$4.00 *
NZ$5.50

TE COVER PROJECTS

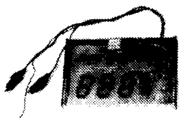
A TALKING ELECTRONICS PUBLICATION

BUILD A KIT!

The cheapest and most convenient way of building any of the projects described in this book is from a KIT. All the problems with the availability of components has been sorted out. You will be surprised at how difficult it is to get some of the chips and most likely you will finish up not being able to buy all the parts from a single supplier. A kit solves this problem and keeps all the parts 'matching' each other. This means all the resistor sizes are the same and the IC sockets come from the same manufacturer. The finished result of a kit looks so superior. Join the many thousands of constructors and send for a kit today. A new constant-heat soldering iron is also available for $19.00 and this will enhance your soldering considerably.

Mini Frequency Counter

A simple frequency counter capable of indicating frequencies up to 5MHz. Ideal for detecting the operating frequency of a digital circuit.



components: $15.60
PC board: $2.70

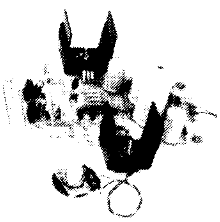
Logic Designer

The Logic Designer contains a number of circuits for testing digital projects. It will also help you with designing your own projects.

components: $17.60
PC board: $3.30

Dual Tracking Power Supply

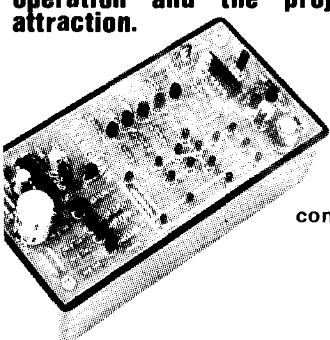
Dual Tracking Power Supply gives a positive and negative rail as required by circuits using op-amps etc. Supplies 7v to 12v (both positive and negative)



components: $15.60
PC board: $3.30

Hangman

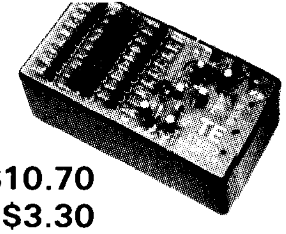
Hangman is an electronic version of Hang The Butcher. It contains a number of digital blocks which are very interesting in operation and the project has great attraction.



components: $9.20
PC board: $3.30

Stereo VU Meter

Stereo VU Meter can be connected to your stereo to give a visual indication of the output of each channel.



components: $10.70
PC board: $3.30

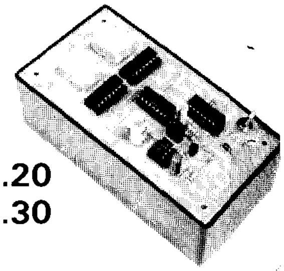
TE Clock

TE Clock runs on the 50Hz mains to keep accurate time. Has an illuminated display for both day and night reading. Ideal for your workshop or room.

components: $19.50
PC board: $3.30

Lotto Selector

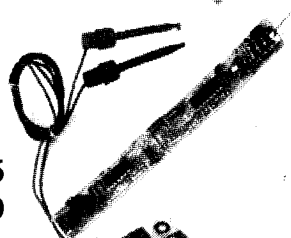
Lotto selector can be used to select your numbers in this week's consultation. It can be adapted for 0 - 40 or 0 - 99 readout. Ideal for war-games or just as a number producer.



components: $13.20
PC board: $3.30

Logic Probe

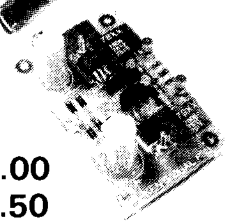
TE Logic Probe is an absolute necessity for trouble-shooting some of our more complex projects. It has a HIGH-PULSE-LOW readout as well as an audible readout via a mini speaker.



components: $8.75
PC board: $2.80

Ken's Dual Power Supply

Ken's Dual Power Supply. Provides a fixed positive and negative voltage for advanced experiments.



components: $8.00
PC board: $3.50

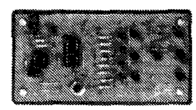
FM Bug

FM Bug. This is one of the simplest and best projects we have produced. Can be hidden anywhere and will transmit up to 100 metres to an FM radio. Will pick up the slightest whisper.

components: $5.00
PC board: $1.80

Led Dice

Led Dice. A very popular project for anyone who likes playing games. The LED readout is similar to the face of a dice (die) and the slow-down effect adds to the realism.



components: $5.75
PC board: $3.30

Note: The price of each project is obtained by adding components AND PC board. These prices are quoted separately as some readers already have the PC board.

Don't forget to order the PC board for each kit.
Post and pack $1.50 for the first kit and $1.00 for each additional kit up to a maximum of $5.00. Small items such as PC boards are 80¢ for the first board and 40¢ for each additional board.

TALKING ELECTRONICS,
35 Rosewarne Ave.,
Cheltenham, 3192.

Tel: 584 2386

TE
COVER
PROJECTS

A collection of our most popular projects appearing in recent issues of TALKING ELECTRONICS.

Including modifications and corrections to various projects.

A TALKING ELECTRONICS PUBLICATION

Introduction

TE cover projects have been an enormous success. The inclusion of a PC board stapled to the cover of the magazine has met with wide approval.

It may not have been a financial success for the magazine but provided a stimulus for readers - which was our intention.

For many, our boards were the first introduction to PC construction. They are certainly one of the best designed boards for the hobbyist and considering that they are mass produced, are of exceptional quality.

And there is another exclusive feature. All the projects can be constrcuted without the need for additional notes or data as the boards contain an overlay. However we have backed up each of the projects with a full description of how they go together and how they work. So anyone and everyone can get the projects to work.

We have used a variety of PC materials from bakelite to fibre-glass and even a double-sided board (Logic Probe) and if you have worked through each of the projects, you will be well acquainted with modern electronic practices.

But this doesn't apply to everybody. Some missed a few issues for one reason or another while others have just noticed our publications on the book-shelves - after 3 years in the field!

Many of these readers are requesting back issues to complete their set and as these are rapidly running out, we have decided to condense all the popular cover projects into a single volume.

Thus the formation of this publication.

If you are lucky, you will find a PC board stapled to the cover and hopefully it will give you the impetus to build the project.

As we are basically an educational organisation, we will continue to provide TECHNICAL SERVICE for all projects we publish.

Our commitment also extends to the marketing of kits for each of the projects and a REPAIR SERVICE for those projects which have been built with our parts.

We receive about 3 projects a week requesting this service and the main problem has been in the poor quality of soldering.

Soldering is a skill which doesn't really take long if you are instructed correctly. That's why you should ask your teacher for some assistance before any bad habbits develop. If you don't attend a class or school, try your local TAFE college for a basic electronics course, they are sure to assist.

All the projects in this book have been built by hundreds of constructors and each is a guaranteed performer.

Schools, groups and colleges are realizing the instructional qualities of our projects and are buying complete class sets. Don't forget to remind your class teacher of the availability of bulk kits and assorted kits so that you will see the whole range of projects built and learn how they work.

Every project uses a different set of building blocks and we have covered over 40 blocks in this book.

Electronics is a matter of understanding simple building blocks and putting these together to form a complex result.

The only way to learn electronics is through practice. And this means construction. We can't expect everone to build every project, but two or three is within the relm of most of us.

After you scale the first hurdle (your first three projects), you will want to get onto more advanced designs. But don't get too carried away yet. Start with one of the simple projects and advance from there.

We have built all the projects in this book and recommend them all.

I hope you like them too.

Colin Mitchell.

INDEX

First Printing 1984

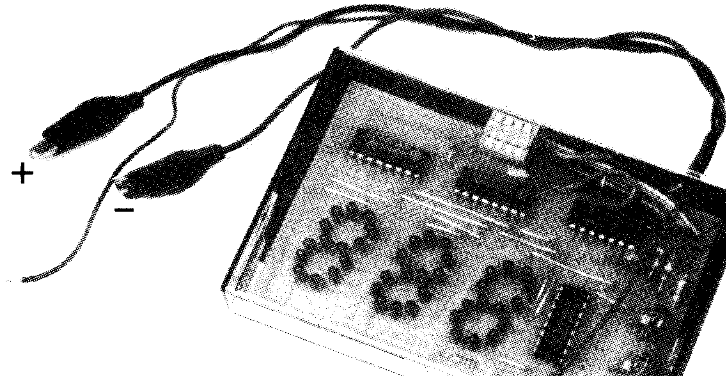
150684 - 035 - 21k

Bulk copies are available at special prices to schools and clubs. Orders can be sent to: TALKING ELECTRONICS, 35 Rosewarne Ave., Cheltenham. 3192.
Tel: (03) 584 2386.

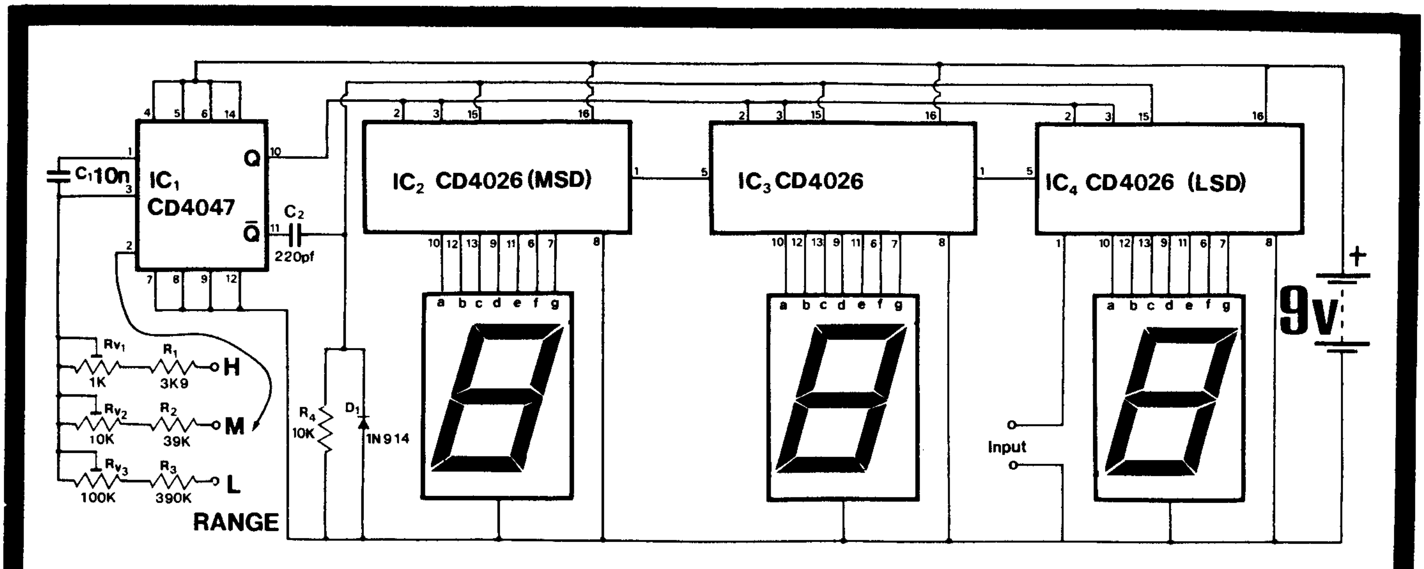
Designed at 35 Rosewarne Ave., Cheltenham, 3192.
Printed in Australia by Standard Printers, Cheltenham, 3192

MINI FREQUENCY COUNTER

A SIMPLE FREQUENCY COUNTER TO DEMONSTRATE THE PRINCIPLE OF COUNTING AND DISPLAYING A FREQUENCY.



Completed counter in a cassette case



MINI FREQUENCY COUNTER CIRCUIT

MSD – most significant digit
LSD – least significant digit

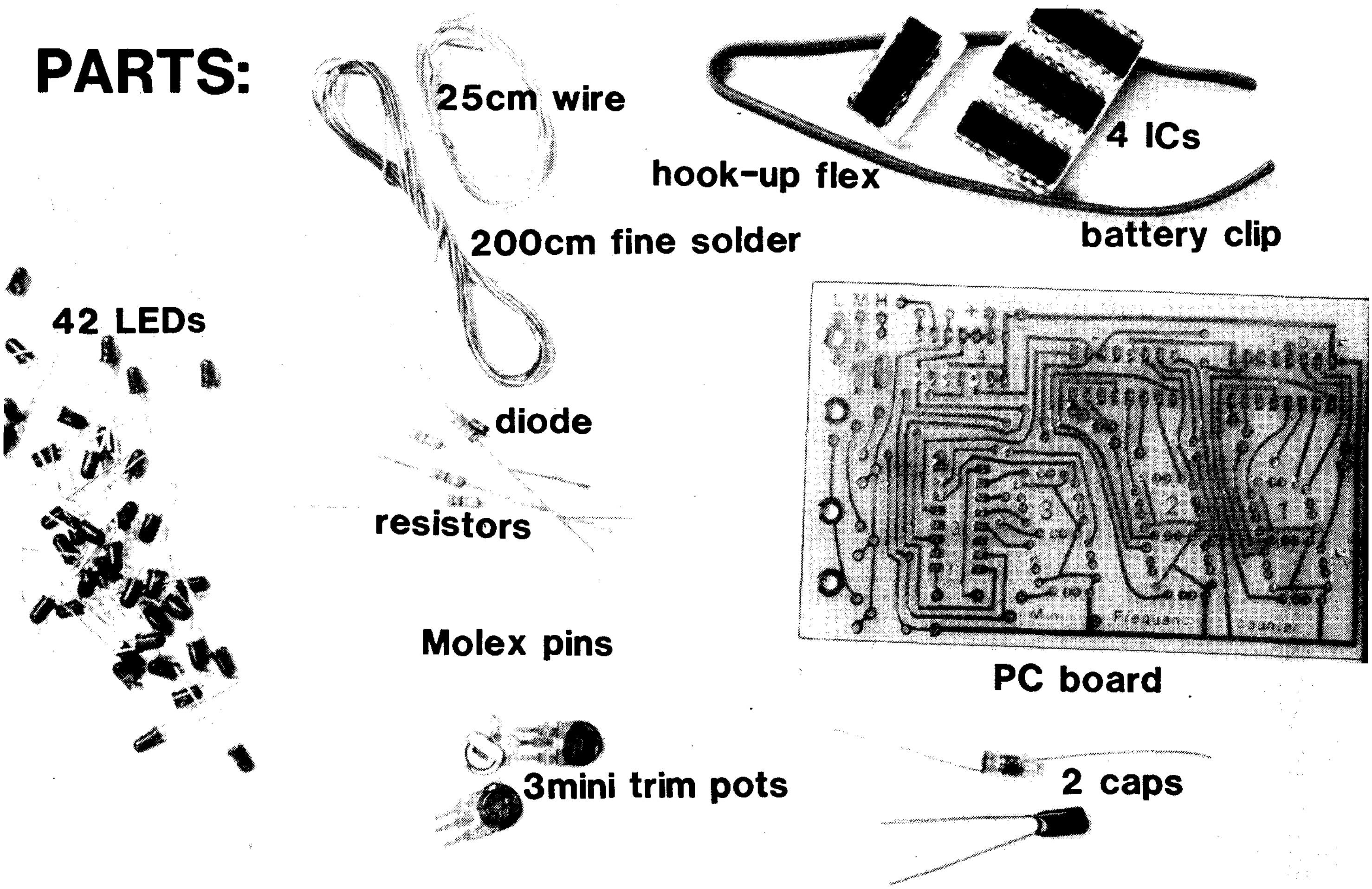
Test equipment is very handy to have but usually costs a fortune. This MINI FREQUENCY COUNTER is an exception. It's not expensive. It will detect frequencies up to 5MHz and can be assembled for about $20 in its basic form. We have presented one of the simplest possible circuits for a counter and kept everything down to a minimum. We have even suggested a 'free' case for the project in the form of a cassette case and a row of Molex pins as the range switch. All the rest of the parts are low-cost standard components including the readout which is made up of numerous miniature LEDs arranged to form three digits. To keep the expenses down we have used a circuit of very clever design. It dispenses with complex counting, holding, latching and multiplexing systems associated with the more complex frequency meters. Yet it reads and performs just like counters costing as much as $100. The absolute simplicity of this circuit is hard to imagine but very easy to see when you look at the circuit diagram.

We have condensed the functions of a frequency meter into 3 basic sections:

A MULTIVIBRATOR	**block 1**
A COUNTING STAGE	**block 2**
A DISPLAY DRIVER	**block 3**

The counter uses only two different types of chips and the three blocks are contained within those two chips. This enables us to create a three-digit readout with three ranges, all contained on a PC board the size of a cassette case. After a little practice you will be capable of reading a frequency to five figures with an accuracy of plus or minus 100Hz. We have also described a number of "add on" circuits to extend the range of the counter. In its basic form the counting IC CD 4026 requires an input amplitude of approximately half rail voltage.

PARTS:



PARTS LIST FOR BASIC COUNTER TO 5MHz

3 - CD 4026
1 - CD 4047

45 - 3mm red LEDs

1 - 10n greencap 100v
1 - 220pf styro

1 - IN 914 or IN 4148 diode

1 - mini trim pot 1k
1 - Mini trim pot 10k
1 - mini trim pot 100k

1 - resistor 3k9 ¼ watt
1 - resistor 10k ¼ watt
1 - resistor 39k ¼ watt
1 - resistor 390k ¼ watt

6 - Molex pins
1 - battery snap
1 - red plastic screen 3cm x 6cm
25cm tinned copper wire
2 metres fine solder

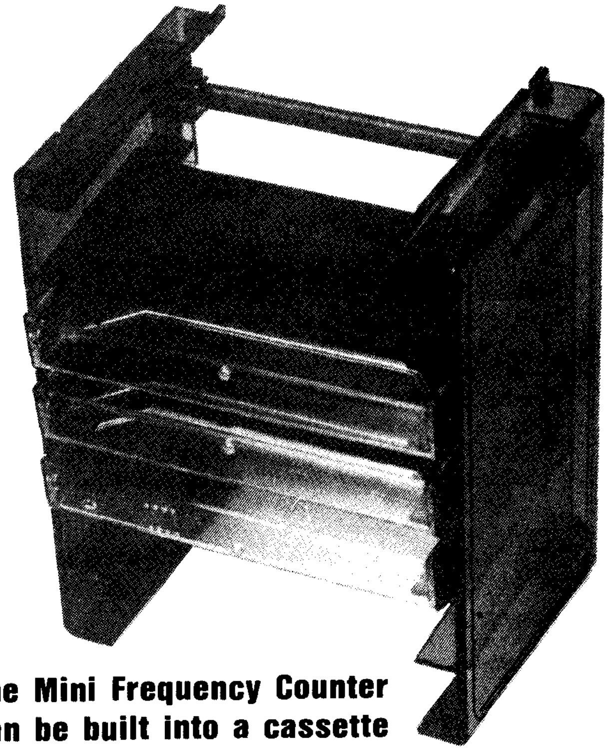
1 - "MINI FREQUENCY COUNTER" PC Board

ADDITIONAL PARTS FOR "ADD-ONS"

This is only a guide to some of the extra parts you may need to calibrate and test your counter. There are **5** methods of calibration. Read the notes first before buying any of the components.

7400
7490
CD 4024
CD 4026
Range change switch
14 Pin IC sockets
16 Pin IC sockets
30 extra 3mm LEDs
BC 547
IN 4001
27.240 MHz crystal
10pf trimmer
100pf capacitor
1n capacitor
100n capacitor
470R resistor (2)
100k resistor
10M resistor
6v battery
hook-up flex
hook-up wire
2 EXPERIMENTER DECK 3 IC's BOARD

Using a fabricated readout in the form of individual LEDs has given us a novel use for a quantity of LEDs and should prove to be an economical alternative to 3 7-segment displays. Since there are so many displays on the market, all of which are generally in short supply, the choice of LEDs makes good sense. Another feature of this project is the PC board. It not only contains the circuit but also serves as the base for the display. When assembled, the project fits neatly into a cassette case. This will be the first of a number of projects designed to fit into a cassette case as they are available in almost every household either full or empty. If you don't have any in perfect mint condition they are readily available at large record outlets. As shown here, they can be kept together in a cassette holder which is also available for under $1.00.



The Mini Frequency Counter can be built into a cassette and mounted in a cassette holder.

DO YOU NEED A FREQUENCY COUNTER?

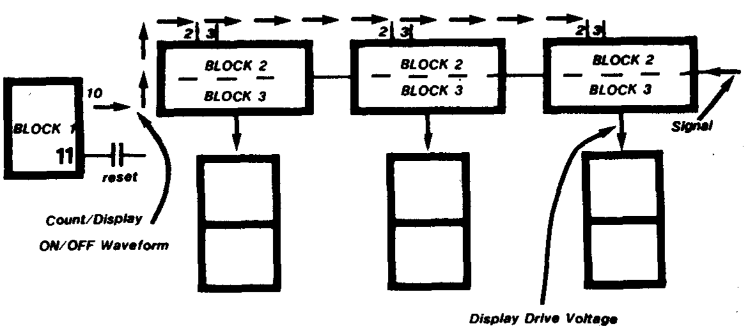
Most hobbyists don't need a frequency counter very often. But when they do, nothing will take its place. This **MINI FREQUENCY COUNTER** is designed to fill that requirement. It is not in competition with the more sophisticated 600 MHz units nor is it intended to be used for accurate alignment work. It is not a highly elaborate device and does not have the accuracy of a crystal controlled unit. But at least it does give you a fairly accurate measurement of a frequency. At some later stage you may be considering building an advanced project requiring an oscillator or clock. With this unit you will be able to set up the equipment and test if it is 'running right'.

☆ **3 Ranges**

☆ **To 5MHz**

☆ **Back-up services**

HOW THE CIRCUIT WORKS



The circuit can be considered as 3 building blocks. The first block, a CD 4047, is a multivibrator running at a frequency set by a **10**n **10,**000pf) capacitor C_1, and a value of resistance determined by the value of one of the trim pots. The three pre-set values of resistance are carefully designed to be decade (divide by 10) values. This means the oscillator will run at a selected frequency on the low range, ten times this frequency on the middle range and one hundred times the frequency on the high range.

