 6BA bolts (countersunk head); 9 nuts.

Price estimate: $45-$50

HOW IT WORKS

The block diagram shown in Figure 2 gives the overall operation of the unit. Very basically, illumination is converted into a current, current into voltage and then voltage into digital readout.

Photodiode D1 produces an output current proportional to illumination. If this is fed into a short circuit, the current versus illumination characteristic becomes extremely linear, an important feature for any light measuring instrument.

The CA3130 op-amp converts this current into a proportional voltage dependent on the value of the feedback resistance (the op-amp tries to make its input current equal to zero so it produces an output voltage which forces a current to flow through the feedback resistors to balance the input current from the photodiode). Capacitor C1 is included for stability.

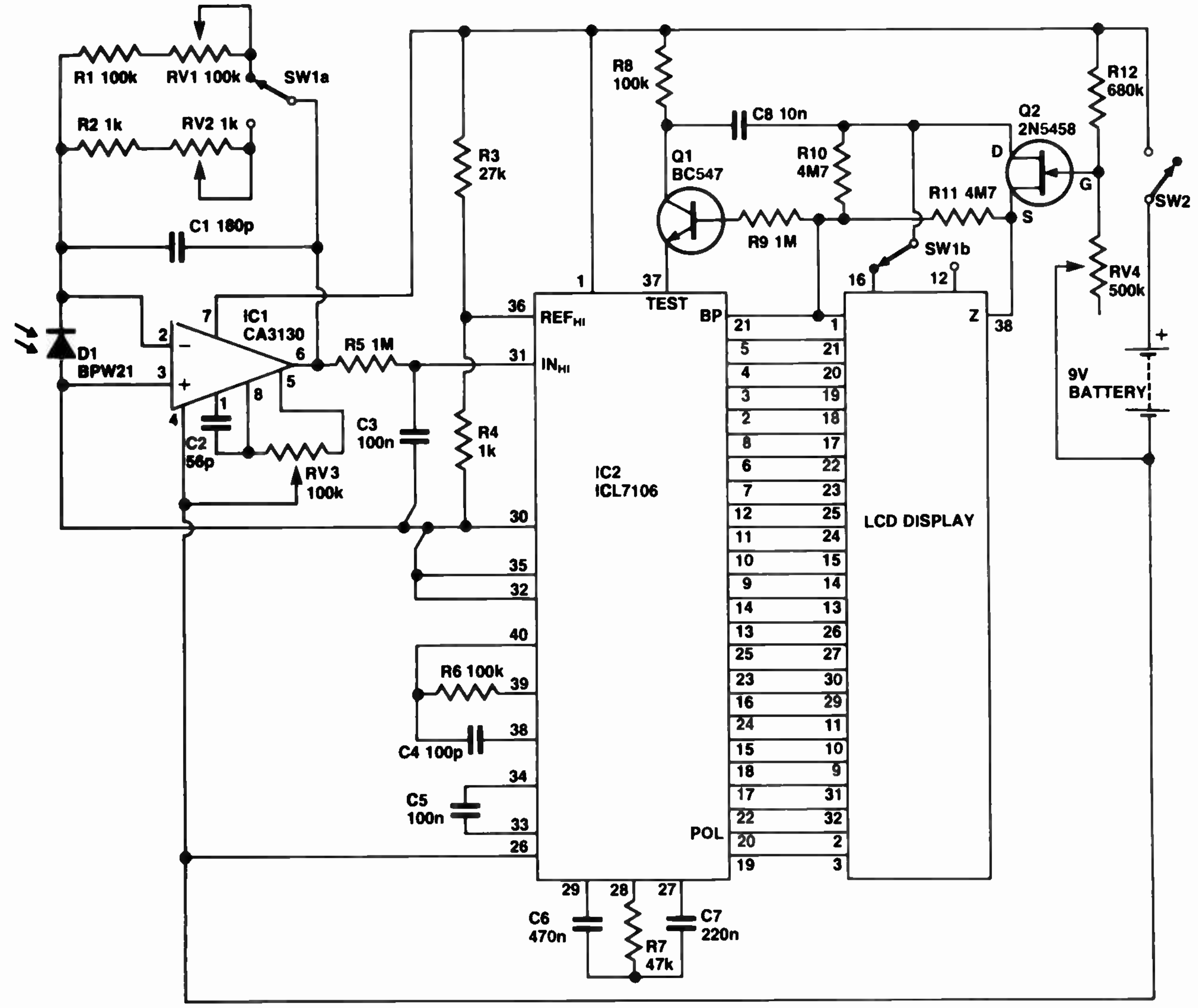
The output from the op-amp will contain an ac and dc component if mains operated lighting is being measured. R3 and C3 act as a low pass filter and reduce the 100 Hz component. The resultant voltage is fed into a ICL7106, a digital voltmeter IC which performs all the hard work of displaying its input voltage. More details of the operation of the ICL7106 can be found in the references given in the main text.