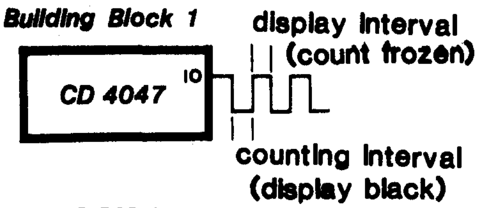
The other two building blocks are contained within each of the CD 4026 IC's. This IC is repeated three times to provide three identical decade dividers, each cascaded together to give a reading on one of the three 7-segment displays.

Cascading simply means the output from one counter is taken to the input of the next counter and this is repeated down the line. Now the signal path between every IC is a **FORWARD** path. In other words there are no feedback loops,

How the circuit works. . .

This means you can follow all the signal paths without any problems. To make the circuit look very similar to the completed project, the incoming signals happen to go from right to left and the blanking/display pulses go from left to right. But these are all still **FORWARD** paths.

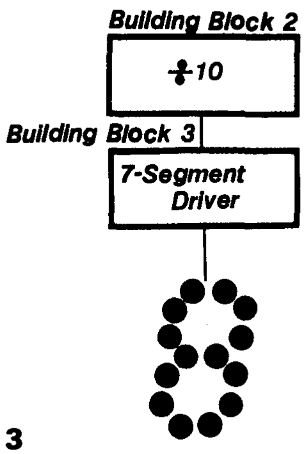
BLOCK 1
The blanking/displaying pulses come from the CD 4047 multivibrator. This IC has two outputs, pins 10 and 11. These outputs are out-of-phase with one-another but this factor is not important in our circuit. One output pin, Q, connects to pins 2 & 3 on each of the CD 4026's and the other, $\overline{Q}$, connects to the reset pin 15 via a 220pf capacitor.



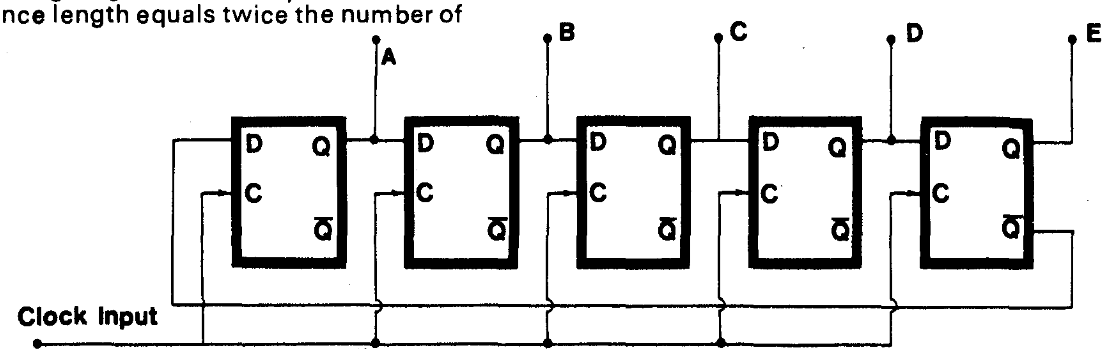
BLOCK 2
Building block 2 is a modulo-ten, five-stage walking ring decade counter and its basic arrangement is shown in the accompanying block diagram. In this type of counter each stage changes only once per every ten clock pulses. This produces no glitches and the counter is fully synchronous. If we follow the basic arrangement we see the output from each stage is connected directly to the Data (D) input of the next stage. D Flip-flops have the characteristic that they store the logic level appearing at the D input until the arrival of the next clock pulse. This means that the flip-flop will remain in its "latched" state for many clock pulses until the level at the D input changes. In our circuit this happens to be 10 clock pulses. Initially the 10-state sequence starts with 00000 by using the reset pin. The output of the last stage is complemented and passed to the first stage where it holds a "1" at the D input ready for the clock pulse. On clocking the five stages once, the only flip-flop to change state will be the first, giving a result of 10000. The next states will be 11000, 11100, 11110, 11111, 01111, 00111, 00011, 00001 and finally back to 00000. There are ten stages in this even-length walking-ring counter and you will notice the sequence length equals twice the number of stages.

Pin 2 on the CD 4026's is the **CLOCK INHIBIT** pin. When this pin is **HIGH**, the IC will be prevented from counting any input pulses. When this pin is **LOW**, the IC will count. Pin 3 is the **DISPLAY ENABLE** pin. A **LOW** on this pin will shut off the display. This is achieved by the IC producing a logic **LOW** on all outputs. A **HIGH** on pin 3 will produce a bright display. Thus by connecting these two pins together, and alternately taking them **HIGH** and **LOW**, we can achieve a display-and-blank from the one clock line. If we consider a single CD 4026 we can see the potential for it to form a counter, albeit only a count up to 10. If you intend to count a frequency, it is important that the count be updated as often as possible. This is achieved by connecting one of the outputs of the multivibrator to the reset pin.

These three functions must now be co-ordinated. Pins 2 and 3 are wired together so that the display-and-blank is made to follow each other and the reset appears immediately after the display interval. With digital IC's there can be virtually no alteration in the frequency over the complete range we are offering. This will produce an accurate frequency meter, provided the first CD 4026 can pick up the input waveform, and produce a clean count.



BLOCK 3
The output from each of the CD 4026 decade counter stages is passed to a set of gates within the chip and they are decoded to produce a 7-segment readout. The **AND** gates are all active **LOW**, with the outputs fully buffered and capable of driving a common-cathode 7 segment display.



BLOCK 2 A"WALKING RING"COUNTER

BEFORE YOU BEGIN ASSEMBLY

Before you commence assembling the parts, make a decision on these three points:

1. Are you going to use sockets?
2. Are you going to make a neat job of the soldering?
3. Are you going to fit the project into a cassette case?

SOCKETS?

I strongly suggest the use of sockets for a project such as this. Whenever you have a particular item repeated more than once, you have a built-in checking system. Having three CD 4026's enables you to do some swapping around if the unit fails to function after assembly. To take advantage of this you will need to provide sockets for the IC's. In any case it is a safe, wise move to consider sockets for all projects. At the low cost of sockets, it removes a lot of worry and frustration if the circuit fails to operate on switch-on.

SOLDERING

If you want to make the underside of the board as presentable as the top, you will need to invest in some fine solder and a small soldering iron. An instant-heat iron will NOT work on this board. It will lift half the lands off the board in the process. Don't be fooled, there is a lot of soldering on the board. The neatness and compactness may be hiding the complexity. We have taken great pains to present a PC board which follows the circuit diagram as closely as possible to produce a large easily-read display at low cost.
You should be fairly experienced with soldering before attempting this project.
Practice on another project - **NOT** this one. Read some notes on soldering if you are not sure and take special note of these 10 points:

1. Buy a low wattage iron - 15 watt to 60 watt
2. Use fine solder
3. Keep iron clean
4. Only solder clean leads
5. Apply a little solder to the iron before every connection. This is called "tinning". It makes soldering faster and easier.
6. Tap excess solder into tray of soldering iron stand.
7. Feed solder onto the joint and not the iron.
8. Keep soldering time short. Use your fingers as a heat-sink and don't overheat the components.
9. Don't carry the solder to the joint 'on-the-iron' as the flux will have gone up in smoke.
10. Don't move the joint until set.

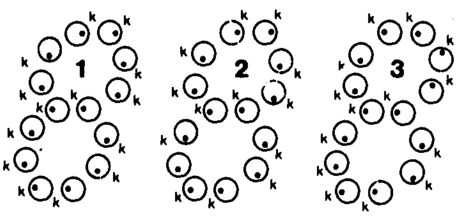
CASSETTE CASE?

If you are going to fit the completed project into the cassette case, you will need to consider the range-switch as it will be difficult to change ranges via Molex pins if the board is in the case. We use a 4 position DIL switch of the rocker-switch variety. You can get two other types but they are harder to turn on and off. Also the size of the mini trim pots will be critical as they just fit into the case when mounted on a slight angle. Otherwise the cassette case produces no further problems.

Once these preliminary requirements are accepted, you can lay the parts out as shown in the photo, ready for ....

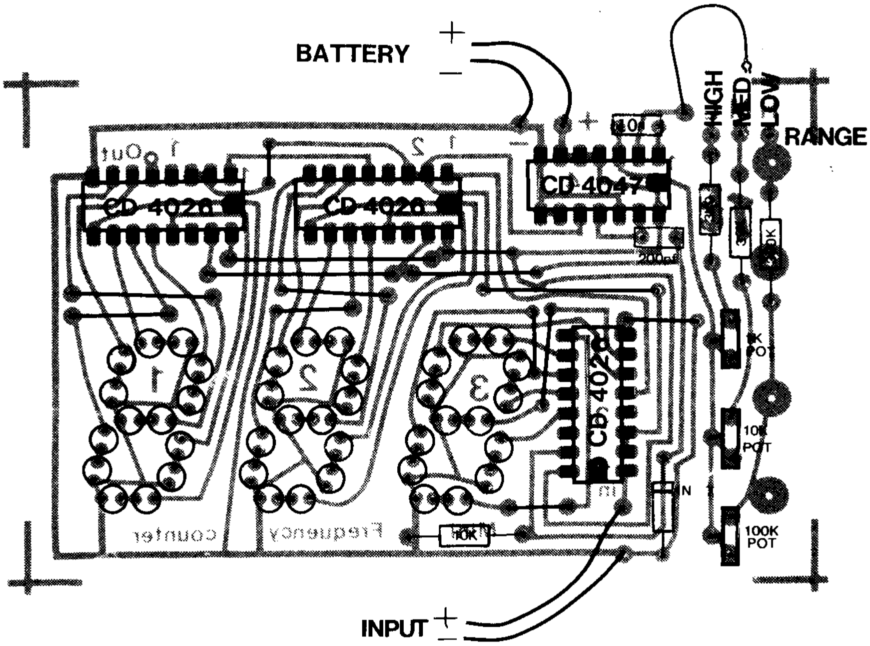
ASSEMBLING THE PARTS

The most time-consuming part will be inserting the 42 LEDs. Refer to the enlarged diagram to identify which way round each LED fits into the board. Insert them one at a time and hold them steady by splaying the leads slightly. Solder the two leads quickly and cleanly, making sure the LED is sitting squarely on the board. Cut each lead close to the board. Remember, these LEDs are quite easily damaged by excessive heat and



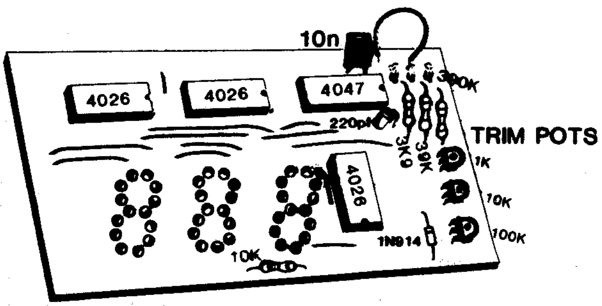
The dot shows the cathode lead

See next page for enlarged diagram



COMPONENT OVERLAY

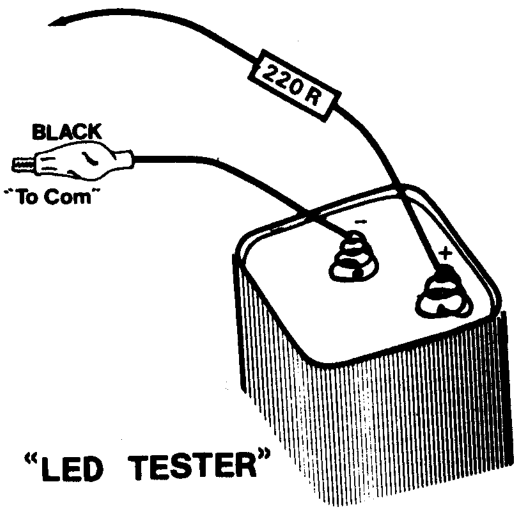
you should hold the top of the LED with your fingers while soldering to act as a heatsink. If you can't hold your fingers on the LED, you know it is taking too long to solder! An easy way to solder these LEDs into circuit is to hang the solder a few cm above the board and move the board and iron onto the solder to make each connection. This eliminates the need for a third hand.
Continue fitting all the 42 LEDs. When they are all mounted, they should be tested.
Use the test circuit suggested. Use a 220 ohm resistor as the dropper if you are using a 6v battery.
Clip the negative lead onto the earth rail which runs around the outside of the printed circuit board and use the free end of the resistor to touch each of the LEDs in turn. If you find a pair of LEDs fail to light, test them individually. You can fairly accurately assume their failure was due to excessive heat. This is why the kit contains 45 LEDs. It allows for 3 mistakes!

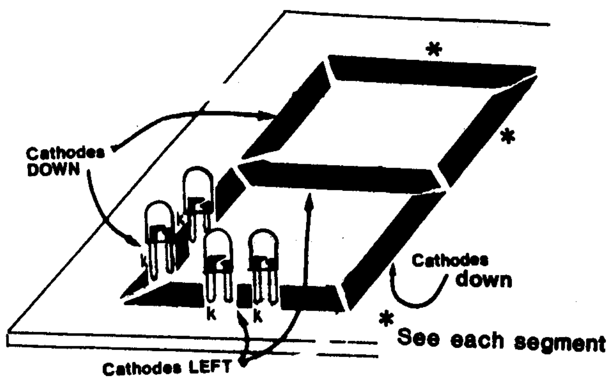


Next fit the IC sockets. This will remove many of the holes and make it easier to see where the remainder of the parts are to be positioned.
Fit the 4 resistors, 3 mini trim pots and the diode. Lastly fit the 10n capacitor and 220pf styroseal. Do not overheat this little capacitor as it melts very easily! If you are going to use the Molex pins for the range switch, these are now fitted along with the jumper lead which should be a length of hook-up wire and not flex as it has to fit into the Molex pins without weakening their grip.

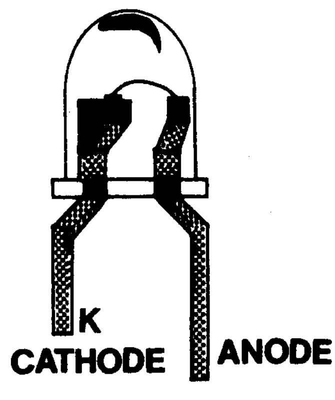
The unit will operate from 6v to 9v and we have included a 9v battery snap in the parts list. Fit the battery clip and insert the 4 chips. Note the position of pin 1 for each of the chips. This is most important. They have only one life and will be ruined if inserted around the wrong way. The 3 CD 4026's do **NOT** run along the top of the PC board. Only 2 of them are above the 3 digit readout. The other top chip near the 10n capacitor is a CD 4047! In addition, the letters 4026 and 4047 may be up-side-down on the chip when it is inserted so don't go by the code numbers. Just go by the identification of pin 1.

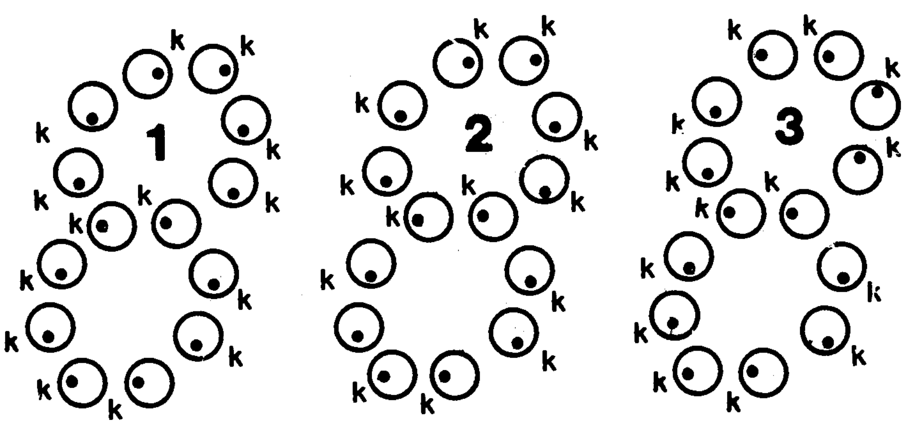
You are now ready to go. Fit the jumper lead into the **LOW** Molex pin and connect the battery. You will see three zero's light up. You are now ready to make some preliminary readings.





Look into each LED:



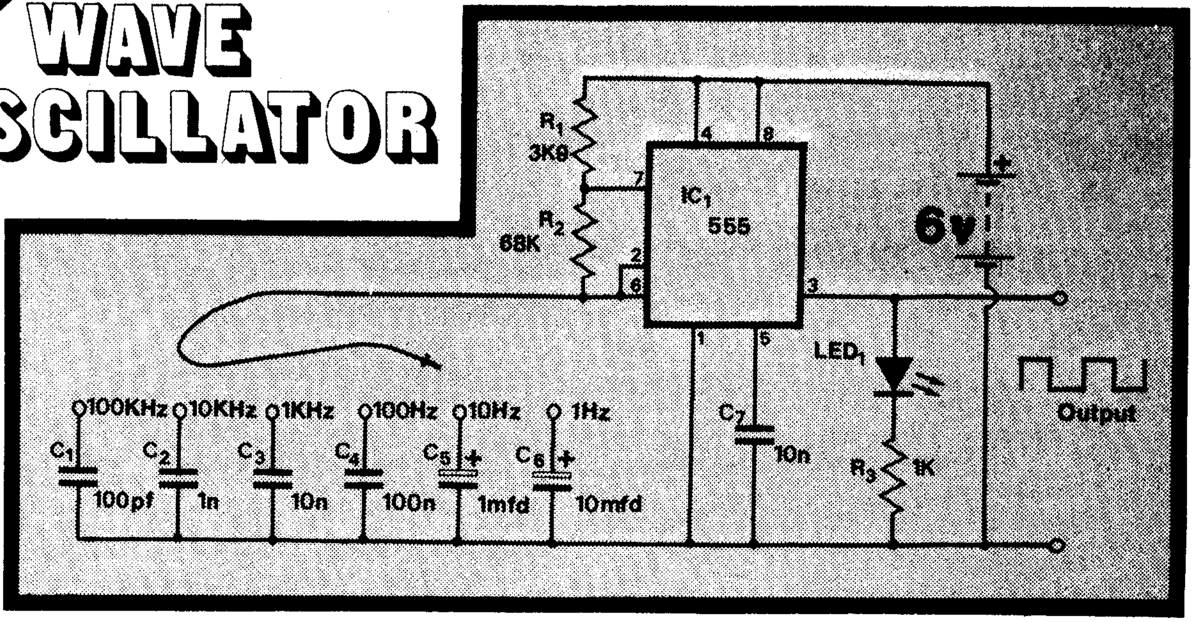


A "CLOSE-UP" SHOWING EACH KATHODE

Test Equipment

SQUARE WAVE OSCILLATOR

A selectable square wave oscillator with 6 ranges suitable for connecting to the Mini Frequency Counter.



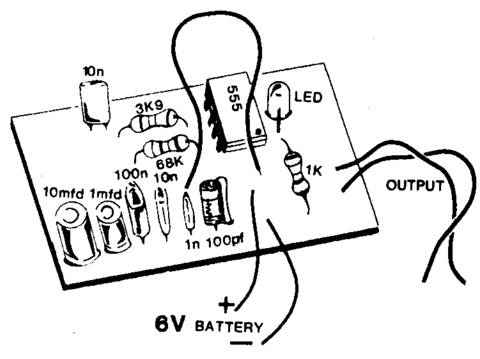
SQUARE-WAVE OSCILLATOR CIRCUIT

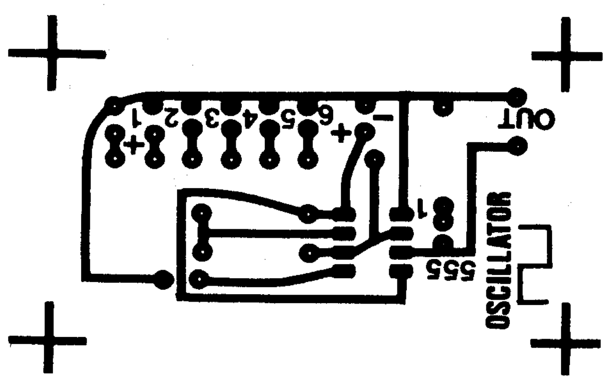
Once you have constructed the **MINI FREQUENCY COUNTER**, the first thing you will want to do is connect it to an oscillator to see if it records a frequency. A simple arrangement is this 555 circuit. It is a repeat of the project from **TALKING ELECTRONICS** issue number 3, page 26. The circuit is a simple **SQUARE WAVE OSCILLATOR** using a single 555 chip. It provides 6-ranges from 1 Hz to 100 kHz and is constructed on its own Printed Circuit Board.

It is essential to use only one 6v lantern battery to power both units so that peak waveforms are compatible.

Assemble the parts on the printed circuit board as shown in the layout diagram and connect the output of the oscillator to the input of the counter. The earth line will provide the other rail.

This oscillator is not accurate enough to become a calibration source. The frequencies 1Hz to 100kHz listed on the circuit are only a guide and are very approximate. Once you are satisfied the counter is accepting a frequency and giving a readout, it is calibrated via an accurate oscillator. It can then be brought back to this Square Wave Oscillator to determine the actual output frequencies of each range.





READING THE COUNTER

You may be wondering how we can read frequencies in the millions with a counter having only 3 digits. This is where your skill and interpretation comes in. The first thing we must point out is the counter cannot read any frequency down to the last two digits. For instance, a frequency such as 12,586 Hz would be read as 12,500 Hz or 12,600 Hz. The lowest frequency we can detec is 100 Hz and the highest 5 MHz or so. Because it can't display the last two digits, you will have to visualise two zeros on the end of the frequency when in the **LOW** range, three zeros in the **MIDDLE** range and four zeros in the **HIGH** range. This is not as awkward as it might first appear. The small 'cut-out' card on the data page will give you future reference.

A frequency such as 126,000 Hz will read as follows:

LOW 260
MED 126
HIGH 012

$\boxed{2}\boxed{6}\boxed{0}$ 00 **Hz**

$\boxed{1}\boxed{2}\boxed{6}$,000 **Hz**

$\boxed{0}\boxed{1}\boxed{2}$ 0,000 **Hz**

By changing ranges you will be able to collect information on the incoming frequency and by combining these three readings see the value to be 126,000 Hz.

Take another example:

27.240 MHz:

This frequency will be too high for the counter and the CD 4026 will not respond.

Only three zeros will be displayed on the output.

Example 3:

2.7240 MHz (This is the frequency after passing through the decade counter.)

LOW 240
MED 724
HIGH 272

$\boxed{2}\boxed{4}\boxed{0}$ 0 0 **Hz**

$\boxed{7}\boxed{2}\boxed{4}$,000 **Hz**

$\boxed{2}\boxed{7}\boxed{2}$ 0,000 **Hz**

Combining these three readings we obtain a frequency of 2.724,000 Hz.

Example 4:

54,675 Hz

LOW 546
MED 054
HIGH 005

$\boxed{5}\boxed{4}\boxed{6}$ 0 0 **Hz**

$\boxed{0}\boxed{5}\boxed{4}$, 0 0 0 **Hz**

$\boxed{0}\boxed{0}\boxed{5}$ 0 ,000 **Hz**

Putting these three readings together we obtain 54,675 Hz.

CALIBRATION

Now comes the most important and critical operation. As with all test equipment, its accuracy is only as good as the setting up technique and the reference source. If you want the counter to read accurately in the field, you must use the most accurate source available for setting up.

There are 5 methods for calibrating:

1. Using a frequency generator.
2. Using a crystal locked oscillator,
3. Using an oscillator connected to a frequency divider,
4. Using a CRO,
5. Using another frequency counter.

These 5 methods have varying degrees of accuracy. They vary from extremely accurate to only about 10% accurate. The final choice will depend on the amount you have to spend on calibration and the availability of a reference source.

Obviously the ideal situation would be to borrow a frequency generator for an evening and calibrate the unit against it. This is how we calibrated our prototype unit. Once it is on frequency, the pots are sealed and must not be touched for the life of the unit. Being digital, it will give a linear accuracy up to 5 MHz or more. Obtaining the use of a frequency generator is not beyond the possibilities of members of a club or school as many clubs have this sort of equipment and it would take only a few minutes or so to set the trim pots. Some clubs even loan out this type of equipment for a night or week-end so ask around and see if this facility is available to you. If it is not, you have the alternative of building your own crystal oscillator. Our circuit uses readily available parts and will double up as the prescaler to convert the counter to a 50MHz counter.

1. USING A FREQUENCY GENERATOR

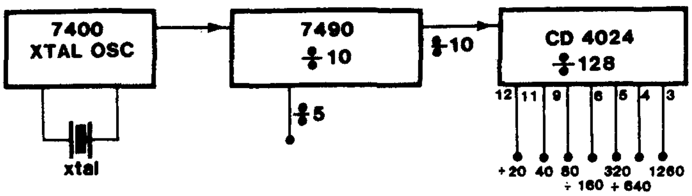
If you have access to a frequency generator it will be possible to set up the three ranges to an accuracy of plus or minus 2 counts of the third digit. When selecting a frequency on the generator it is not wise to choose decade values such as 1kHz or 1MHz as these are difficult to interpret when you are changing ranges. Select an odd frequency such as 123kHz or 4.7386MHz and see how closely you can align each of the three ranges. On the low frequency range the stability of the unit has a little to be desired as the readout tends to flicker quite considerably on the second and third digits. This is generally due to the incorrect frequency being displayed on the low range. Under correct conditions, the display will respond satisfactorily. Once you are satisfied that the readings are in line with the applied frequency, place a little silicone sealant on the rotating part of the trim pot near the base. Do not glue the spot which touches the carbon track as the glue may run between the wiper and track to produce an intermittent contact.

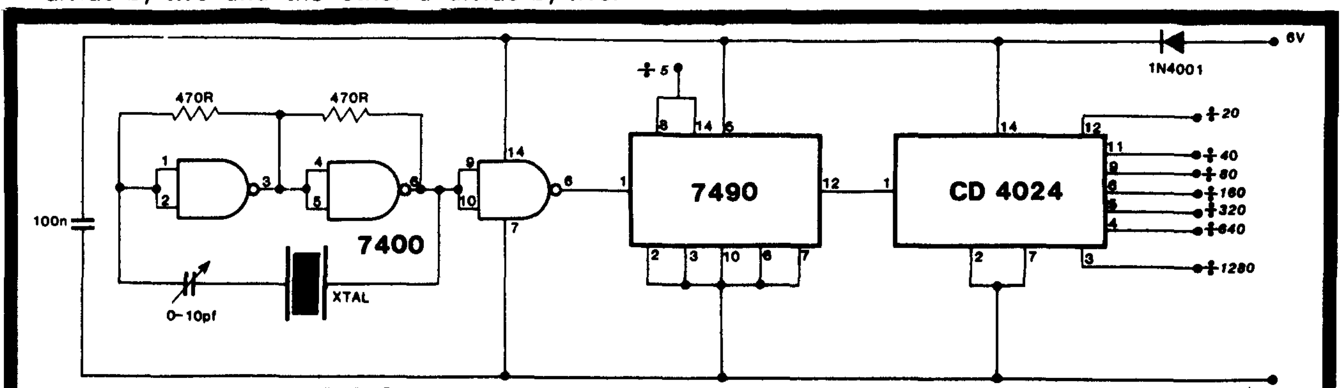
2. CRYSTAL OSCILLATOR

These "add-ons" could be a project all on their own. Basically this circuit consists of 3 building blocks as shown. It can be used at a later stage as a crystal tester and is capable of identifying crystals of any unknown frequency. The first block is TTL crystal oscillator which will accept any crystal up to 50MHz and enable it to oscillate without loading it. The circuit is quite reliable in starting and is trimmed via a piece of ribbon cable acting as a small series capacitor. For a 27.240 MHz crystal the twin wire is 14cm long. The wire can be replaced with a 0-10pf trimmer if desired. The output from the oscillator is fed to a 7490 Decade Counter. Again, this is TTL device as we are still operating at frequencies well above the capabilities of CMOS. The output from the 7490 is in the range 1 MHz to 5 MHz depending on the crystal fitted, and this frequency is suitable for a CMOS device. The CD 2024 is a binary ripple counter having 7 stages of division. This will divide the output of the crystal by two at each stage. It is quite an interesting exercise to tap off each output on the 4024 and record the half, quarter and eighth frequency etc. In our prototype we used a 27.240 MHz crystal. It is one of the cheapest crystals available and can even be purchased from most of the surplus stores. It does not have to be this specific frequency as any crystal can be fitted and will work perfectly. The 7490 employs two separate stages of division inside the chip. One stage is a divide-by-two and the other a divide-by-five. These must be connected together externally to obtain the divide-by-ten capability. The output of this decade divider is connected to a CD 4024. This chip will divide the frequency by 128, making a total division of 1280.

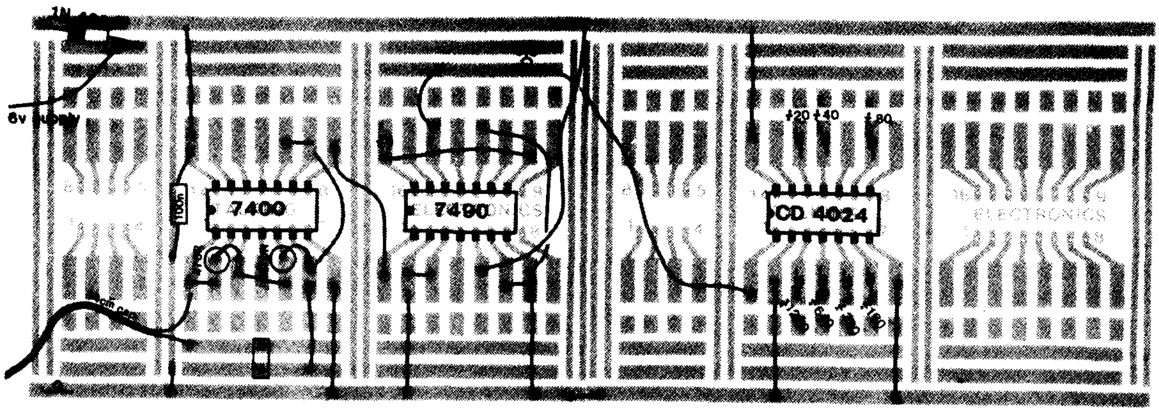
The 27.240 MHz crystal was trimmed to the exact frequency and the results from each of the outputs were as follows:

Ref.			
	Pin 8	7400	27,240,000 Hz
÷ 5	Pin 8	7490	544,802 Hz
÷ 10	Pin 12	7490	2,724,000 Hz
÷ 20	Pin 12	4024	1,362,000 Hz
÷ 40	Pin 11	4024	681,000 Hz
÷ 80	Pin 9	4024	340,500 Hz
÷ 160	Pin 6	4024	170,250 Hz
÷ 320	Pin 5	4024	85,125 Hz
÷ 640	Pin 4	4024	42,563 Hz
÷ 1280	Pin 3	4024	21,281 Hz





XTAL OSCILLATOR AND FREQUENCY DIVIDER



3. USING AN OSCILLATOR CONNECTED TO A FREQUENCY DIVIDER

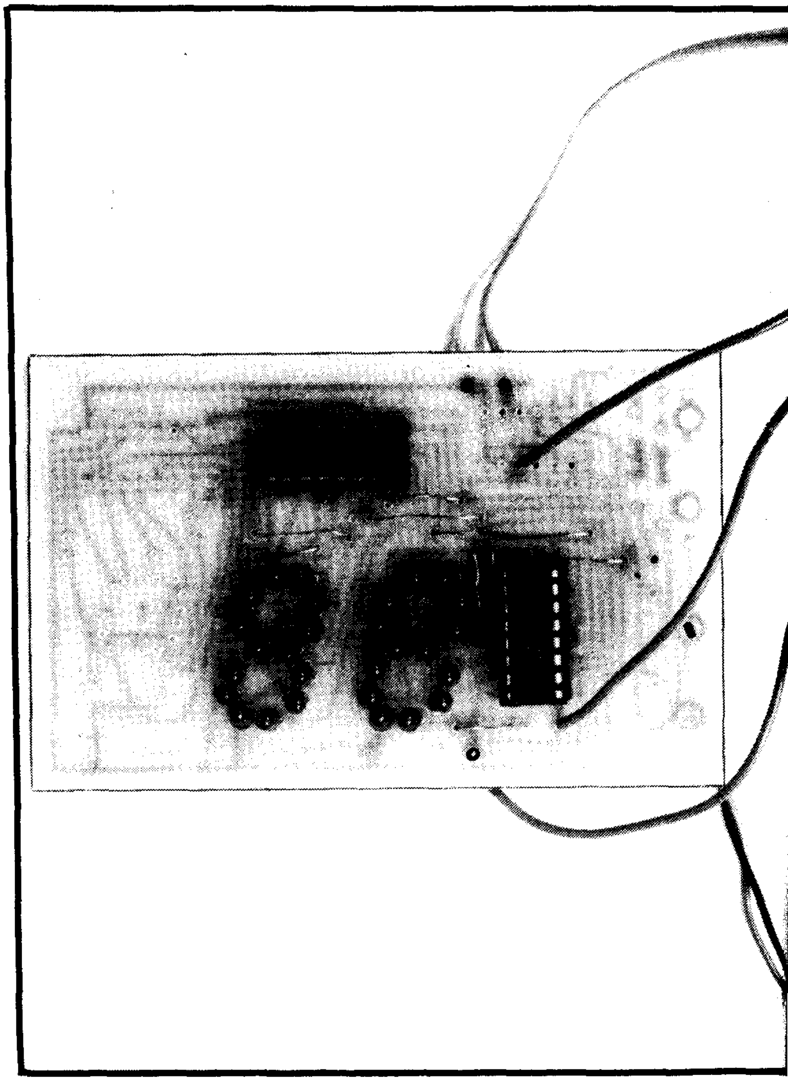
This is the most inaccurate of the 5 methods and is only to be used as a last resort if you are unsuccessful in obtaining a crystal. The circuit arrangement for this method of calibration is to set up the Square Wave Oscillator and feed it into a divide-by-128 counter. The output of the counter drives a LED which will flash on and off at regular intervals. Providing the flash rate is not too high, it will be necessary to count these flashes during a one minute interval and thus determine the frequency of the oscillator. The only drawback with this arrangement is the maximum frequency we can obtain with the 555 and the number of flashes we can reliably count per second. In all probability you will need to go to a 4040 or a 4020 where the number of stages is dramatically increased so that the oscillator can function near its upper limit. In any case this would be able to give the counter a reading on its low range and if you were to scale up the reading to the other 2 positions, the final accuracy would be very poor indeed.

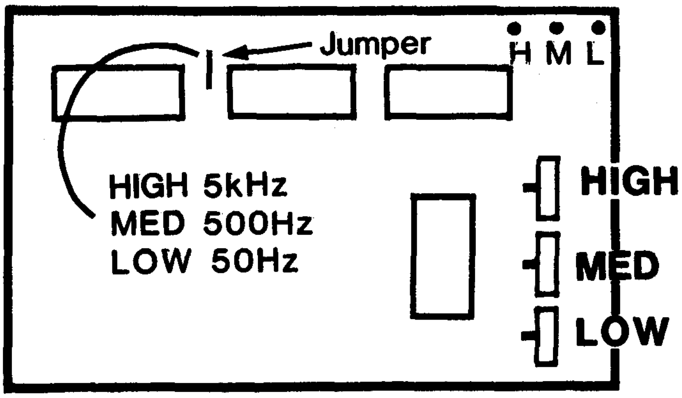
4. USING A CRO

With most CRO's it is surprisingly difficult to accurately determine a frequency on the screen. With just a CRO at your disposal, it is only possible to set up the internal multivibrator frequency. This can be detected at the jumper between IC's 1 & 2. The reading must be interpreted very accurately as these form the reference frequency for the 3 ranges. The **LOW** frequency is 50Hz, the **MIDDLE** range is 500Hz and the **HIGH** range is 5,000Hz.

USING ANOTHER FREQUENCY COUNTER

With a frequency counter we can only set the internal multivibrator frequency. Locate the jumper between IC 1 & 2 and attach the frequency counter. Set the **HIGH** range to 5 kHz, the **MEDIUM** range to 500 Hz and the **LOW** range to 50 Hz. These are the frequencies you will read on your 'test counter' when the Mini Frequency Counter is switched on but not connected to any incoming frequency. It will be reading 000



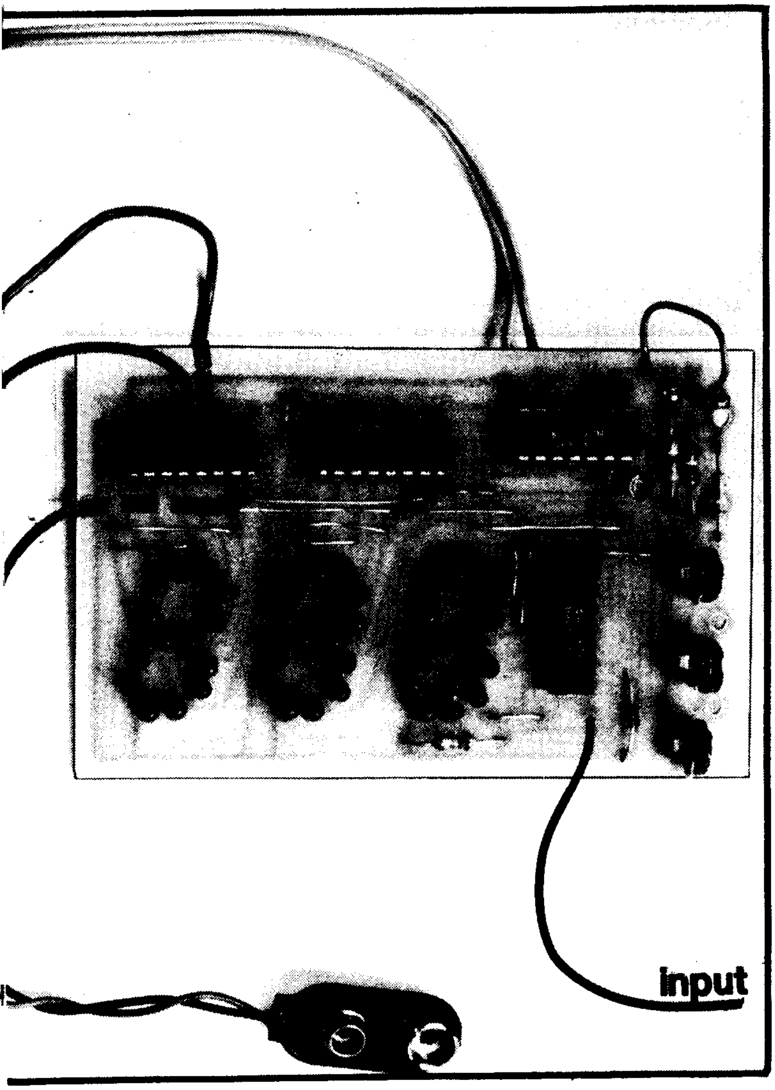


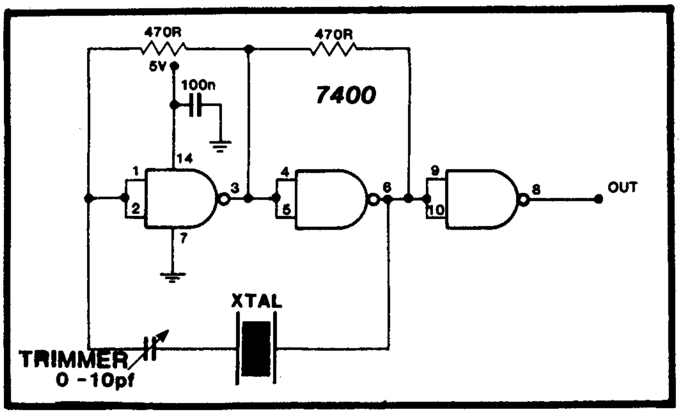
DISPLAY/BLANKING PULSE

during this operation. Unfortunately this set-up is only giving us a 2% accuracy on the **LOW** range and we would like the counter to be better than that. To improve the **LOW** range (especially), the answer is to use both......

A FREQUENCY COUNTER AND AN OSCILLATOR

If you can also borrow a frequency counter you will be able to achieve better accuracy by combining it with an oscillator. Even the Square Wave Oscillator will be suitable. After making preliminary settings on the trim pots via the previous method, connect both counters to the output of the Square Wave and select the 100 kHz frequency. Adjust the **MEDIUM** range trim pot until both counters are reading the same frequency. Change the range switch to the **HIGH** reading and trim this range. Repeat this for the **LOW** range. When you are satisfied with the readings, apply a little silicone sealant to the wiper of each pot. Make sure it doesn't get in the way of the pick-off point on the carbon track.





CASCADING

Two units can be wired together quite easily to give a 5 digit readout as shown in the photo. The technical department rushed together the additional board with the two digits, to show the number of components required, the jumpers needed, and the set of connecting wires required. And it looks very simple. This arrangement does not improve the capabilities of the counter in any way nor give it any additional accuracy. It merely saves moving the range switch from one range to another to determine the numbers which will be on either side of the three digits being displayed. The battery snap is designed for 9v but a 6v battery made up of 4 penlite cells can be used and will give good results. The top connecting leads in the photo are the positive and negative supply rails. The next lead from the top is the display/blanking line. The next lead down takes the carry-out from the "hundreds" counter to the "in" of the next IC. The bottom lead connects the reset line to the cascade board. The actual point at which these leads solder to the board is marked on the photo with a dot. This will make it easy for you to duplicate this set-up.

INPUT SENSITIVITY

As a basic unit, the **MINI FREQUENCY COUNTER** has an input sensitivity which will only respond to inputs greater than half rail voltage. This means it requires at least 4v P-P input.

IN CONCLUSION

We hope these detailed plans have covered every aspect of the construction so that your counter will be a success. If you experience any difficulty in building the unit, try asking your friends for assistance. The circuit is simple enough, and the operation is fairly straightforward. Go over the instructions again and again in case you missed an important point. If all this fails, the unit can be sent to the Author for diagnosis and repair. This is a back-up service to maintain faith with our readers, especially the younger ones, who may give up on electronics if they have too many failures. Obviously the repair department will not go into re-designing the counter or work on any units which have been constructed on boards other than those supplied with this magazine. They have a jig in which they fit the board and a set of test prods to located the fault in a matter of moments. Wholesale damage to the board will not be repaired nor will extremely bad soldering. In fact the limit will be one IC and up to 5 LEDs. Beyond this the cost of repair is getting close to the cost of the project. Most of the units posted to the address on the cover of the magazine will require simple alignment and this will be done via a set of predetermined frequencies for each range. The pots will then be sealed and they should not be touched for the life of the counter. We hope you will not require either of these services as we have included sufficient information in this article for you to be able to carry out your own alignment.

Before we conclude, and wish you every success with the project, may I request a little feedback from you?

Whether it be good, fair or indifferent, how about letting us know how you like the idea.

All the best

Colin Mitchell.

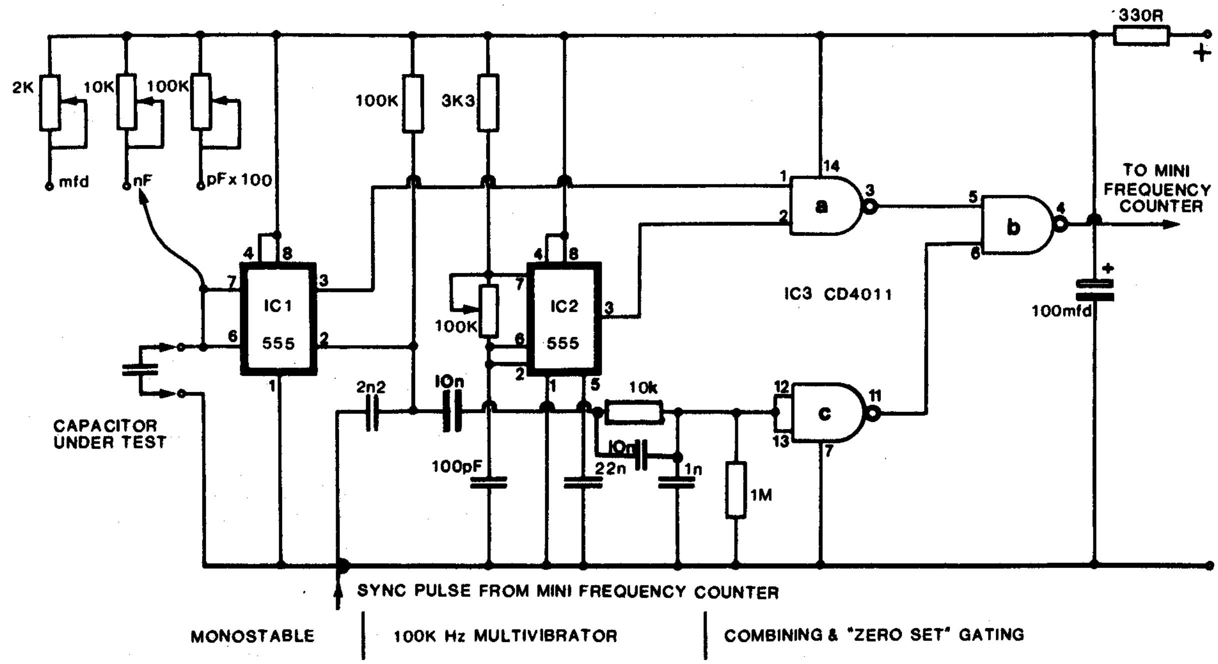
TEST EQUIPMENT

☆ Add-on to our Digital Frequency Counter

☆ Reads 100pf to 10mfd.