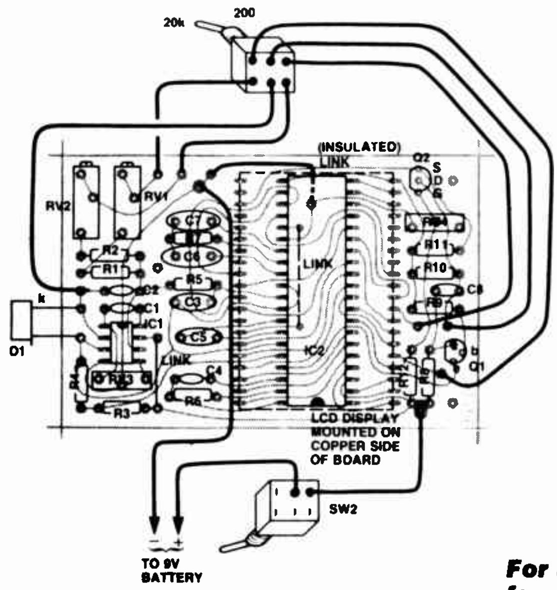
One interesting aspect of circuit operation is the method used to provide split rail voltages for the op-amp. The ICL7106 has a built in voltage reference of approximately 2.8 volts between pin 1 (Vcc) and pin 32 (common).

By connecting common, REF LO (pin 36) and IN LO (pin 30) together, this produces voltages of +2.8 and −6.2 to power the op-amp if measured with respect to the fake earth (junction of common, REF LO and IN LO). In actual fact, if the voltages are measured with respect to the negative terminal of the battery, the fake earth point is at +6.2 volts and Vcc is at 9 volts.

To provide a full scale reading of 200 mV, the voltage applied between REF HI and REF LO should be 100 mV. This is provided by voltage divider R3 and R4.

The rest of the circuitry switches the decimal points and performs the low battery monitoring function. This operates exactly as for project 161 Digital Panel Meter in ETI August '82 which can be referred to for more details.





For a guide to components and kits for projects, see SHOPAROUND this issue.

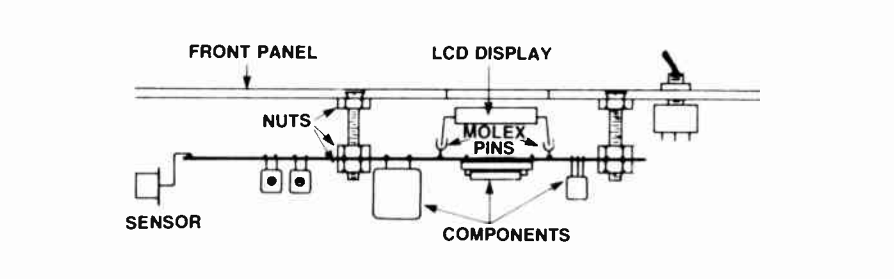
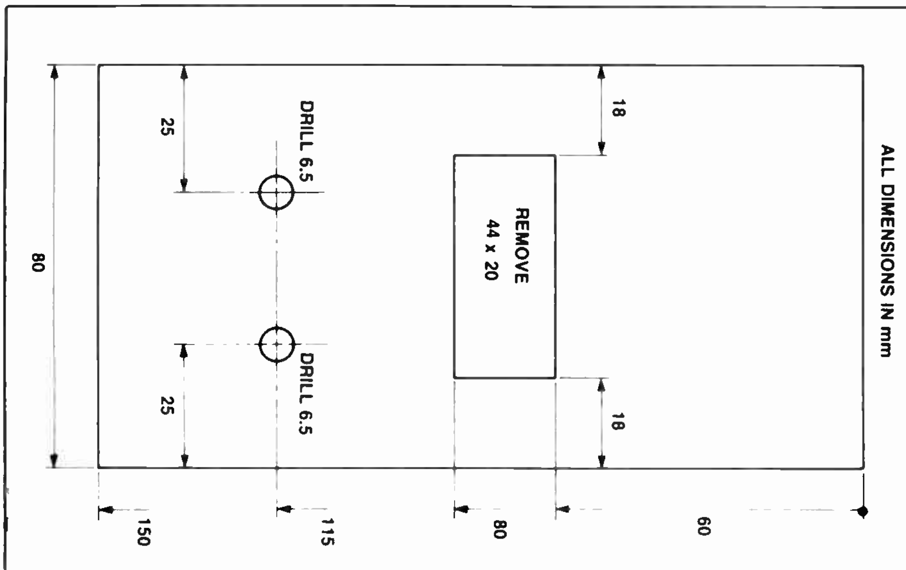


Figure 3. Mounting the pc board.