DIGITAL CAPACITANCE METER

For about $8 you can extend the value of the Mini Frequency Counter to test capacitors.



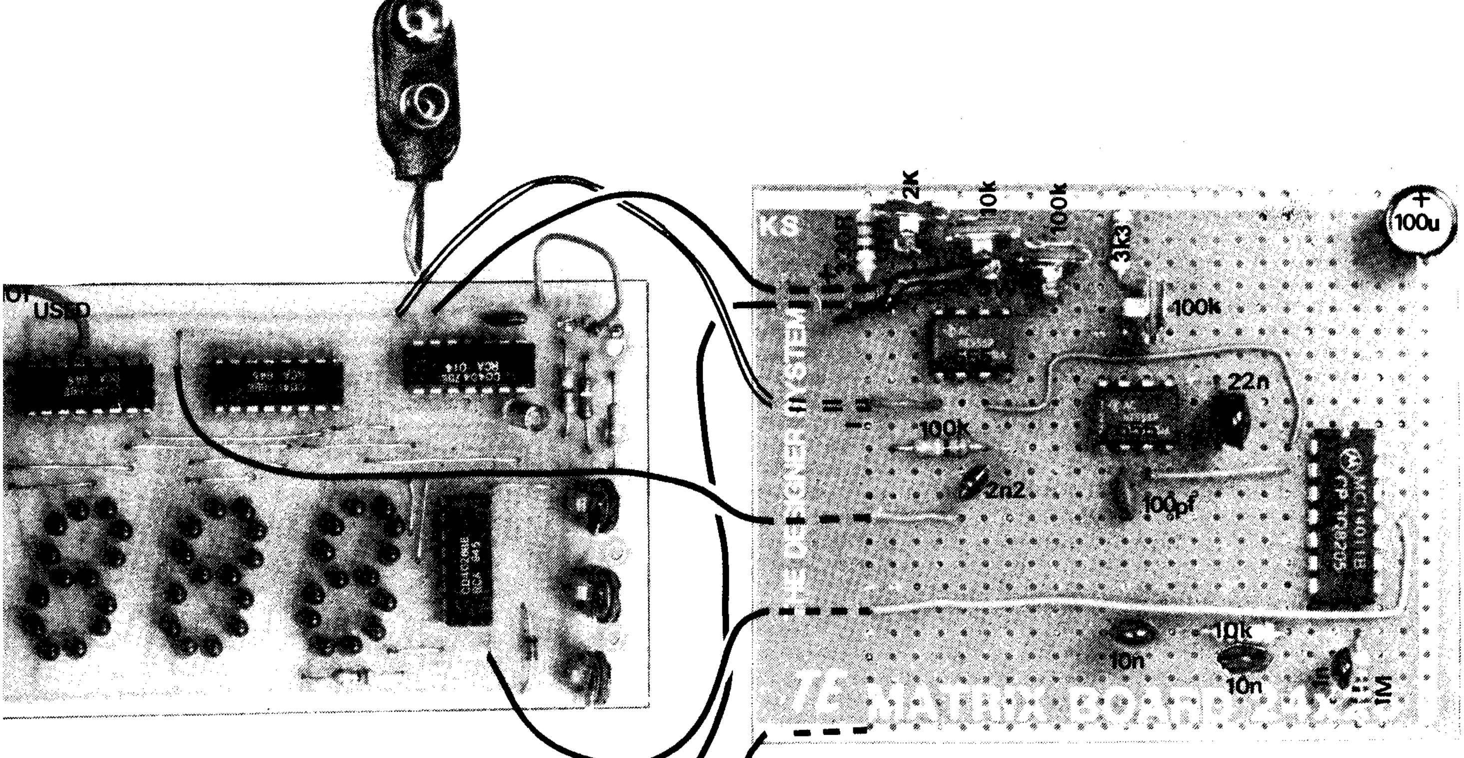
Now you can double the value of the Mini Frequency Counter by adding this DIGITAL CAPACITANCE METER to the front end. By simply attaching this pulse circuit to the input of the counter, you can measure capacitors from 100pf to about 10mfd. This is very handy as a double-check for those values which are hard to decipher or where the value has rubbed off.

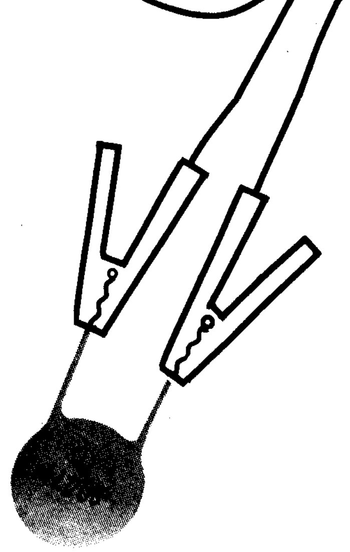
Once the unit is set up accurately, you will be quite surprised how much capacitors vary in value, especially electrolytics. When a manufacturer says his electrolytics can vary by as much as +80% −20%, this variation really comes home when the values are read. Two electrolytics from the same batch can vary by as much as 50%.

HOW THE CIRCUIT WORKS

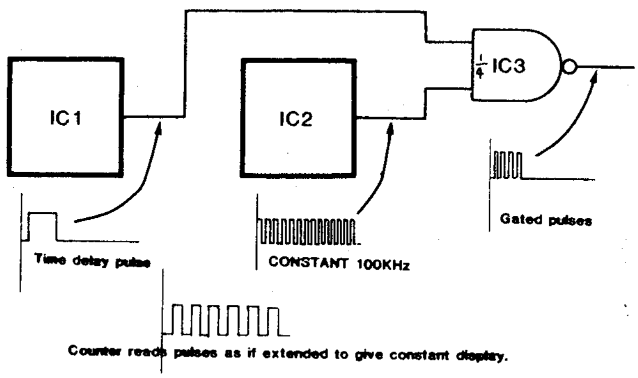
IC1 is a 555 timer arranged as a "one-shot". This is a monostable multivibrator in which the unknown capacitor provides the time delay. When the capacitor is included in the circuit, the 555 is triggered from the count-display pulse of the mini frequency counter. This keeps the two units synchronised so that the display is steady. The output of the first 555 is initially LOW and goes HIGH for a period according to the time delay provided by the value of the unknown capacitor. This is fed to pin 2 of the NAND gate contained in IC3.

The second 555 is a free running oscillator set at 100kHz. The output pin 3 constantly feeds the





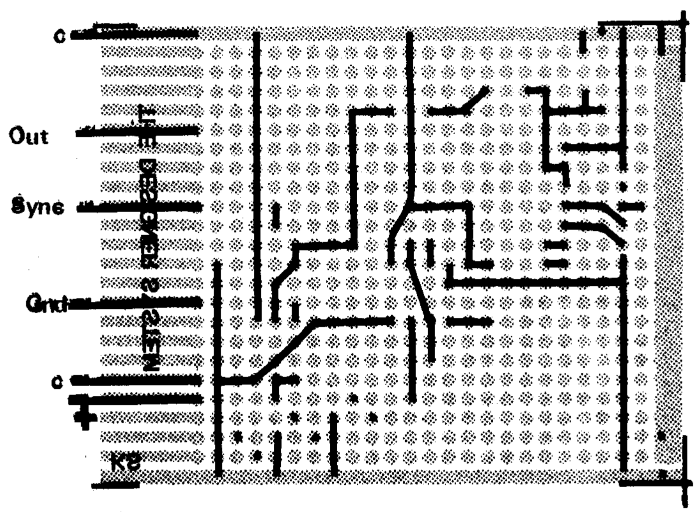
1 - 330R
1 - 3k3
1 - 10k
1 - 100k
1 - 1M
1 - 100pf styro
1 - 1n 100v
1 - 2n2
2 - 10n
1 - 22n
1 - 100mfd 16v
1 - 2k mini trim pot
1 - 10k mini trim pot
2 - 100k mini trim pot
2 - 555 timer IC
1 - CD 4011 IC
hook-up wire



NAND gate and becomes our reference source. The output pin 3 of the NAND gate could be fed into the mini frequency counter for quite a successful reading except for one small problem.

The gated 555 (IC1) is triggered into timing from pin 2. If no unknown capacitor is present at the input, the sync pulse still trips the IC and the small internal capacitance of the IC causes it to perform a short cycle. In fact a very short cycle. This would open up gate 'a' of the CD 4011 for a short time and allow a few cycles of the reference frequency to pass through. Thus we could not obtain a zero-set on the frequency counter. To overcome this problem we have gated the sync pulses out of the transmission time via gates 'b' and 'c' to obtain a zero set.

It works like this: gate 'b' will pass the burst coming from pin 3 except when the sync pulse is detected at pins 12 and 13 of gate 'c'. It is delayed and extended by the integrating network made up of the 1n and 1M components. The



result is 3 zero's on display when no capacitor is being tested.

To explain how the Mini Frequency Counter converts the pulses to a reading, we will take two examples:

Firstly we will exaggerate the count-display interval of time to make it easy to describe. Suppose the counter had a count period of one-half second and a display time of half-second. This means pulses present for the first half second will be displayed but those appearing for the remaining half second will not be counted. Now our capacitance meter sends out bursts of pulses and to prevent them overshooting into the display time, we provide a sync pulse. This means our circuit will only send pulses when the counter is ready to accept them.

Let us compare the different effects of a 1n capacitor with a .47mfd capacitor. When the 1n capacitor is being tested, the sync pulse turns on the first 555 and its output goes HIGH for a period of time to let 100 cycles of the 100kHz frequency through gate 'a' of the CD4011. When the .47mfd capacitor is tested, the monostable time is lengthened considerably and 47,000 cycles are gated through the NAND gate. This gives the display reading of 470.

You can see that accurate timing is required and this is entirely possible with digital circuits and accurate timers such as the 555, having a linear response to RC values.

CONSTRUCTION

Our model was assembled on a DESIGNER BOARD 24 x 25 and this is a very suitable medium for producing a neat result at the least expence.

The PC wiring is created with fine tinned copper wire after all the components have been soldered in place. But before any of the components are added to the board, the underside must be marked with a spirit pen to indicate where the underside wiring will be positioned. This will enable you to mount the components in the correct places and provides easy referencing.

Start construction by mounting the IC's. This is not the normal practice but provides a reference for the other components. You can use sockets for the IC's or mount the chips directly.

When all the parts have been fitted, use fine tinned copper wire to complete the underside "wiring".

Four lengths of hook-up flex connect the capacitance meter board to the Mini Frequency Counter. And two lengths of flex are connected to alligator clips for the CAPACITOR-UNDER-TEST.

SETTING UP

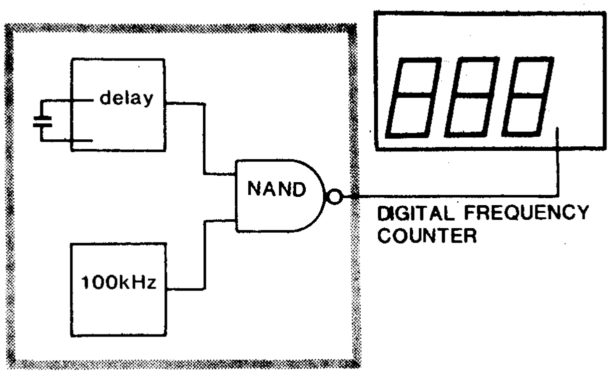
IC2 provides the reference frequency and should be set to 100kHz. You can use the Mini Frequency Counter to adjust this frequency via the 100k mini trim pot. Connect the input of the counter to pin 3 and adjust the pot with an insulated screwdriver. Do not touch any of the components when performing this task as it will alter the frequency of the oscillator.

You will now need 3 capacitors of known value. They should also have a close tolerance as the accuracy of your piece of test equipment can only be as accurate as the test specimen. Try for 2% or 5% tolerance components having these values: 330pf, 3300pf 330n amd 3.3mfd. They should be either mica or polyester. Do not use tantalum or electrolytic at this stage.

Move the flying jumper lead to the first terminal to bring the 2k mini pot into circuit. This range will test capacitors from 1 to 10mfd.

Connect the 3.3mfd capacitor and turn the 2k2 trim pot until the reading is 330. If you have a 6.8mfd or a 1mfd polyester capacitor, you can double check the acuracy and linear response. Ours was spot on. Move the jumper to the mid position. This is the nanofarad range. In other words you can consider the three zeros as .000mfd. Select the 330n capacitor and set the readouts to 330 via the 10k pot. Double check the reading with the 3n3 capacitor. You should get 003 on the readout.

Finally set the jumper to the pf x 100 range, and set the readout to 33 for the 3n3 capacitor. The 330pf capacitor will read 003 on the scale. This completes the setting up. Now you can go through your parts box and test all those "odd-ball" capacitors. You may get a surprise too, seeing how widespread capacitors really are! Some types are inherently accurate while others don't come anywhere near the stated value.

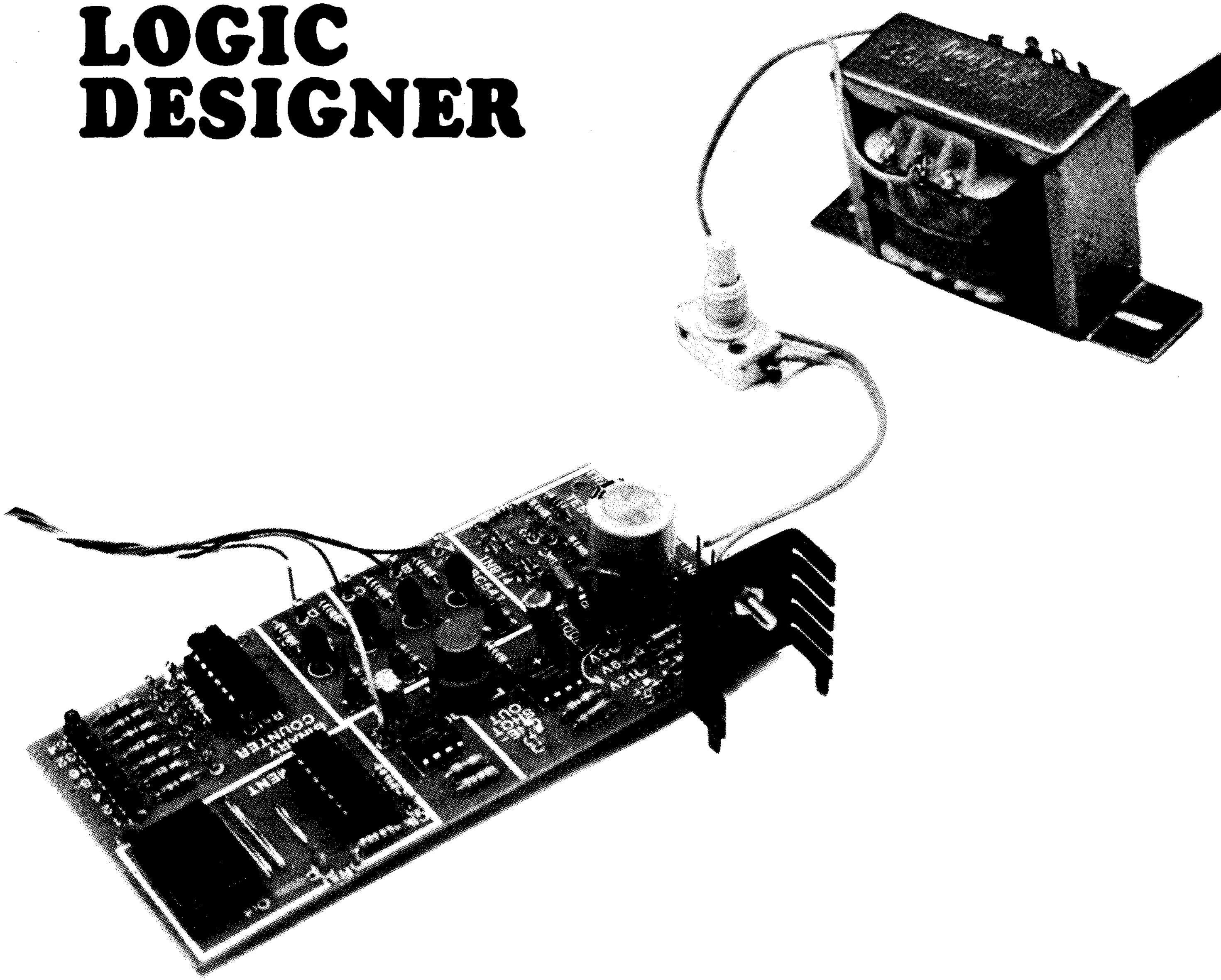


BLOCK DIAGRAM DIGITAL CAPACITANCE METER

The 100kHz multivibrator must be set to 100kHz by using an accurate reference. You can use the Mini Frequency Counter for this task, before connecting the two units. Four leads connect the two boards. Set the Mini Frequency Counter to RANGE L and connect a 6v or 9v battery. You will see three zeros on the display. This will indicate all is working. Calibrate the instrument as detailed in the first part of this article. Then you are ready to test all those unknown capacitors.

By adjusting the jumper lead on the Cap Meter board you will be able to test capacitors from 100pf to 10mfd with over-range. (Over-range means a 15mfd capacitor will read as '500' on the scale with the '1' at the front being the over-range.)

LOGIC DESIGNER



If you have never built a digital circuit before - this is where you start.

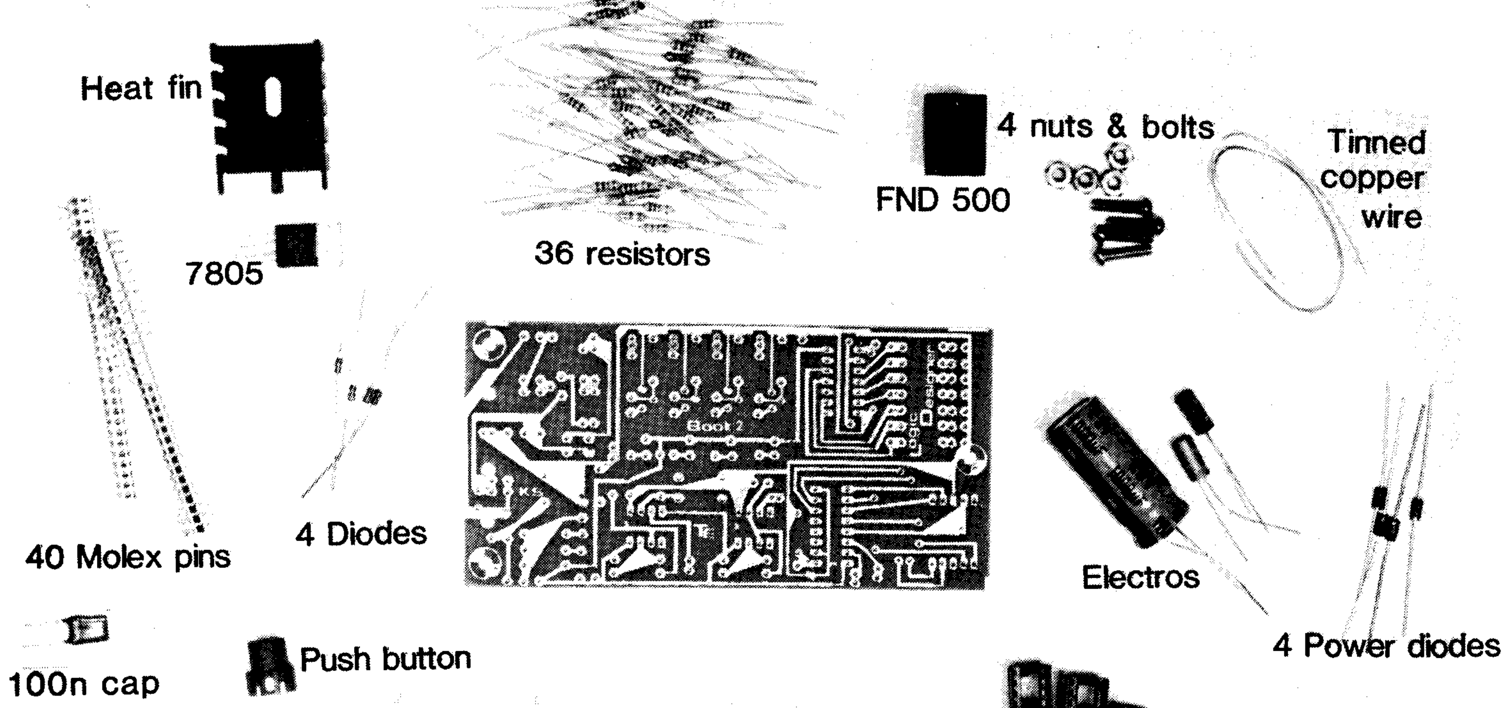
Our LOGIC DESIGNER is the beginning to digital designing and can be used in 2 ways.

Firstly it can be used by itself to perform counting operations in either binary or decimal or it can be combined with any of the "bread-boarding" systems currently available in all major shops.
This will enable you to design and test digital circuits while you are building them.

WHAT DOES THE LOGIC DESIGNER DO?

The logic designer is filled with features. It would not be possible to list all the possible combinations of the unit as the varous building blocks on the board can be interconnected to obtain a wide variety of counting and testing stages. You will be able to see what we mean after you build the project and start to interconnect the blocks yourself.

But we can give an outline of some of the features we have specially incorporated into the board. You can consider the DESIGNER as a piece of test equipment. It is specially designed for testing digital circuits. In fact it will only test digital circuits. So don't expect to get any results from an audio amplifier or radio. The type of digital circuits we are interested in have appeared in the pages of TALKING ELECTRONICS and with this tester you will be able to detect the logic levels at 4 different points at the one time. This is very handy when you are attempting to locate a HIGH on the output of a decade counter such as a CD 4017. With its 10 outputs sequencing in a rather awkward order of 3, 2, 4, 7, 10, 1, 5, 6, 9, 11, it would be handy to connect up the outputs and read them off in a logical order. This



PARTS LIST

2 - 555 timer IC's
1 - CD 4024 binary counter IC
1 - CD 4026 decade divider & driver
4 - BC 547 transistors
1 - 7805 regulator

1 - FND 500 display

12 - 3mm Red LEDs
1 - 3mm Green LED
4 - 1N 914 or 1N 4148 diodes
4 - 1N 4001 power diodes

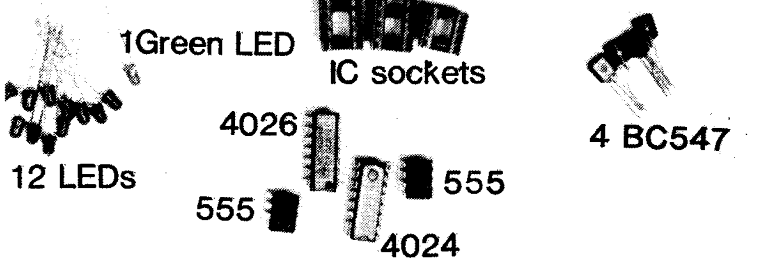
1 - 100n greencap @100v
2 - 1mfd 16v PC mount electroytic
1 - 1000mfd 16v to25v electrolytic

1 - 150R resistor ¼watt
2 - 220R
2 - 270R
4 - 330R
7 - 470R
2 - 1k
2 - 3k9
8 - 10k
1 - 22k
2 - 47k
4 - 100k

Note: Use 220R in place of 150R in power supply for accurate 9v range.

1 - 8 pin IC socket
1 - 14 pin IC socket
1 - 16 pin IC socket

push button (either round or square)
Heat fin for 7805
40 Molex pins
4 sets of nuts and bolts
10cm of tinned copper wire
fine solder
20 lengths of solid-core hook-up wire 15cm long



ADDITIONAL PARTS

These parts must be bought from you local electronics supplier.
Read through this project completely before ordering any of these parts.

POWER TRANSFORMER
To make a 150mA power supply you will need a transformer type 2851*. This is 240v to 12.6v CT @ 150mA.
If you want 5v or 9v but not 12v out of the power supply,you can get type 2840. This is 240v to 9v CT @ 150mA.

For a 1 amp power supply, use a transformer type 2155 This has tappings up to 15v @ 1 amp.

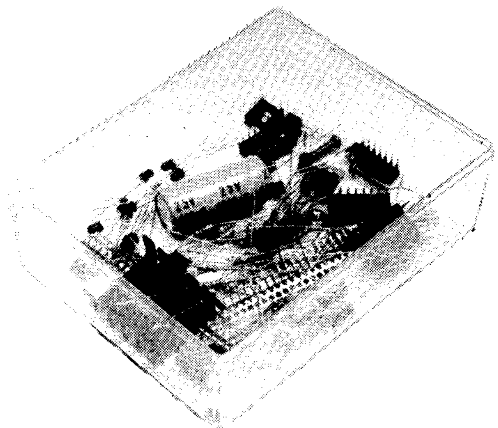
Power cord and 3 pin plug.
Jiffy or Zippy box type UB3 Part number 2753 (Dick Smith) or H 0103 (Altronics).
The LOGIC DESIGNER board is designed to fit onto these boxes in place of the aluminium lid.

*These numbers were original A&R TRANSFORMER numbers but are now an accepted part number throughout most electronic suppliers. So remember to give A&R credit for the original design, as they are made over-seas these days!

LED - Light Emitting Diode
FND - Fairchild Numeric Display

you can do with the 4 buffered HIGH detectors on the DESIGNER. Or take the situation where one delay circuit is to trip the next circuit and finally gate a counting circuit. Here you can use two of the buffer detectors and either the decade counter or the binary counter to get a readout. You will then be able to compare the tally of the project under test with the LOGIC DESIGNER tally.
The "one-shot" is especially useful for pulsing a circuit. Since it is fully debounced, it will produce a clean pulse to clock any digital counting circuit, just ONE count. And there is no chance of it giving two quick pulses. The 555 has a delay of about .1 seconds so the maximum repetition rate is 10Hz. But you wil not be able to achieve this manually. If you wish to clock a circuit at a faster rate, the 10Hz clock is available. This can be divided down to one of eight slower frequencies via the binary counter chip. As I said, it is not until you get into it yourself that you will understand the full range of possibilities for this piece of equipment.

■■■■■■■■■■■■■■■■■■



All the components come in a plastic box. They come properly packed in a bag and the IC's are protected in either foam or an IC sleeve. When you take them out, don't throw the IC's into the box with the other components as shown in the photo because the static electricity will damage them. Always keep them protected and don't handle them unnecessarily. Once they are fitted into IC sockets, they are quite robust.

Note: The foam we mention above is special CONDUCTING FOAM and not white styrene foam.

FEATURES OF THE LOGIC DESIGNER

Summary:

1. **AC power**
2. **Battery power**
3. **1.5 amp in-built power supply**
4. **'One-Shot'**
5. **10Hz clock**
6. **Divide by 10**
7. **0 - 9 counter**
8. **Transistor Tester**
9. **4 Buffer Amplifiers**
10. **Binary counter**

1. An in-built fully-regulated power supply using a 3-terminal regulator to supply 5v to 12v @ 1 amp.
2. The availability of using almost any power transformer from 6v @ 100mA to 18v @ 2 amp to supply the LOGIC DESIGNER as well as some to spare for any external circuit being constructed.
3. Battery operation: 6v to 12v.
4. A "one-shot" circuit with 300mA capability using a 555 to clock an external counting circuit. The push-button is fully debounced and has a maximum repetition rate of 10Hz.
5. A 10Hz clock using a 555 to give good driving facility to any circuit you are designing.
6. A decade divider provided by a CD 4026 to give a 5Hz or 1Hz clock rate.
7. A 0 - 9 counter using a CD 4026 and a FND 500 7-segment display. This must have a noise-free high-amplitude input signal to give an accurate readout.
8. A transistor tester to test all those unknown PNP or NPN transistors. You will be able to find out their correct lead configuration, their polarity and the fact that they do actually work.
9. Four buffer amplifiers to detect HIGH pulses on digital circuits. This will save you trying to find 4 multimeters.
10. A binary counter reading up to 128. This will teach you binary readouts and give you the availability of frequency division by 2, 4, 8, 16, 32, and 64.

By combining two or more of these features you will be able to create your own detection circuits, frequency dividing circuits and counting circuits. The scope is enormous and it will take but a few moments of experimenting for you to really become familiar with the possibilities.

HOW THE CIRCUIT WORKS

The transformer T1 supplies 12v AC into a full-wave bridge rectifier. You can use either a 9v transformer or a 15v transformer with equal success. Read the rest of the construction notes for the pros and cons of each voltage on the regulator chip if you wish to use anything other than a 12v transformer.

The output of the full-wave rectifier (in conjunction with the 1000mfd electrolytic) will produce about 17v DC with about 400 millivolts ripple when the LEDs in the project are NOT lit. The output ripple of the 7805 is only about 2 millivolts at 5v rail voltage. When all the LEDs are illuminated, the input ripple increases to 1v P-P and the output ripple of the regulator becomes 5 millivolts.

The regulator is designed to provide an output voltage which is 5v higher than the common lead. So, by increasing the voltage on the common lead we can produce 9v or 12v out of the regulator. The 100n capacitor in the output removes any high frequency noise spikes.

Each of the IC's must be considered as separate blocks as they are inter-connected via jumper leads. This is why the block diagram does not have any signal lines between blocks.

IC1 is wired as a one-shot. Pin 2 is made HIGH to prevent the IC from oscillating. When the push button is pressed,the chip will complete one cycle to give a HIGH for about one-tenth of a second and then go LOW.

The second 555, IC2, is wired as an astable multivibrator set to oscillate at about 10Hz. This is our main clock for the LOGIC DESIGNER. It is sufficiently slow enough to see most counting stages in operation.

IC3 is a self-contained chip capable of driving a 7-segment display. It has internal current-limiting so it is not necessary to provide each segment with a current limiting resistor. Both CLOCK and RESET pins are made available so you can control the display. The word CLOCK on most of these chips means you need to drive it with a clock pulse. It does not mean there is a clock inside the chip!

When theLOGIC DESIGNER is turned on, you will need to take the RESET pin of the CD 4026 to HIGH to reset the chip and produce a zero on thedisplay - otherwise almost any combination of segments will appear on the screen and there is no guarantee where the chip will begin counting.

If you look at the circuit diagram carefully, you will notice the TRANSISTOR TESTER section is entirely powered from the AC side of the transformer. The two 1k resistors become voltage dividers, providing the circuit with 6v AC.The two base-biasing resistors reduce the input voltage of the transistor tester even further to about 3.5v. This is just sufficient to operate the LEDs and yet low enough to prevent damage to the transistor being tested. The 270R resistor provides current limiting for the LEDs.

When the AC supply poduces a positive on the 270R resistor, and an NPN transistor is fitted into the socket, the voltage on its base will turn it on and it will short out the voltage to the two LEDs. When the AC reverses, the transistor will be turned off and the second LED will be forward biased to indicate a good NPN transistor.

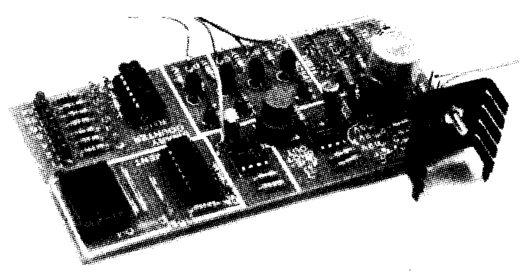
For a PNP transistor, the same reasoning will apply but this time the first LED will turn on when the transistor is in its non-conducting condition.

The 4 diodes are designed to detect a short in either junction of a faulty transistor. If this is the case, one half of the transistor acts like a diode and would normally conduct to indicate a good transistor. To overcome this, the 4 diodes are added in series with the collector.

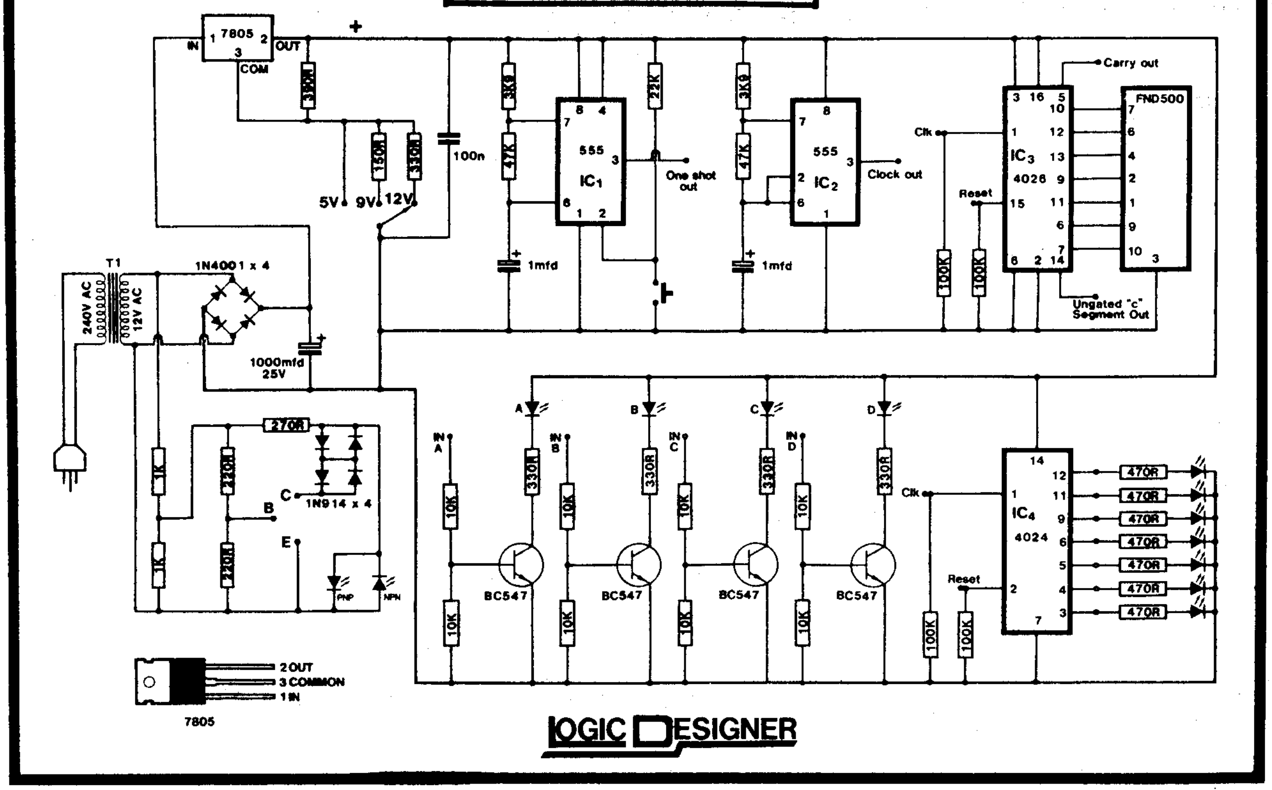
4 Buffer amplifiers are provided to detect HIGH pulses. They can all act at the same time to give 4 simultaneous readouts. This is very handy when reading the input and output of a particular circuit at the same time.

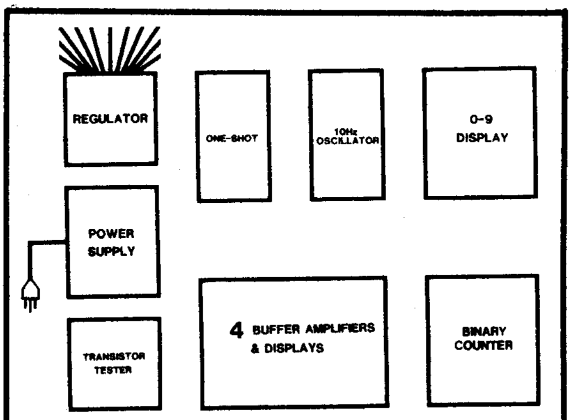
The buffer transistors are wired as common emitter amplifiers, enabling them to register any voltage above 1v, with a low loading on the circuit being tested.

The remaining block, IC4, is a 7-stage binary counter with each output terminating at a LED. If you use only one output at a time, the possible divisions are 2,4,8,16,32 and 64. other divisions are possible if more than one output is used at a time. This is called-DECODING. See block 33 of the 10 MINUTE DIGITAL COURSE for a full explanation of decoding the CD 4024. The readout can be set to zero by taking the RESET pin HIGH at any time at all.

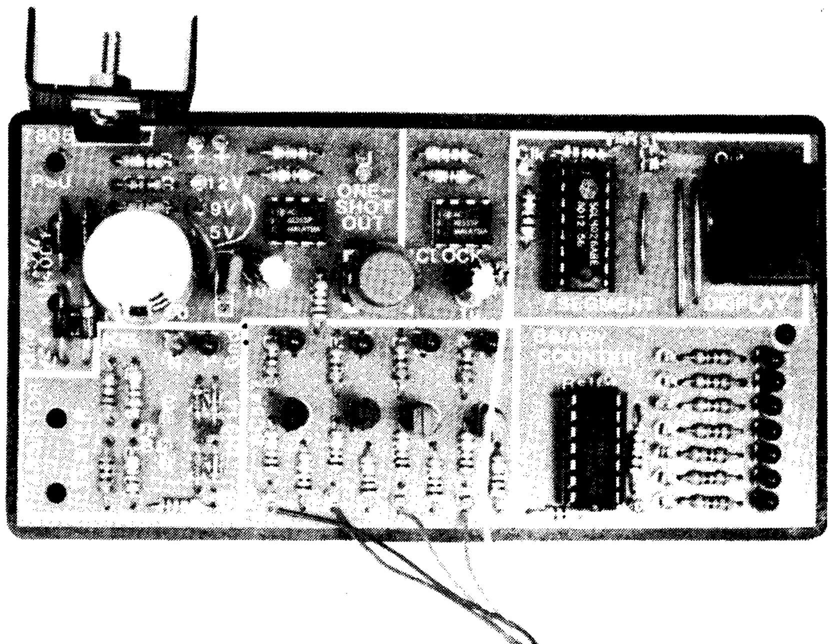


THE CIRCUIT





BLOCK DIAGRAM



The block diagram is shown as separate blocks, with no inter-connecting lines. This is because each block is independent. They are connected together with jumper leads during experiments.

Notice the position of the squares in the block diagram correspond almost exactly with the building blocks on the printed circuit board. Also, if you look at the circuit diagram, the components are in a very similar position to the PC layout. This makes for easy circuit tracing and fast parts locating.

To keep the circuit diagram uncluttered, we have omitted R1......R36 and C1.....C4. Parts should always be located by reference back to IC pin Nos or transistor leads.

POWERING THE UNIT

CHOOSING A TRANSFORMER

You will need to decide the amount of current you will need for powering projects outside the LOGIC DESIGNER. To power the LOGIC DESIGNER itself, almost any power transformer will be suitable as it takes a maximum of 150mA when all the LEDs are illuminated. Even the smallest of power transformers will be sufficient to supply this project. It's only when you wish to supply outside projects that you will need to consider a larger transformer. The diode bridge will pass 1 amp and the regulator will supply 1 amp. The regulator rating will depend on the supply voltage and the output voltage. You will need to read the set of notes included in this issue to understand the amount of heating which will take place in the regulator at the various voltage levels. Remember, it is only necessary to have the input 3v higher than the output for the regulator to operate correctly. And this will ensure the greatest output current for the least heating.

If you are at all unsure about wiring power transformers, there is an alternative in the form of a plug-in transformer.

Dick Smith markets a plug pack containing a 240v to 12v transformer capable of delivering 500mA. This comes in a sealed package which plugs straight into the power point, and has a couple of metres of cable supplying the 12v AC. This can be soldered straight onto the AC on the PC board and you don't have to worry about wiring any 240v leads.

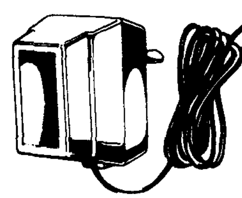
If you don't like the AC at all, you would be quite safe choosing one lantern battery to create the 5v supply or two batteries to produce 9v via the regulator. The regulator would have to be omitted if you require 12v. The batteries should be connected directly across the 1000mfd electrolytic. From my experience with lantern batteries, they don't seem to last very long and would have to be checked after a few weeks of operation. You will possibly find the output voltage has fallen to 10v or less and they do not maintain a constant voltage under load. This is when "dry batteries" let you down. They begin to lose voltage very quickly and this state of affairs can not be accepted at all with TTL circuits. Also don't forget, you cannot use the transistor tester when supplying the DESIGNER with DC.

These transformers are imported by Dick Smith, Altronics and Ellistronics:

(your local stockist will also have supplies).

12v AC POWER SUPPLY

@ 500mA



M - 9555

12v AC

Ideal power supply for projects requiring 12V AC at 500mA. Terminations are wire for easy connection into any circuit.

MAINS MINI POWER

240V 6 12.6V CT at 150 MA
SEC Certified to ASC 126

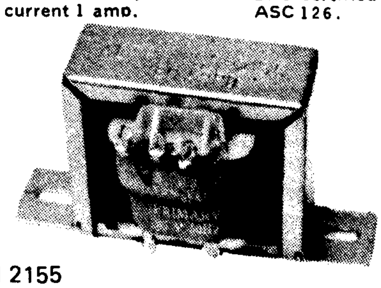
M 2851

MAINS LOW POWER
MULTITAPPED

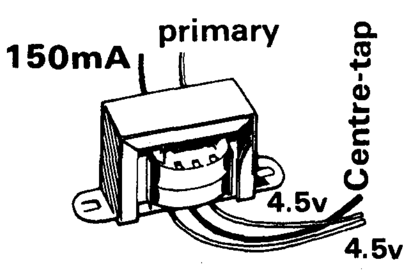
Primary 240V AC
Secondary volts 6.3, 7.5, 8.5, 9.5, 12.6, 15V

Max. secondary current 1 amp.

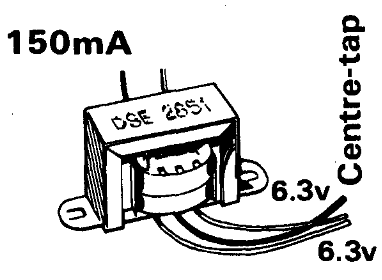
SEC Certified to ASC 126.



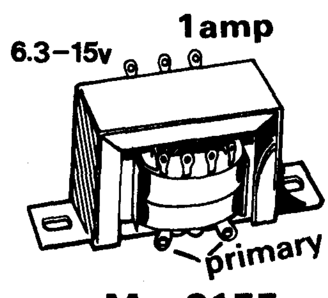
M 2155



M - 2840



M - 2851



M - 2155

POWERING THE UNIT

WIRING THE TRANSFORMER

The primary and secondary leads of the transformer should be identified with colour-coded wire. Check this out before making any connections and especially before cutting any leads to length. The primary winding goes to the Mains via the fuse and switch. When you look into the transformer, you will be able to see the various gauges of wire used in its construction. The primary winding uses the finest gauge and has the greatest number of turns. This is always the safest way to locate the primary from the secondary, if they are on separate formers or bobbins. You can also locate the primary by using a multimeter set to low ohms range however the resistance of the winding can sometimes be so small (15 to 50 ohms) that you cannot always distinguish it from the secondary. So always look first.