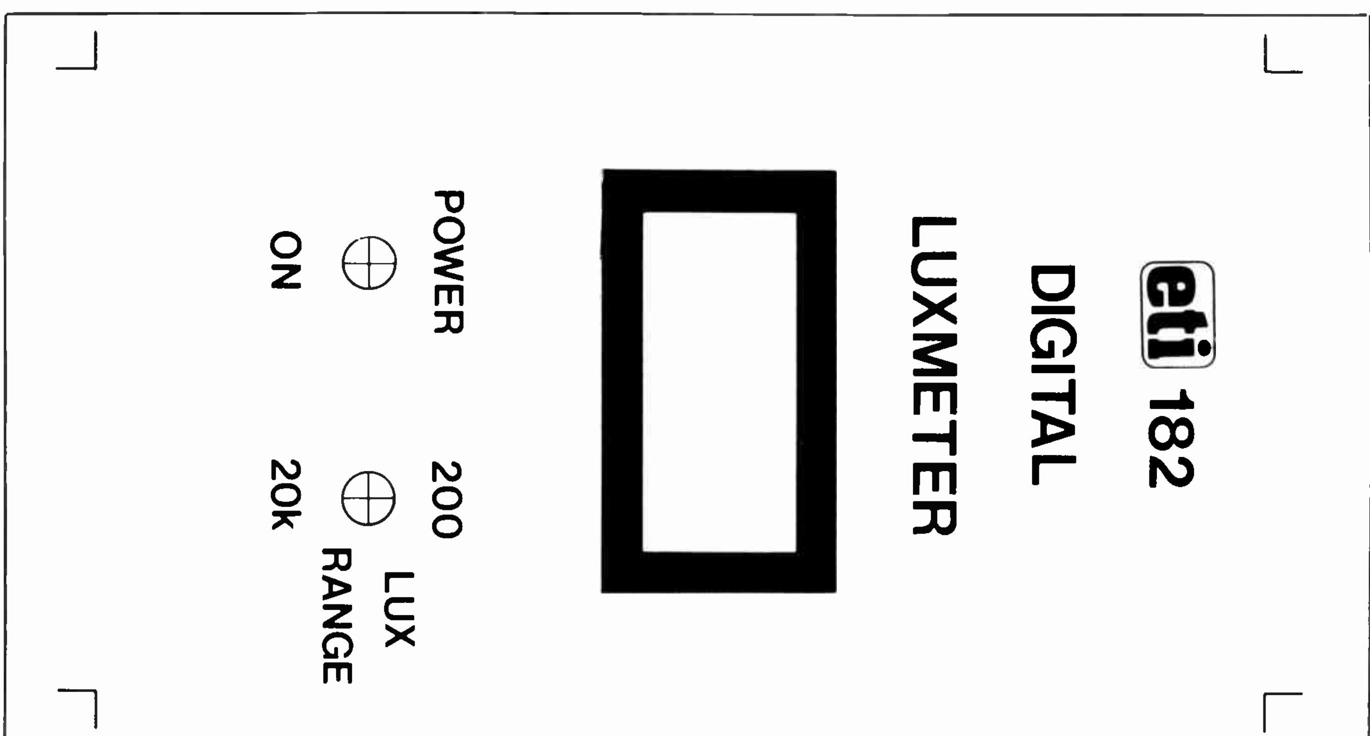
Front panel drilling details

To orientate the display correctly, tip it at an angle and look at the light reflecting off its surface. You should be able to see the shape of the seven segment displays and the decimal points — the decimal points will sit at the bottom when orientated correctly. Plug it carefully into the Molex pins (you may have to spread the legs slightly). Leave the connection of the photodiode till last.

I housed the prototype in a 150 x 80 x 50 mm multi-purpose box but any other type of case can be used (you could try a 130 x 75 x 40 mm Zippy box but things may get a bit tight). Use the front panel artwork as a template and mark out the holes to be drilled. I drilled numerous holes around the rectanglar section and then filed it into shape.