When you have located the primary, it does not matter which wire goes to the switch. Just make sure that the lead is properly soldered to the switch or held firmly into the terminal block.

If the secondary widing is centre-tapped, you will need to use our other notes.
For a single winding, connect the two wires to the AC input on the PC board. If the transformer has lug terminals, you will need to supply lengths of flex to the PC board.

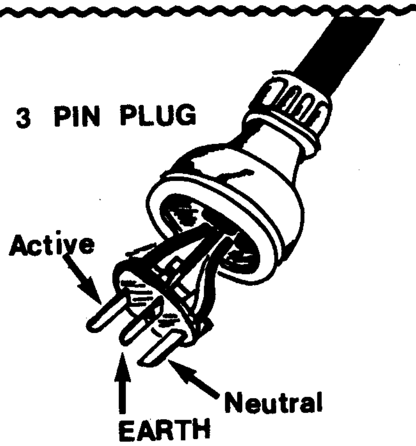
These are the 4 transformers we have found to be suitable:
M - 2840 9v CT @150mA
M - 2851 12.6v CT @ 150mA
M - 2155 up to 15v @ 1 amp
M - 9555 AC plug-pack 12v AC @ 500mA

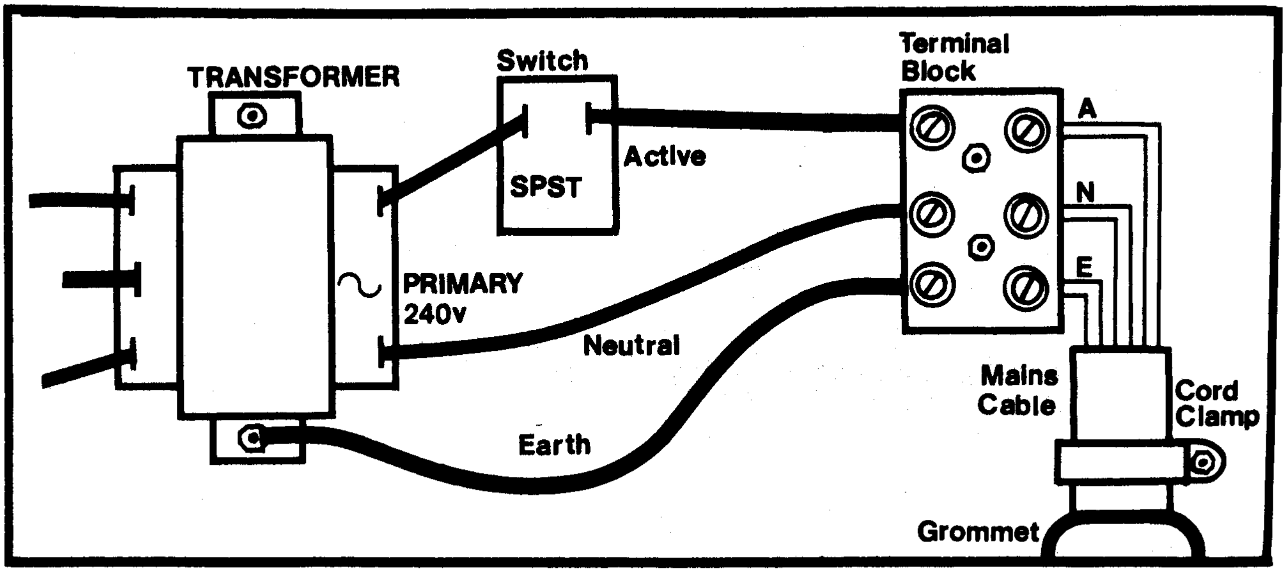
WIRING A PLUG-TOP

This simple operation looks fool-proof but needs concentration to avoid a mistake, because the new colour-code for the flex is so confusing.

Begin with the GREEN or YELLOW striped lead to the EARTH or plated pin. The RED or BROWN goes to the left-hand ACTIVE pin and the BLUE or BLACK to the right-hand or NEUTRAL pin.

3 PIN PLUG





INSIDE THE JIFFY BOX.

Keep to this layout to prevent errors. No earth connection is required with double insulated transformers. You can use 'twin-figure-eight' lead if you wish.

POWERING THE UNIT

HOW TO WIRE A CT TRANSFORMER

Two of the transformers we have suggested for this project are centre-tapped types. This means they have a tapping in the centre of the secondary winding which effectively splits the designated voltage in half.

This means the rating of 12.6v and 9v is the result of connecting across the two outer leads and if one lead and the centre-tap were chosen, the result would be 4.5v and 6.3v.

For the LOGIC DESIGNER, the two outer leads must be used and the centre tap can be cut off. But in actual practice the centre tap must not be removed. This is because the centre tap forms part of the secondary winding.

What happens is half the secondary is wound on the bobbin and then about 8 or 10 cm of it emerges. This is folded back and twisted together and the remainder of the secondary wound on the bobbin.

If this tap were to be cut off, the two windings would be separated and the transformer would not work.

The best solution is to tape up the centre tap and carefully tuck it out of the way.

The two outer leads are now connected to the terminals marked AC on the PC board and the primary of the transformer wired up in the normal way.

Remember, these transformers are only very low output current types and the Logic Designer itself takes nearly 80mA, leaving very little for any additional projects.

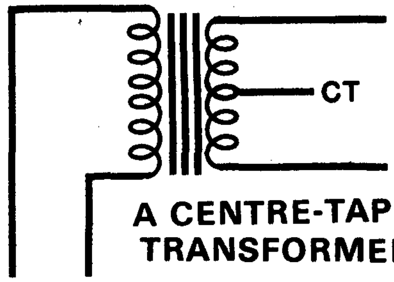
NOTE: One small design fault exists in the power supply section of the LOGIC DESIGNER.

If the voltage is adjusted when the power is applied, the output voltage will rise to 14 or 15v. This is due to the common terminal of the regulator being allowed to 'float' high when the range-switch is opened. This voltage may be too high for some chips and damage could result.

To prevent any damage from occuring, turn of the power before changing the voltage.

The LOGIC DESIGNER can be used to test other projects in this book as well as supplying power to them.

It can be teamed up with the LOGIC PROBE to test some of the more complex projects such as the LOTTO SELECTOR, CLOCK and HANGMAN to gain a full view of what is happening.



A CENTRE-TAPPED TRANSFORMER

A 12.6v centre-tapped transformer does NOT have two 12.6v windings but two 6.3v windings, making a total of 12.6v across the two outer leads. The two outer leads are always the same colour to aid identification.

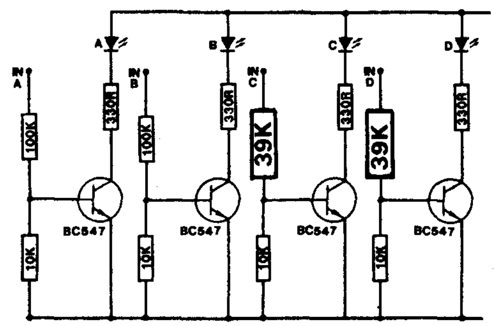
MODIFYING THE BUFFERS

An interesting modification can be made to one or more of the BUFFER transistors to make them detect a LOGIC HIGH.

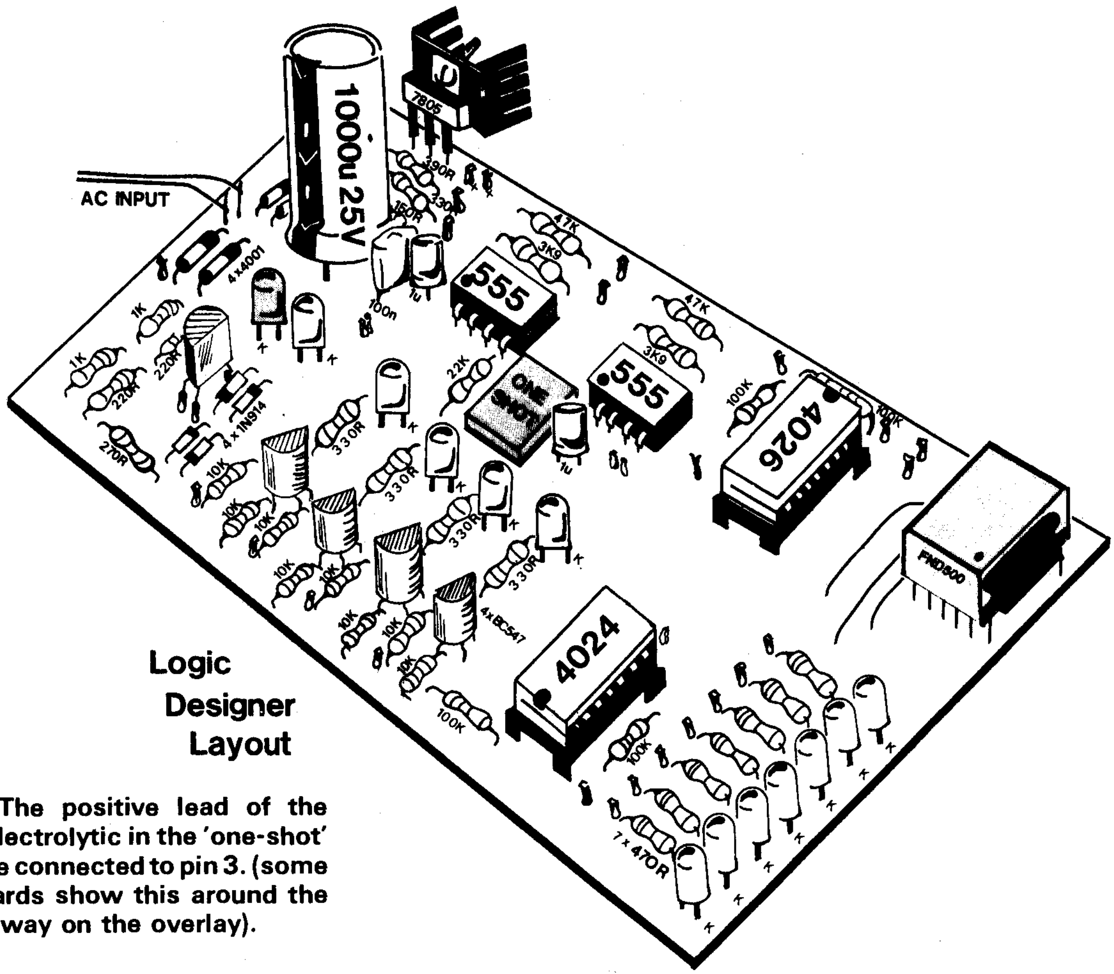
At the moment they will detect a voltage which is higher than about .9v but this does not necessarily mean it is sufficiently high enough to clock a logic circuit. To be rated as a LOGIC HIGH voltage it must be above 70% of the rail voltage.

By changing the buffer resistor to 100k, the transistor will only turn on when the applied voltage is greater than 8v. This arrangement is suitable for CMOS circuits working in the 10v to 12v range.

For TTL circuits use a 39k for the buffer resistor.



CONSTRUCTION



Logic Designer Layout

Note: The positive lead of the 1mfd electrolytic in the 'one-shot' must be connected to pin 3. (some PC boards show this around the wrong way on the overlay).

There are two ways of gathering all the parts required for a project such as this. One way is to search through your parts box for all the resistors and capacitors LEDs and diodes and then go out and buy the components which are missing. This is both uneconomical and unwise. Most often you don't have the correct value component and try to make do with the nearest value. Additionally the components are not the same size and the project finishes up looking like a pile of junk. We have seen it so many times with the units sent in for repair. The most economical and smartest thing to do is buy a kit of components. At least all the parts will be co-ordinated and designed to fit into the space available on the board. And with all new components the project is sure to work. Most of the large electronic suppliers will have a kit of components or stock the individual components for this project as all the items are standard values and should be readily available. When you have collected all the components, lay them out on the bench and make sure you can identify each of the values. Start at the left hand side of the board and insert the four diodes. Progress across the board inserting each component as you come to it. The overlay on the board makes it very easy to identify the positioning of the polarized components such as the electrolytics integrated circuits transistors and diodes. Four jumper links are required near the 7-segment display and these can be made from the ends of a resistor. When soldering in the molex pins, keep them attached to the carrier buss by just bending them at right angles. After soldering in position it can be bent off the carrier. This saves burning your fingers.

The overlay on the board should be enough guide for inserting the parts. If you are still unsure, use the overlay included in this section. it is most important that you correctly position the IC's and 7-segment display. The notch or dimple on the IC's must be positioned as shown in the layout as the pins will not prevent you inserting the component around the wrong way.

The FND 500 7-segment display has no pin identification at all and you must rely on the part-identification on the lower edge. This may be just a code number or a manufacturers name. Without this guide you will have to follow our other notes on identifying the FND 500.

When all the parts have been inserted, go over every component again and confirm that it has been fitted in the correct place on the board and that it is positioned around the right way. So many times we find transistors and electrolytics incorrectly fitted. It's always the simplest thing that lets you down.

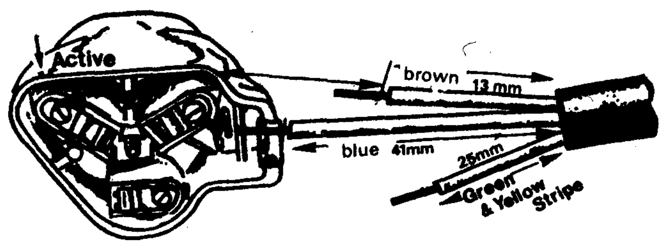
All that is required now is to fit a power transformer. You should already know the current rating of the transformer you are using and if it has a centre-tapped secondary or a single winding. To power this project and any other circuit you are designing (containing up to 4 or 5 IC's) I would suggest 150mA to 300mA secondary would be sufficient. This type of transformer can be mounted inside a Jiffy box and the board fitted on top (in place of the lid). This will hide the dangerous 240v leads and give you plenty of mounting room for the transformer. If you are not 100% certain about wiring up the mains side of the project, I suggest you approach a qualified person to at least inspect what you have done. He shouldn't charge you anything for inspecting it and should be pleased to assist.

This may be your electronics teacher, a friend with electrical know-how or your local electrical shop. Don't take any risks. Especially with the 3-pin plug and terminal block. Three-core leads are now required to conform to the new colour coding.This has been mainly designed for color-blind people as it does not follow any electronic logic. The brown wire is taken to the active pin. This pin is sometimes identified on the plug so look for it. If you are not sure, the active pin is the left-hand hole of the wall-socket. The blue lead goes to the neutral pin - previously the black lead and finally the green lead with the yellow stripe goes to the earth pin.

This lead should be a little longer than the other two so that no strain is placed on it. This means it will be the last lead to fracture if the cord is tugged.

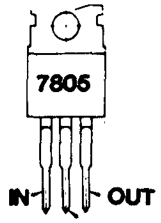
Keep the same wiring arrangement inside the Jiffy box as shown in the layout diagram. The earth wire can be taken to the fixing lug of the transformer. In some cases it will not be needed as the transformers are classified as double insulated.

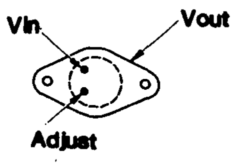
WIRING A PIGGY-BACK PLUG



VOLTAGE REGULATOR

We have chosen a 7805 voltage regulator to supply this project. It is the cheapest and easiest to fit on the board. If you intend to power other projects from this supply it will be necessary to remotely mount the chip on a large heatsink. Under these conditions it would be possible to use the LM 317k 3-terminal adjustable regulator. Since it is in a TO-3 package, it will require 2 screws for mounting and thermal compound for adequate heat conduction. If possible, try to avoid a mica insulating washer. This will reduce the efficiency slightly. Keep the whole heat fin away from any eathing points to preventing it shorting out.





HEATSINKING

The heatsink shown in all the photos is only suitable for dissipating the power requirements of the LOGIC DESIGNER itself. The heat fin will only dissipate about 1 watt.

If you take the case of 12v being supplied to the input terminal of the regulator and 5v emerging from the output; the wattage being lost in the regulator will be very close to 1 watt when 150mA is flowing. Any higher current requirements will overheat the regulator and it may be damaged. I have not had very much success with the current limiting ability of the 7805 in a shorted output mode - it just gets fiercely hot and fails to regulate. Nor does it like to supply any more than a few hundred milliamps into a free-standing heat fin. If you will be requiring more than 500mA for long periods of time you will need to remove the regulator from the board and mount it on a solid piece of aluminium capable of dissipating the excessive heat. When clamping the regulator, use some form of thermal compound and connect fine flex to the three leads to prevent them from fracturing and falling off.

After checking all the AC side of the transformer wiring and looking over the circuit board for any solder shorts, the power is applied. All the circuits in the LOGIC DESIGNER have been designed for 5v operation. This will enable you to experiment and build up circuits with either CMOS or TTL chips.
All the IC's in the LOGIC DESIGNER will operate in the range 5v to 12v so we have chosen 5v as being common to both chips.
When the power is applied, the 7-segment display will light up along with the LEDs in the transistor tester. These act as power-on indicators. By their intensity you will know the supply is delivering 5v.
If the LEDs do not light up, check the regulator for temperature rise. If it is getting very hot, you may have a short in the output. For best operation, the power transformer should be rated a 9v. This gives the minimum input -output voltage on the regulator and will create the least amount of heating inside the regulator.
If the regulator is still cold, feel the power transformer for overheating. A temperature rise here will indicate a short in the diode bridge or an incorrectly inserted diode.
Don't forget the possibility of solder bridging two tracks and causing a short-circuit. If the LEDs light up you can assume the power supply section is working.
The 555 (nearest the power supply) is connected as a "one-shot". This means it will produce an output of one pulse every time the push button is pressed. We have already explained on numerous

MOLEX PINS

To keep this project inexpensive, we have chosen Molex pins for the input and output terminals. These aren't the best choice in the world as they fracture easily and break off. But if used carefully,they will last quite a long time. If you have any better suggestions, please let me know.

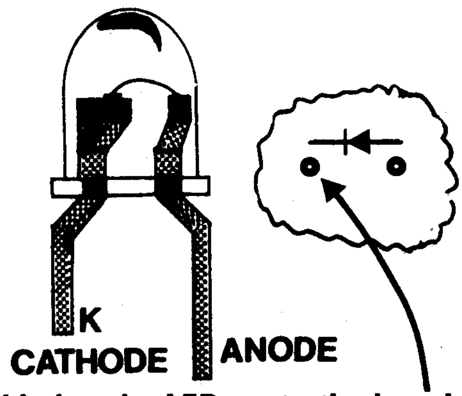
INSERTING THE MOLEX PINS

If you find it difficult holding the Molex pins while soldering, you can make up an inserting tool by soldering a Molex pin onto a piece of copper wire which is about 8cm long as shown in the sketch. This will save burning your fingers.

TESTING THE LEDs

The 13 Light Emitting Diodes are opaque and you can see inside them. The cathode lead is shorter than the anode when you buy the LEDs but how do you tell which is the cathode lead after you have snipped them both to the same length? Since you can see inside the body of the LED, you will find that the cathode lead 'dog-legs' over towards the other lead or it is the larger of the two.

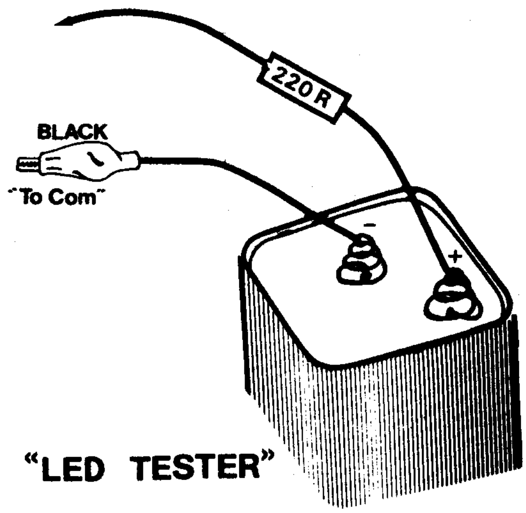
Look into each LED:



When soldering the LEDs onto the board, the short lead goes down this hole.

On the PC overlay, the symbol for a LED is the same as for a diode. Don't get mixed up. See the photo and layout diagram before fitting any parts.

After soldering the LEDs into cirucit, they should be tested as they can be easily damaged by heat. Use a 6v lantern battery fitted with a 220R dropping resistor to check each LED individually.



"LED TESTER"

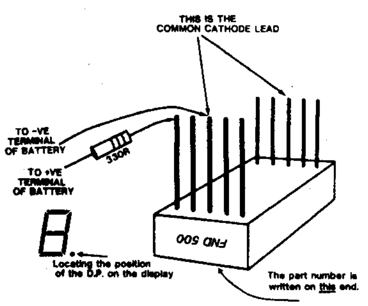
occasions that a push button has contact bounce and no matter how well the push button is constructed, it will send more than one pulse to a digital circuit if it is used by itself. It is absolutely essential that all push buttons are de-bounced. This entails some sort of delaying circuit so that the closing of the contacts are all grouped together to register as a count of one. The other necessary part of a "one-shot" is the speed with which the circuit changes from a LOW to a HIGH. If the rise time is slow, the digital circuit accepting the pulse will count up to 10 or 100 counts since there is an interval between the LOW and HIGH called the indeterminate zone. If you sit in this area with a voltage of about ½ Vdd the IC will tick over rapidly.

Our "one-shot" circuit avoids any of these troubles. It provides a clean pulse with a very high slew rate. All CMOS circuits will see this as one single pulse. To make the de-bouncing even more effective, we have introduced a long delay-time so that you cannot produce any more than about 10 pulses per second.

IDENTIFYING THE FND 500

Since we could not find any identification on the FND 500, such as a dimple in the case or a chamfer on one of the corners, we had to conclude that the identifying printing on the end of the pack was the sole identification. And believe me, it wasn't very satisfactory. Within a short time, handling the FND 500 had rubbed the numbers off. To identify the decimal point on the display we had to use a battery and 330R resistor as shown in the diagram.

If you solder the FND 500 into the project around the wrong way by mistake, it will not be damaged. All that will happen is the outer segments will light up in an odd pattern and the decimal pont will light in place of the central segment. You will not get any recognisable numbers. So, to avoid having to desolder the display, check first.



The secret to making a 555 into a "one-shot" lies in pin 2. By keeping it tied high via a resistor the 555 will be turned off. To turn the 555 on, pin 2 must be brought down to ⅓ of the supply voltage - or less.

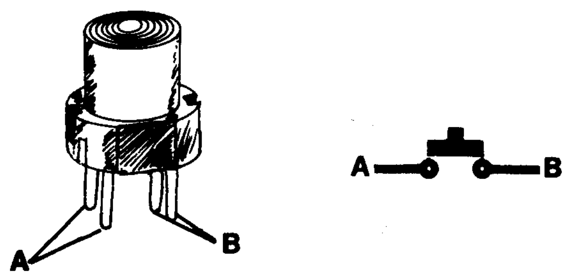
When the 555 is sitting ready to give its pulse, pin 7 is effectively connected to ground via the circuit inside the IC. This means the electrolytic will be sitting in an uncharged condition - because pin 6 does not provide any power out of the IC. It only detects the voltage on the electrolytic. When the voltage on pin 2 comes down to ⅓ of the rail voltage, the IC turns on. The short on pin 7 is removed and the electrolytic begins to charge up via the two resistors.

When the voltage rises to ⅔ of the supply voltage, pin 6 turns the IC off. We are assuming pin 2 has been taken HIGH before the electrolytic has triggered pin 6. If pin 2 is still LOW, the IC will remain turned on and will turn off as soon as the push button is released.

THE PUSH BUTTON

The Printed Circuit Board will accept either a square push button or a new-style push-button as shown in the sketch. These new buttons are cheaper and look much better than any of the previous types.

Either of them can be soldered onto the board in either direction (that is: either way round) as we are using only the diagonally opposite corners for the switching circuit.



To see the one-shot in action, connect the output of the 555 to one of the inputs of the buffers. Touch the pulse button very quickly. The LED will turn on and remain on for a short time, and then go out. If you keep your finger on the button for a long time you will not see the action of the one-shot. If the LED stays on in this manner, you can be sure the one-shot is operating. Test the other three buffer transistors for similar operation.

If the circuit does not show any delaying action on any of the LEDs, you should try the electrolytic for correct insertion. Also check the value of the timing resistor. By placing a multimeter on pin6 you will be able to detect the increasing voltage as the electrolytic charges. The output pin 3 goes HIGH when the IC is turned on. It is normally sitting in a LOW condition.

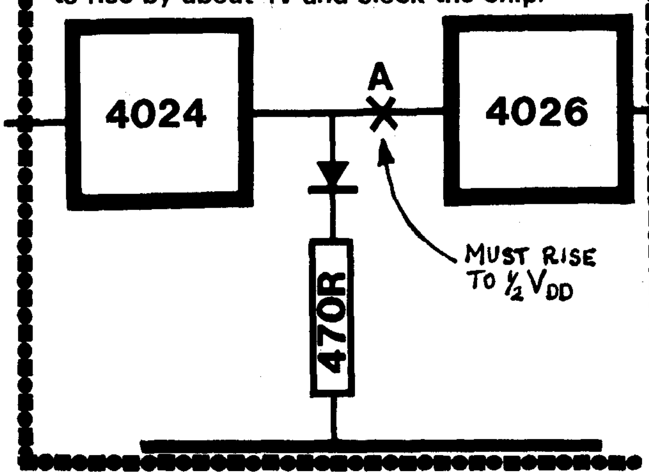
The second 555 is connected in an astable or free-running mode and will produce 10 pulses per second from the output pin 3.
You have three ways of testing this circuit. It can be connected to any the buffer transistors or the 7-segment readout via the CD 4026 or it can be used to pulse the binary counter IC (CD 4024). If you try all the three methods you will become acquainted with the ways of detecting an output. This just about covers the LOGIC DESIGNER. The only section left is the TRANSISTOR TESTER.

The transistor tester is extremely handy. How many times have you required a lead-out for a particular transistor only to find it unlisted, even in a book of 30,000 types. It always happens. The very transistor you require for a cross-reference is missing from the list. With this very simple device you can test almost any type of transistor and identify its important static characteristics. By this I mean you will be able to determine if it is a PNP or NPN transistor and identify the COLLECTOR, BASE and EMITTER leads. It will also show if the transistor has a short in either junction.The characteristics it will not reveal are its current handling capability, maximum frequency of operation, leakage or gain.These you have to ascertain from data.
To operate the transistor tester it is necessary to power the LOGIC DESIGNER from the mains. The 50Hz AC is rectified by the transistor under test to light one of two LEDs. Battery operation would not give this AC input and unfortunately the transistor tester is in-operative with battery power or even when using a battery eliminator such as a plug pack. There is one plug-pack however which has an AC output and this is described in the section headed CHOOSING A TRANSFORMER.
The testing of the DESIGNER has now been covered. Make sure it woks properly before using it as a test rig. If you have any difficulty with one of the stages, read the 10 MINUTE DIGITAL COURSE. It covers 4 of the circuits. The DESIGN YOUR OWN POWER SUPPLY series covers the power supply section. The only two new sections are the buffer transistors and the transistor tester but these are extremely simple.

MEASURING THRESHOLD VOLTAGES

Digital IC's must have at least ½Vdd (rail voltage) to enable them to clock.

This situation becomes critical when an output is trying to drive a LED and also clock another IC. Normally you would provide a buffer to supply the LED with full illumination, but in our simple project we omitted this luxury. The LED and dropper resistor shown in the diagram will reduce the output voltage of the 4024 to very near ½Vdd and it may be difficult to produce enough swing to drive the 4026. If you find the 4026 does not clock, the 470R resistor can be increased to 560R. This will allow the voltage to rise by about 1v and clock the chip.



DIODES

For this project, almost any 1 amp diode will be suitable. 1N 4001 are 50v diodes, 1N 4002 are 100v diodes, 1N 4004 are 400v diodes and 1N 4007 are 1000v diodes.

Since our power supply is only producing 12 to 20v, we can use any PIV rated diode and still have ample to spare.

USING SOCKETS

I can think of nothing more appropriate than to use sockets for this project. This will give you the added advantage of being able to test the 555 IC's, 4024 IC's and 4026 IC's. This will also prevent the possibility of damaging the IC's when they are soldered into the project.If you mount the FND 500 on Molex pins, you will be able to test other FND 500 displays for segment luminance. This is very handy when you buy a bag of mixed displays and wish to test them for output before soldering them into a circuit. At some later stage we hope to produce a designer board to take 10 of our most popular chips and set them into a fully operational circuit. This will take so much of the frustration out of solving faulty circuit operation by eliminating the chip as a suspect. You merely transfer the chip from your current project and place it in the tester for a low-frequency test. You will be able to test all the gates and any in-built counting operations. Until this board is designed, you will have to accept the 3 test sockets in this project.

LAST MINUTE THOUGHTS

Hopefully, these notes will cover all your questions and problems with building this project. As such, we are not offering a repair service on this board as we think it is not necessary. If you do have any problems with the construction you are encouraged to phone us on our query line: 584 2386, for assistance. Many people are taking advantage of this query service to get them out of a problem.

Take it from me, the board is so simple that the project is guaranteed to work. If you have any trouble with one of the sections, get someone else to follow through your construction - because you will be oblivious to the mistake. This usually comes from incorrectly inserted parts, poor soldering or cooking a component. A good soldering iron is essential. Use a low-wattage type and don't think of using an instant-heat type.

We produced a number of proto-type units of this project and they all worked very well. After a little experimenting, one of the staff (Ken) found a couple of additional uses for the project which I hadn't even thought of. Most of these have been presented in the following section 13 EXPERIMENTS WITH THE LOGIC DESIGNER. These will highlight the versatility of the LOGIC DESIGNER and I am quite sure you will find even more circuit arrangements.

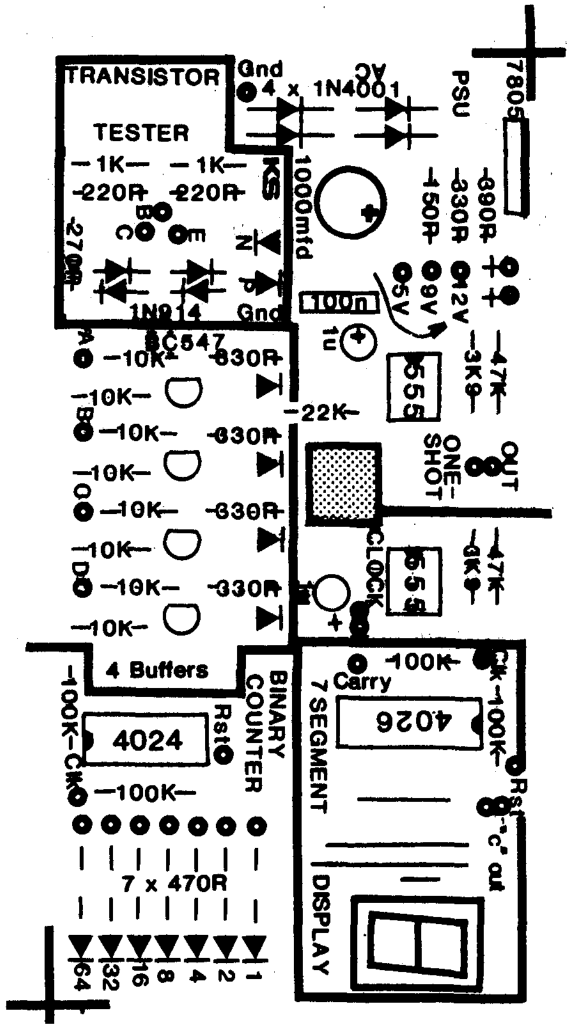
USING THE 240v

Many schools do not permit the construction of mains operated projects when live wiring is required in its construction. That's where you'll find the Dick Smith AC Plug-Pack saves the day. At a pinch, some of the other plug-packs can be taken apart and the diodes removed along with the smoothing electrolytic. So you have a number of alternatives to having to wire up a power transformer.

As with some of our past projects, this board is modular. It fits neatly onto a Jiffy box and the transformer can be fitted inside. Don't forget to add a few ventilation holes to allow the heat to escape and use a cable clamp to keep the power cord from pulling on the wiring.

If you haven't already realized, this LOGIC DESIGNER is intended to be used with bread-board. These breadboarding systems have proven immensely popular. The last section of this book contains an assortment of layout grids the exact same size as the most popular bread-board. The bench layout we have found to be best arrangement is to stick the bread-board onto the work-bench with a piece of double-sided sticky-tape and place the LOGIC DESIGNER on the far side. This will allow you full access to the bread-board and yet keep close proximity to the LOGIC DESIGNER so that the jumper leads can be kept short.

Note the positive lead on the 1mfd electrolytic in the 'one-shot'.



OVER-LAY OF PC BOARD

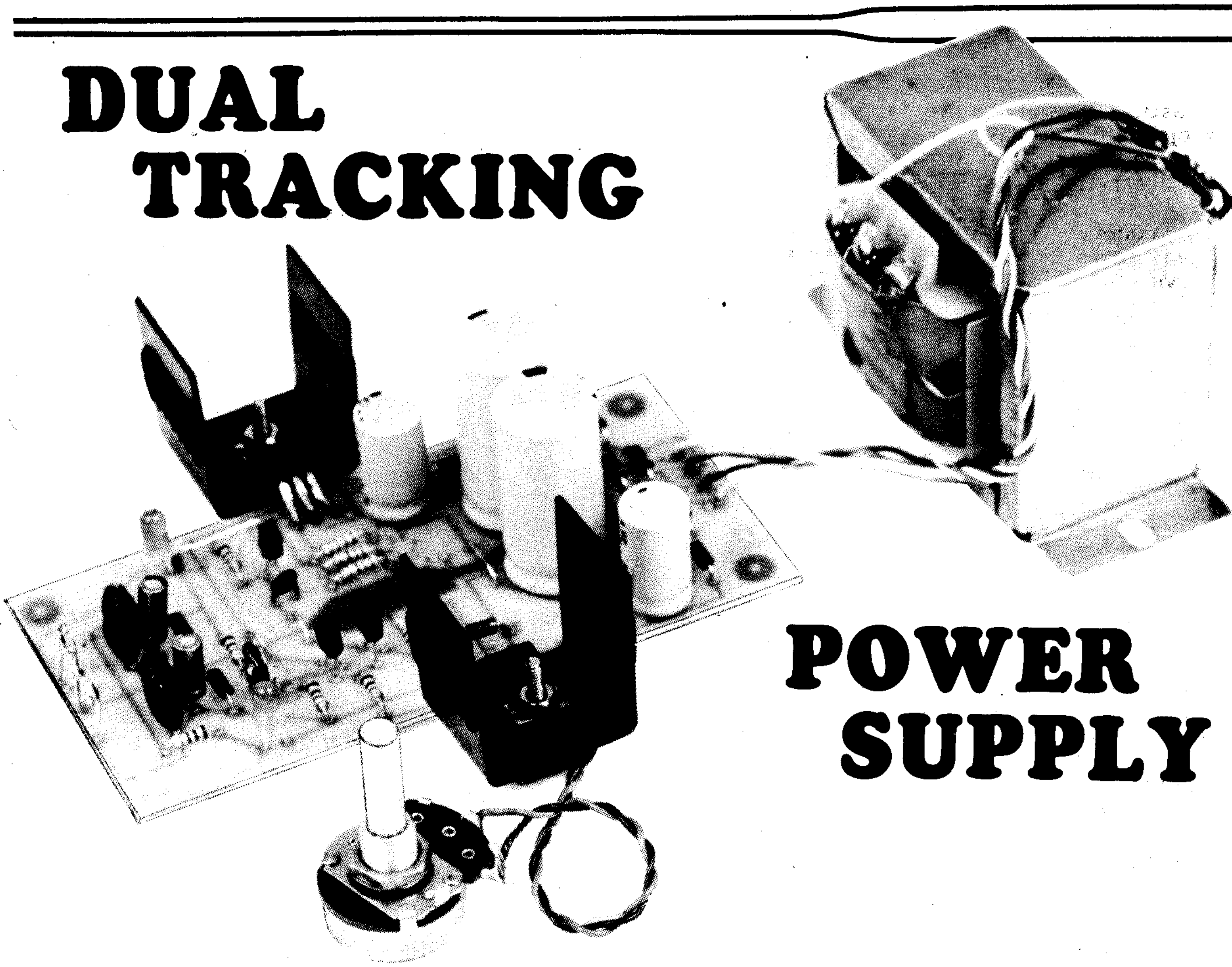
Included in the kit is a bunch of hook-up wires similar to telephone wire. If you require more of this particular wire, it is available from Dick Smith. It comes in 2 sizes. 2-pair and 6-pair. Don't use ordinary hook-up wire as it spreads the contacts in the bread-board and damages the Molex pins in our project.

This just about completes the project. I hope you have as much enjoyment in construction this design-aid as we had in preparing it. With encouragement from you, we will bring out a more complex version using Schmitt triggers, inverters and flip-flops.

Until that time,
the best of luck with the project,

Colin Mitchell.

DUAL TRACKING POWER SUPPLY



A power supply is a very nice piece of equipment to have on the bench.

It produces a steady, reliable voltage with very low ripple and can take the place of myriads of dry cells or ni-cad batteries. It comes to its own when a heavy current is required for an extended period of time. In fact it has all the advantages of an accumulator supply without the hassles.

Power supply projects have been around for a long time but most of them had a number of short-comings. They were either too expensive to build or provided only some of the features required for modern experiments.

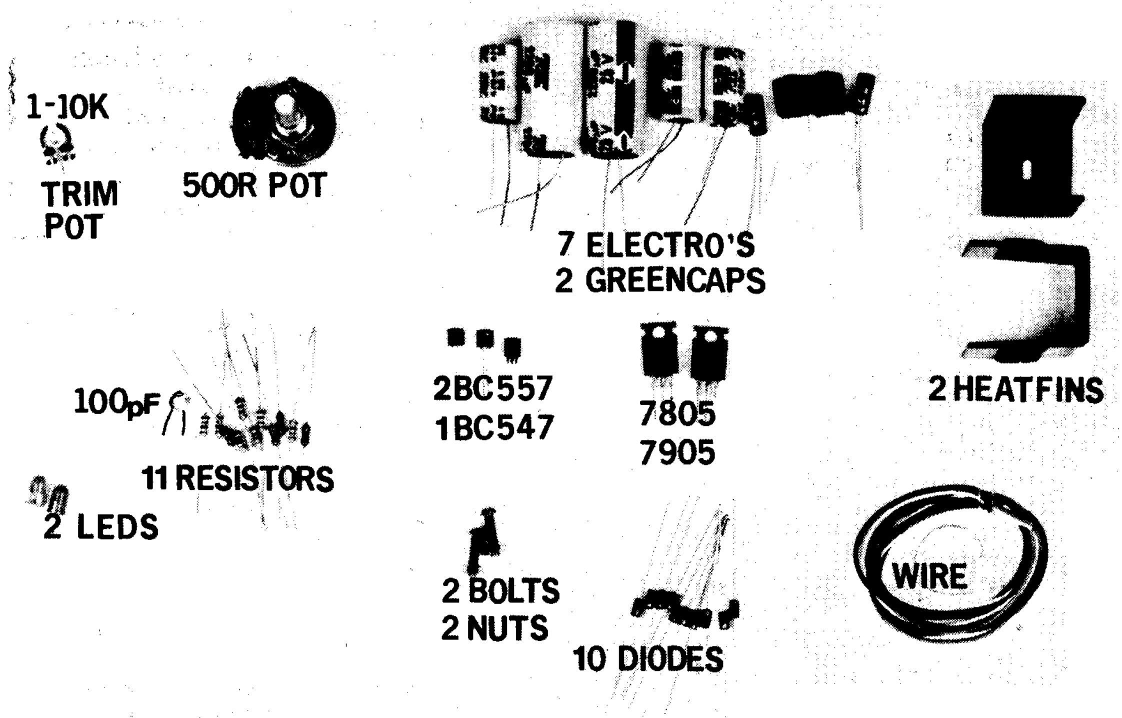
Our project fills the bill exactly. It provides all the necessary voltages for TTL and CMOS and incorporates the very important feature of having a positive and negative voltage of equal amplitude for op-amp investigation.

This has all been designed to fit on a simple PC board with only the minimum of wires to the front panel and power transformer.

One clever feature has been incorporated in the design to allow the positive voltage to reduce to reduce to zero when using a 5v regulator. This will be discussed in HOW THE CIRCUIT WORKS. The negative voltage does not track below -8v because the minimum operating voltage for most of the op-amps we will be discussing is ±9v and there is no advantage in the negative rail going below -9v.

The circuit has been specially designed to look simple and use readily available components. This makes the project very economical and possibly the only parts which will be new to you will be the 0 - 30v transformer and the 7905 regulator. The transformer can be duplicated in a number of ways but the 7905 is a negative 5v regulator and cannot be substituted with a positive type. In fact the lead-outs are different for positive and negative types so they cannot be inadvertently interchanged.

Let me not kid you. A dual supply has only very limited application. But when a split supply is needed, this one will be greatly appreciated. The most common need for a dual supply is when an op-amp circuit is being developed. They require a positive and negative rail in the



PARTS LIST

COMPONENTS FOR THE PC BOARD

1 - 220R ¼ watt
2 - 680R
2 - 1k
1 - 1k5
1 - 4k7
4 - 10k

1 - 5mm Red LED
1 - 5mm Green LED

1 - 100pf ceramic
2 - 100n ceramic
2 - 10mfd electrolytic 25v PC mount
3 - 470mfd electrolytic 25v PC mount
2 - 2200mfd electrolytic 25v PC mount

1 - 500R linear pot.
1 - 10k mini trim pot (can be 1k, 2k or 5k)

10 - 1N4002 diodes
1 - 7805 regulator
1 - 7905 regulator
1 - BC 547 transistor
2 - BC 557 transistors

2 - heat fins for regulators
2 - sets of nuts and bolts for heat fins
2 - sets of nuts and bolts for heat-sink
coloured hook up flex. 10 colours @ 20cm.
length of fine solder
small amount of thermal compound

ADDITIONAL PARTS

Transformer type 6672 or similar or 2 - 2155's
1 - knob for 500R pot
1 - SPDT power switch
1 - terminal strip for mains lead
3m - power cord (3-core)
1 - 3-pin plug (plug top)
1 - cord clamp
nuts and bolts as needed
3 - binding post terminals (1 red, 1 green 1 white)
3 - output leads (multistrand wire)
3 - coloured alligator clips to suit

HARDWARE ITEMS:

Parts for the cabinet include sheet aluminium, plyboard or particle board, nails, glue, screws, nuts, bolts rubber feet etc.

This will be a good fabrication exercise and don't forget to add holes or slots for the release of heat.

region ±*10 to* ±*15v.* Most op-amps will not work successfully on less than 9v and few can withstand voltages higher than ±18v.

To achieve a stable positive supply is quite an easy matter but a negative voltage is more difficult to obtain. You could use a set of batteries, another supply connected in reverse or build a miniature supply especially for the occassion. All this would tend to deter you from experimenting but now you don't have an excuse.

With this project you have the facility of producing both a positive and negative voltage and later in the TE programme we will be presenting projects with op-amp chips and other circuits requiring a split supply.
But don't let these forward applications deter you from building the supply now. It's construction is so economical that all its features will cost less than a normal single output arrangement.
We have used a very simple circuit and our first prototype operated first go without any need for modification.
The trim pot needed only a slight adjustment to equalize the output voltages and the supply was loaded to full current for a load test. Everything worked fine.

The main limitation with the supply is the power transformer. A 6672 is rated at 30v @ 1 amp and this means the output is equivalent to a 30v battery capable of supplying 1 amp for an extended period of time.
We found this transformer had a slightly lower rating but was adequate for the job. It has been chosen because it is cheap and can be used in three different modes to create 3 different output arrangements.
These are the three modes:

1. The 15v - 0 - 15v mode. This produces 0 - 15v output and -8v to -15v output.

2. The 0 - 30v mode. This produces 5v to 30v output. No negative output is available.

3. The 30v - 25v - 0v mode. This produces 0 - 22v output. No negative output is available.

15v - 0 - 15v MODE:

This arrangement provides dual output capability as described in the main section of this book. The PC board has been designed for this situation and the overlay allows for the connection of the 0v, 15v, 20v and 30v leads of the 6672 transformer.