Since the pc board is mounted behind the front panel, sit it in its correct position and mark where the three mounting holes need to be drilled. When drilled, mount the three support bolts into place and countersink the heads. On the prototype, I filled the dimples left by the countersinking operation with Araldite and sanded them flat.

If you intend to use a plastic Scotchcal label, spray the panel with a matt white paint to cover any imperfections which may otherwise show through the thin material. When dry, apply the Scotchcal using the wet technique — soak the label and panel in water and remove the Scotchcal backing sheet. This will allow the label to slide over the panel to be lined up correctly. As long as both surfaces remain wet, you can play around for ages getting the label just right. Then simply wipe the assembly and allow it to dry — hey presto, one perfectly mounted ▶



label. Mounting dry Scotchcal onto dry panels only allows you one chance of getting it right — usually you miss.

Next, drill an 8 mm diameter hole in one end of the case to hold the sensor. If your particular case has internal ribs, remove those in the vicinity of the hole to allow the front window of the sensor to sit flush with the end of the case. Mount the two toggle switches and wire the unit as shown in the overlay diagram. Place another nut on each of the support bolts and fit the board into place. A final nut on each bolt will secure the board (refer to Figure 3).

Setting up

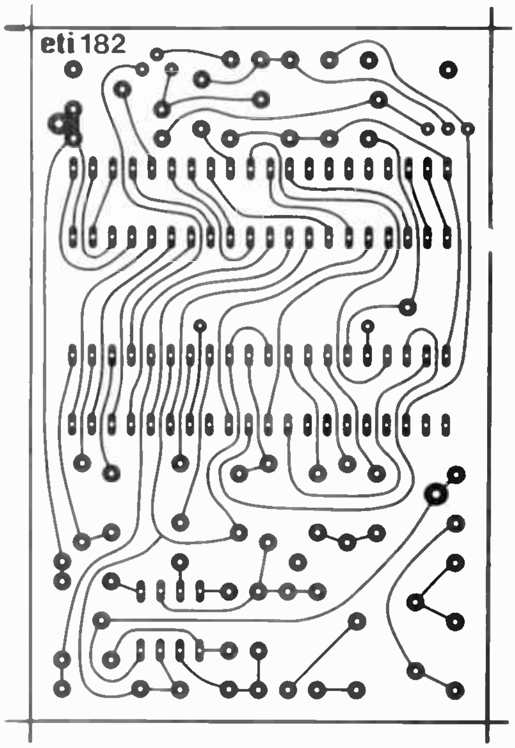
The first adjustment to be made is the offset trimmer RV3. This should be done BEFORE the photodiode is connected. Simply switch the unit on and adjust RV3 until the display shows 0. If this is not possible with your unit or if the reading on the display wanders randomly, check the orientation of semiconductors and all your soldering. It's no use continuing until this offset adjustment can be made.

If all is well, the photodiode can be attached — check orientation carefully. Use some tinned copper wire to extend the leads by about 15 mm. To set the low battery warning trimpot (RV4), run the unit from an adjustable power supply set at 9 volts. Watch the reading carefully while decreasing the voltage. When the reading starts to differ drastically from the original, set RV4 so that the LO BAT indicator just comes on. Do not exceed 9 volts on the supply leads when doing this test.

The final adjustment to be performed involves the calibration of the unit. The most accurate way of doing this is to compare the reading with that of a commercial unit. Set up a fixed light source, such as a light globe, and place the commercial light meter at a position which gives a reading of 100 lux. Place the ETI-182 at this same distance, select the 200 lux scale and adjust RV1 until the display shows 100 lux. Next, repeat the exercise with a reading of 1000 lux (by shifting the meters closer to the globe), use the 20k lux range and adjust RV2.

If you don't have access to a commercial light meter, the following not-quite-so-accurate method can be used for calibration. Set up a 100 watt light globe in an area where there are no reflecting surfaces, brightly coloured walls or mirrors. The illumination level at 300 mm from the globe will be 1000 lux and at 750 mm, 160 lux. Simply place the meter at each of these distances and set the appropriate trimpots.

The unit is now ready for use. ●



SHOPA

THIS PAGE serves as a guide to constructors on where to source components, printed circuit boards and kits for the projects included in this book. While the information included here was checked just prior to publishing, a variety of things may have changed matters by the time you read this. It pays to check with suppliers first, before seeing them or trying to place an order.

Printed circuit boards

There are three suppliers able to provide printed circuit boards for projects included in this book, with certain noted exceptions. They are, in alphabetical order:

Acetronics PCBs
112 Robertson Rd
BASS HILL NSW 2197
(02) 645 1241

All Electronic Components
118-122 Lonsdale St
MELBOURNE Vic 3000
(03) 662 1381

RCS Radio Pty Ltd
651 Forest Rd
BEXLEY NSW 2207
(02) 587 3491

Note that the last two suppliers can also supply front panels and meter scales for many projects.

Now, as for where to chase up kits and components, you should first arm yourself with current catalogues from the major retailers — Altronics, Dick Smith Electronics, Jaycar and Rod Irving Electronics. Where specialised components are required, suppliers are suggested here, but where the exact component may no longer be available, they may be able to suggest a suitable equivalent or replacement. Now read on.

ETI-169 Low Distortion Oscillator

Of necessity, this project makes extensive use of high-spec components. While close tolerance (1% or 2%) metal film resistors are now comparatively common, such things as Bourns and Allen-Bradley pots are not.

To buy components for this project, there are two good specialist stores to try first:

Geoff Wood Electronics
PO Box 671
LANE COVE NSW 2066
(02) 428 4111

Stewart Electronic Components
PO Box 281
OAKLEIGH Vic 3166
(03) 543-3733

They stock suitable high-tolerance resistors and metallised polyester capacitors, ceramic capacitors and suitable pots and trimpots. Note that the case used is a very common item with many electronics retailers.

ETI-170 CRO Calibrator

You should find components for this project relatively easy to obtain as most are standard stock items with most electronics retailers. However, you may have to chase around for the crystals. Note that Rod Irving Electronics stocks a range of quartz crystals.

ETI-172 Bit Pattern Detector

With a modicum of judicious searching through the retailers catalogues and a few well-placed phone calls, intending constructors should be able to source the required components for this useful little project. Rod Irving Electronics stocks this as a kit, cat. no. K41720

ETI-173 ESD Hazard Detector

An unusual project. The specialised components required may be sourced from firms such as Geoff Wood Electronics in Sydney and Stewart Electronic Components in Melbourne; addresses given earlier. Rod Irving Electronics was stocking it as a kit.

ROUND

ETI-174 1 MHz Frequency Standard

To calibrate your other test instruments! This is an ingenious circuit that uses bog-standard components obtainable from almost any electronics retailer worth their salt. The only other thing you need is a TV set . . .

ETI-176 Zener Tester

This project was designed and produced by Dick Smith Electronics and is available as a complete kit, listed in their 1989/90 catalogue as cat. no. K-3051, priced at $39.95. It comes with a pre-punched front panel, making construction that much easier and guaranteeing a more "professional" result.

ETI-179 Analogue Breadboard

Just the thing for dyed-in-the-wool experimenters! All the components for this project are widely available from electronics retailers, including the special "breadboards", so you should have little difficulty getting it all together.

ETI-181 RS232 Breakout Box

Computer enthusiasts should not be without one of these! All Electronic Components in Melbourne (address above) stock this project as a complete kit, so contact them.

ETI-182 Digital Lux Meter

This unusual project depends on the BPW21 photodiode, a Siemens device. Try ordering it through specialist suppliers such as Geoff Wood Electronics in Sydney or Stewart Electronic Components in Melbourne (who may be able to supply a suitable alternative if the Siemens BPW21 is unavailable as they stock a range of opto devices).

The ICL7106 DVM/LCD display driver and 3.5-digit liquid crystal are still widely available and you should find all the other components are pretty well bog-standard.

ETI-183 Op-Amp Tester

A great project for the enthusiast or inveterate experimenter. Components for this project are readily available at electronics retailers; Rod Irving Electronics stock it as a kit, cat. no. K41830.

ETI-185 Versiply

For the experimenter, a very convenient unit to have on your bench always. This project was developed and produced by All Electronic Components in Melbourne who stock it as a kit.

ETI-187 Protopac

A complete kit of parts for this project, as mentioned in the article, is only obtainable through:

Applied Audio Consultants
GPO Box 733
CANBERRA CITY A.C.T. 2601
(062) 43 3345

ETI-190 Digital Transistor Tester

The parts for this project are generally readily available from retailers, with the exception of the 40110 and the case. For these, try Stewart Electronic Components in Melbourne; the 40110 is also stocked by RS Components who have sales offices in each state capital.

ETI-251 Op-amp Power Supply

Another project for the dyed-in-the-wool experimenter or enthusiast, op-amps being so widely used in circuits these days.

Components for this project, as with the majority of others, are widely available from electronics retailers.

ETI-1419 Bench Amplifier

Like most of the other projects, this has been designed with ready component availability in mind, so you should experience few difficulties in sourcing your requirements.