Project cost will range from:
$40 - $60
depending on the type of cabinet used. Or as low as:
$25
if only the basic kit is built.

THE RATINGS

Positive supply:
0 to 15v at 1 amp.

Negative supply:
8 to 15v at 1 amp.

But:

When the positive supply is delivering 15v at 1 amp, the negative supply should only be delivering about 500mA. This is the limitation of the 6672 transformer. Other output arrangements are possible and are covered fully in the notes.

0 - 30v MODE:

Using the 0v and 30v leads will produce a power supply capable of delivering 5 to 30v. The current rating will be:
5v to 15v (approx) - 500 milliamps
20v(approx) to 30v - 1 amp.

The reason for the low current rating at the 5v end of the range is due to the 25v drop across the regulator. This will produce a power loss of 25 watts if one amp is required and this is in excess of the capability of the regulator. It will tend to shut down or fold-back when anything greater than 20 watts is being dissipated.

Note: This arangement is not suggested if 5v @ 1 amp is required.

30v - 25v - 0v MODE:

This will produce a 0v to 22v output and the current ratings will be similar to the above mode. The advantage of this arrangement is the very low voltage available for circuits using 1 or 2 cells in their supply.

HOW THE CIRCUIT WORKS

This is a brief analysis of the operation of the circuit. For a more detailed account, see the last few pages of the power supply notes in the back of the book.

The power transformer can be considered to be a single 30v winding feeding a bridge rectifier with the centre-tap becoming the neutral line. The cathode of the bridge diodes becomes the positive rail and the anodes become the negative rail. See diagram P 13.

The resulting voltage is smoothed by the 2200mfd electrolytic and this produces a DC voltage having a ripple less than 2v P-P.

From now we will discuss the positive section of the power supply and move to the negative section as the signal path takes us.
The DC voltage is fed to the 7805 through the IN terminal and it appears at the OUT terminal at a voltage which is 5v higher than the GND terminal. You will notice tha the GND terminal is not connected to the neutral rail but has a 500 ohm pot in the line. This is to enable the output voltage to be adjusted. The 220R resistor and 500 ohm pot form a voltage divider which "jacks" up the voltage to create the variable output.

These two resistors form a voltage dividing network and the voltage developed across the 500R pot is obtained via the 8 milliamps flowing in the GND line plus the current flowing from the 220R resistor. The actual voltage across the 500R pot can be obtained from a complex equation but for our discussion we merely say the pot increases and decreases the output voltage.

The output from the 7805 flows to a voltage dividing network made up of two 10k resistors and a 10k mini trim pot. One end of the divider is a positive voltage and the other end a negative voltage, having an equal amplitude. This means the centre point of the divider will be at a virtual earth or zero level. The 10k mini trim pot is included to make allowances for the slight difference in resistance of the two 10k resistors as they must be matched very closely for the positive rail to equal the negative rail.

The starting point for the biasing arrangement lies in the zeroing of the voltage on the voltage divider. This will mean the base of Q1 will be at a virtual zero level and the emitter will be about .55v higher due to the natural base-emitter voltage for a transistor which is only just beginning to be turned on. This voltage is transferred across to Q2 and causes it to be slightly turned on also.

The effect of the collector-emitter current flow in Q2 causes a voltage drop to be produced in the 10k resistor between the base and emitter leads of Q3. This voltage turns on Q3 and increases the voltage in the parallel combination of the two 680R resistors.
The -5v from the 7905 is increased by an amount determined by the 'turn-on' of Q3 and creates a voltage which is exactly the same amplitude as the positive voltage (or maybe a few millivolts less).

THE 0v FEATURE:

The positive voltage is capable of being reduced to zero and this has been achieved in a very interesting way.
Just as the voltage of a fixed regulator can be increased by jacking up the voltage on the GND terminal, the output can be reduced to zero by making the GND terminal go negative.

All the regulator "sees" is the voltage on the ground (GND) terminal. It then keep the output voltage 5v above this value if it is a 5v regulator. If the GND terminal is -3v, the output of the regulator will be 2v positive. If the GND terminal is -5v, the output will be zero.

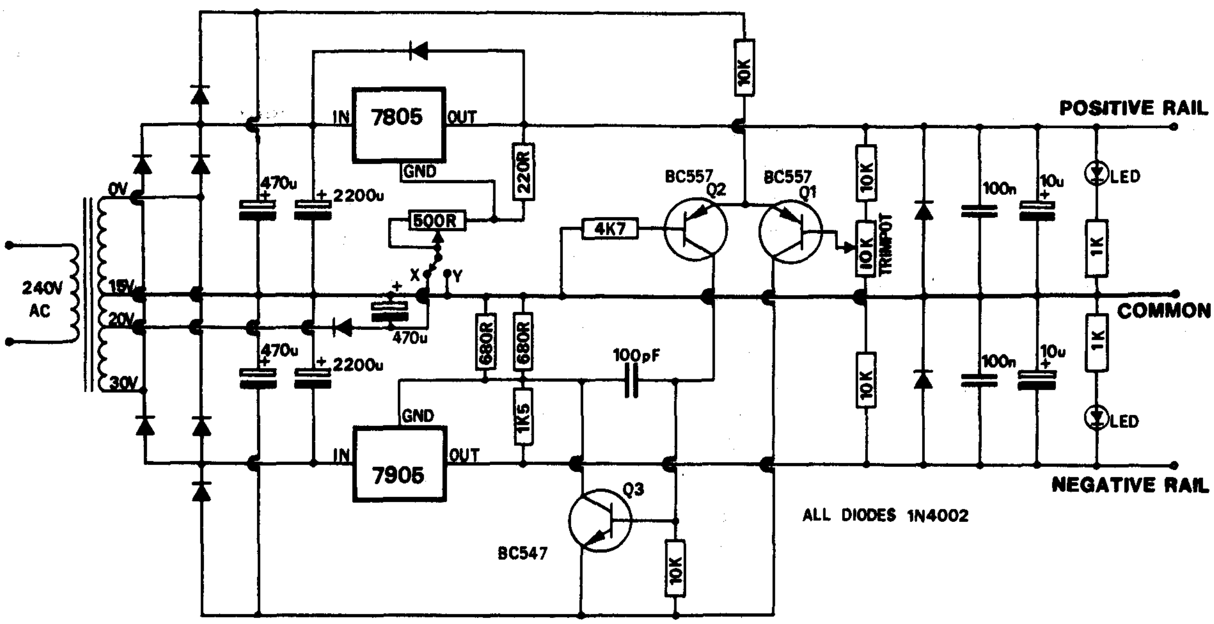
By using a tapping on the transformer which is 5v in the negative direction, we can add a simple rectifier and electrolytic to achieve a negative voltage. This is then fed to the GND terminal via a variable control to achieve a fully adjustable output.

SHORT-CIRCUIT PROTECTION

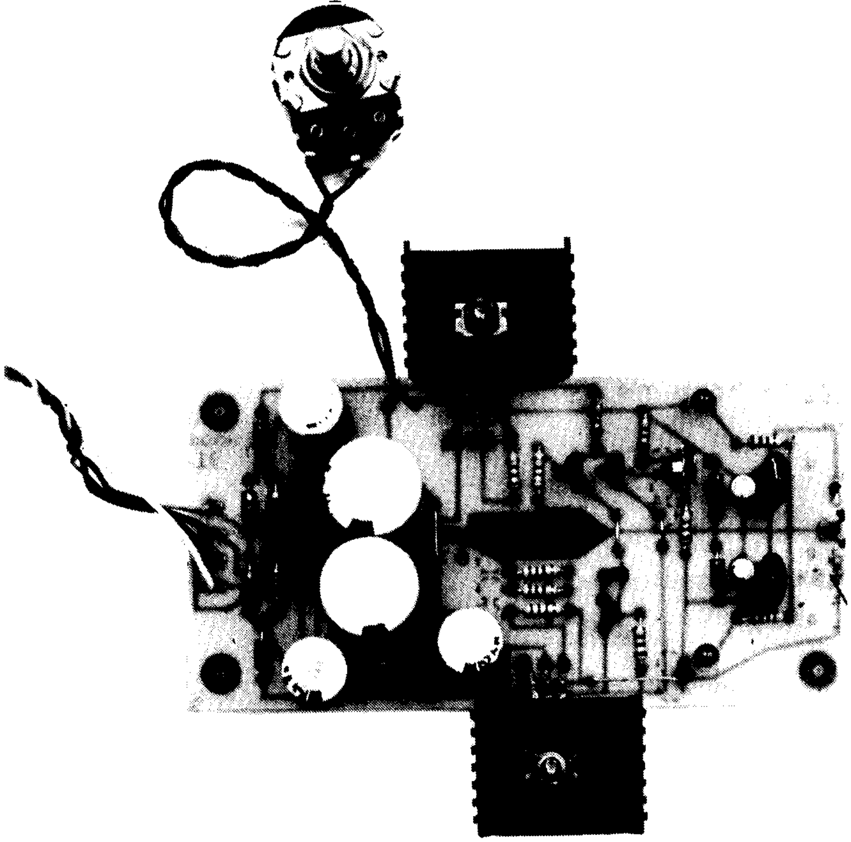
Two features have been included in the power supply to prevent the circuit being damaged by accidental short circuits.
The most important of these is the two diodes in the output. If the positive output is connected to the negative output, the regulators would be in a worse position than if they were shorted to earth. The internal circuitry in each regulator is diode isolated and if the output goes negative, the reverse biased diode becomes forward biased and the circuitry is destroyed.
To prevent this from happening we have added two diodes in the output. On short circuit, the top diode shorts the negative voltage to earth and the lower diode shorts the positive voltage to earth. And the regulators are saved.

The other safety feature is the diode connected across the 7805. This diode protects the regulator against shorts in the 2200mfd electrolytic from damaging the regulator. If the input voltage suddenly ceases, the output voltage will attempt to pass through the regulator in the reverse direction. The diode will reduce this to about .6v.

THE CIRCUIT



DUAL TRACKING POWER SUPPLY CIRCUIT



COMPLETED LAYOUT OF THE POWER SUPPLY

Note the relative position of each of the components corresponds to the circuit diagram.

The notes in this project book will mainly deal with the assembly of the PC board, wiring the transformer and heatsinking the positive regulator.

The PC board shown in the photo is our prototype on fibre-glass. The production run is paper bakelite and has an overlay showing the position of each of the components.

THE TRANSFORMER

A SHORT DISCUSSION ON THE TRANSFORMER AND HOW IT WORKS.

A power transformer is a brilliant piece of deception. It is much more complex than it appears. Nearly everyone takes a transformer for granted. They say "It works, so why worry."

But when you are designing a power supply, the transformer all of a sudden becomes a very important part of the design. Quite often the transformer becomes the limiting factor in the design and it may not perform as you had expected.
So it can be worthwhile to understand a few of its characteristics.

Take for instance, these three features:

Why is the core of a transformer laminated?
Why does an unloaded transformer take so little power?
How does the primary consume power according to the load on the secondary?

These are the points we will briefly discuss.

The whole secret of the operation of a transformer is the fact that it operates on alternating current. The AC mains.
So you cannot apply DC theory to its operation.
The fact that the primary has a DC resistance of 50 ohms does not come into any of our calculations at this stage.

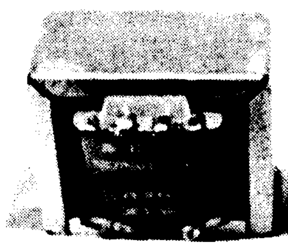
A transformer is basically 2 windings which are connected magnetically.
We will firstly deal with the primary winding.
When a transformer is connected to the mains, the core is initially devoid of any flux or any magnetism. If we assume the voltage of the AC is passing through the zero point on the graph, we can explain what happens as the voltage rises.

Each turn of the primary begins to have current flowing through it and produces magnetic flux in the core. This flux crosses each other turn and this induces a voltage in each turn which is in the opposite direction to the supplied voltage. At the same time the other turns produce a magnetic flux which cuts our first turn to produce a voltage in the reverse direction.
This all occurs at the same time and thus the mains voltage finds that the reverse voltage is neary as high as the incoming voltage. Take this example: If the mains impresses a voltage of 600millivolts on each turn, the revese voltage being generated by the flux will be about 585 millivolts. Thus only 15 millivolts is available for each turn to produce the flux. This voltage is impressed across a resistance of about 1/20 ohm (for each turn) and will produce a current flow which is the quiescent current for the transformer. This is just sufficient to produce the maximum flux lines in the core material.
It should be noted that the flux is also producing a voltage in the secondary winding but since this winding is not connected to a load, no current flows.

If a load is connected to the secondary, a current flows. This also causes a higher current to flow in the primary.

The reason for this can be explained as follows:

The secondary supplies current to the load becauses of the voltage it generates. The current flow is determined by the value of the resistance of the load. This current and voltage is a direct result of the flux passing through each secondary turn. The current thus supplied, decreases the flux density in the core of the transformer and the primary receives slightly less back-voltage per turn due to the lower flux.



THE 6672 TRANSFORMER

The result is a higher millivolt forward voltage and a higher current flows in the primary.

This effect can be increased to the limitations of the transformer. If the current-increases excessively, a point is reached where there is insufficient flux to maintain a constant output voltage.
The output voltage will "drop off" and regulation will be lost.
Regulation is the ability to maintain an acceptable output at the required load current.

Now we come to the question:
"Why is the core of the transformer laminated?"
The core has one basic function. To concentrate the magnetic lines of force so that they pass through the windings of the transformer. These lines are commonly called FLUX.

The core provides a magnetic path for these. They pass down the middle of the core and around each side of the windings.8y making the core out of thin sheets of metal we can produce a product in which the flux will perfer to travel in the longitudinal direction. In addition, the outside layers of the laminations are treated with an insulating material to make electrical conduction between sheets very difficult.

The reason for this is very interesting. Flux cannot differentiate between the core material and the copper of the windings.Whenever it passes through a metal it produces a voltage and according to the resistance of the material, a current will flow.

Take the core for instance. If the outside legs of the transformer were solid, the flux would pass through the outside leg and the material at the very edge of the leg, called the SKIN, would act as one large, thick, turn of wire. In fact it would act as one completely closed turn which we call a shorted turn and thus a heavy current would flow around the skin of the leg. The core material would get very hot and the transformer would cook.

By splitting the leg into lots of thin sheets, the outside path (or shorted turn) is now converted to the outside skin of each sheet but this time the generated voltage is much smaller, being only a fraction of the original due to the small amount of flux in the thin sheet. Thus the losses are reduced considerably by laminating the core.

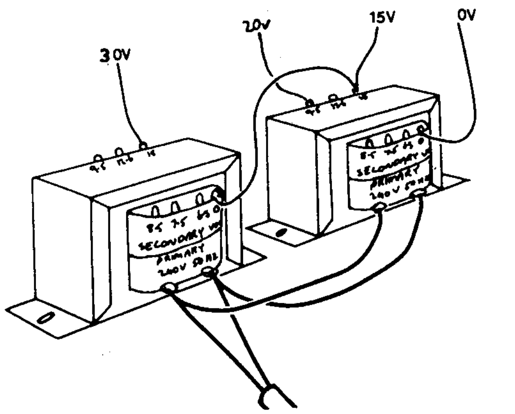
USING 2 2155 TRANSFORMERS

Two 2155 transformers can be used in place of the 6672 transformer with equal (if not better) performance.

A number of factors influence the ability of a transformer to supply the necessary voltage at full current.
In simple terms, this feature is called regulation and usually improves with the size of a transformer. Small transformers usually have poor regulation and for this reason it is best to use a single large transformer in place of two or three smaller units.

A single transformer is both lighter and cheaper than its equivalent in smaller units.

Thus the statement that two 2155 transformers will work as well as one 6672 is debatable.



PHASING TWO 2155 TRANSFORMERS

Two 2155's have some advantages and some disadvantages. Here they are:

ADVANTAGES:

1. Two 2155's are smaller and easier to mount.

2. They will distribute the load in the cabinet into two areas.

3. They will provide more area for cooling and ventilation.

4. 2155's are readily available.

DISADVANTAGES:

1. Two 2155's will cost more than one 6672.

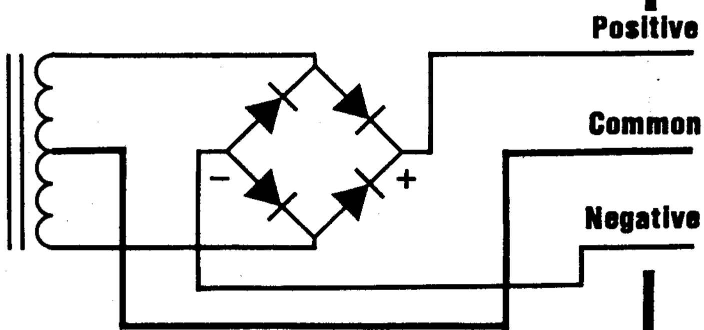
2. Two 2155's will need additional wiring and phase determination.

PHASING THE 2155's.

Two 2155's will need to be phased so that the output voltage is presented to the PC board as a single 30 volt winding. If the phase is correct, the voltage between the 0v tap on one transformer and the 15v tap on the other will be 30v. If the phase is incorrect, the output voltage will be zero across these terminals.

To produce the correct phasing, the 240v inputs on each transformer are connected in parallel as shown in the accompanying diagram and the secondary terminals are connected in series. We are assuming that both transformers have been manufactured with their primaries wound in the same direction and also the secondaries.

Whether you are using one transformer or two, the circuit can be simplified to a single centre-tapped secondary feeding into a bridge...as shown in the diagram.



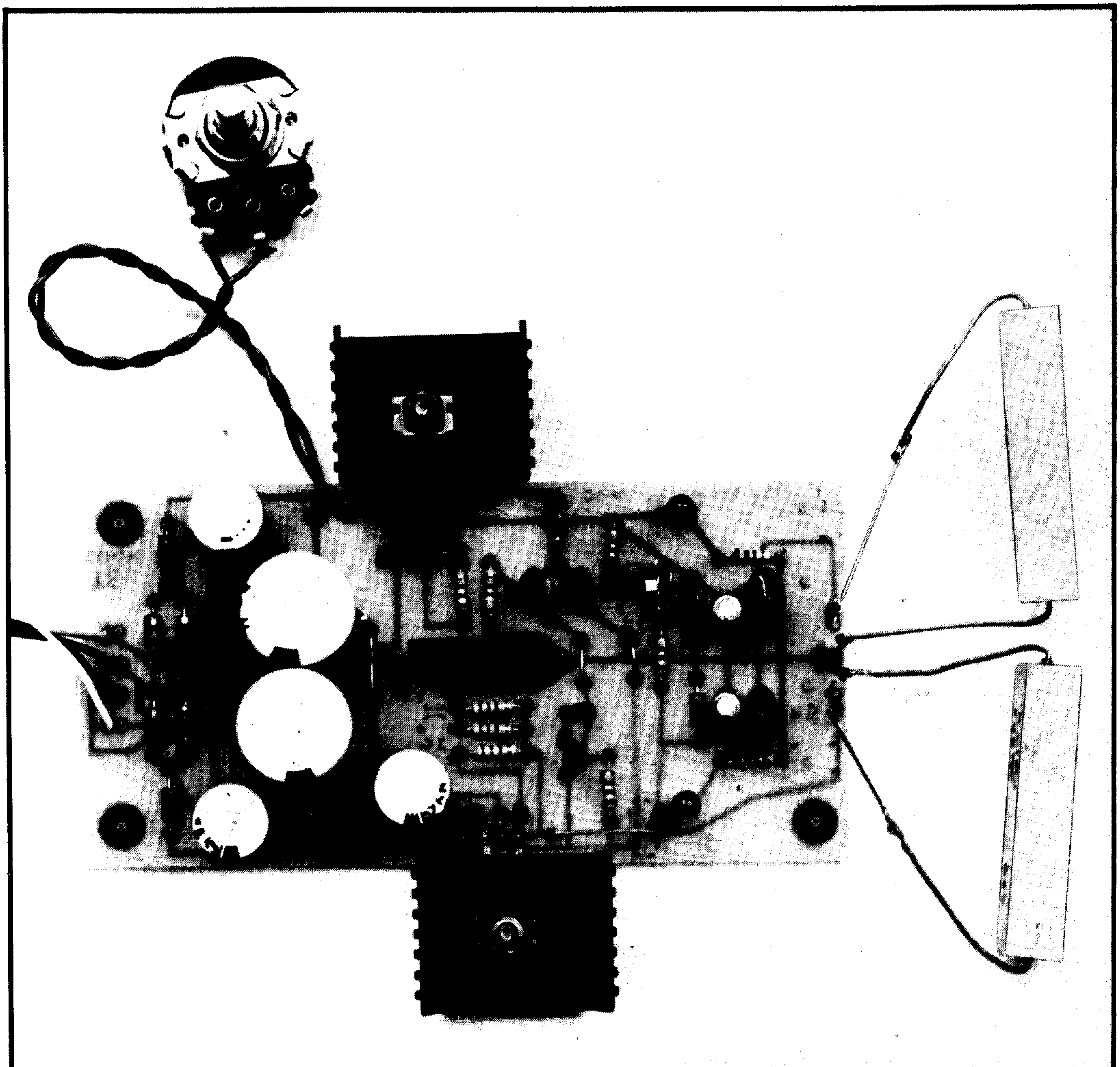
OUR FRONT-END

CONSTRUCTION

Before warming up the soldering iron you must get all the components organized. If you buy a kit, all the parts will be pre-selected. They will be the right size and shape to fit the board. If you intend to purchase the components from different sources and use some of the parts from your stock, you may fall into a number of traps. The most common of these is non uniformity of components. To me, neatness is very important. All the resistors should be the same size and all the electrolytics single ended types (PC types) and the mini trim pot should be small enough to fit directly into the holes.

Make sure the leads of the components fit into the holes on the board before starting to solder. This applies to the diodes, mini trim pot, regulator and large electrolytics. The resistors are all ¼watt. None of them will get hot when the power supply is in operation so tiny resistors are quite suitable.

Gather all the components and lay them out so that they are in a position exactly relative to the final placement on the board. This is very important because when you are soldering,



you cannot concentrate on selecting the correct component and make a good connection at the same time.

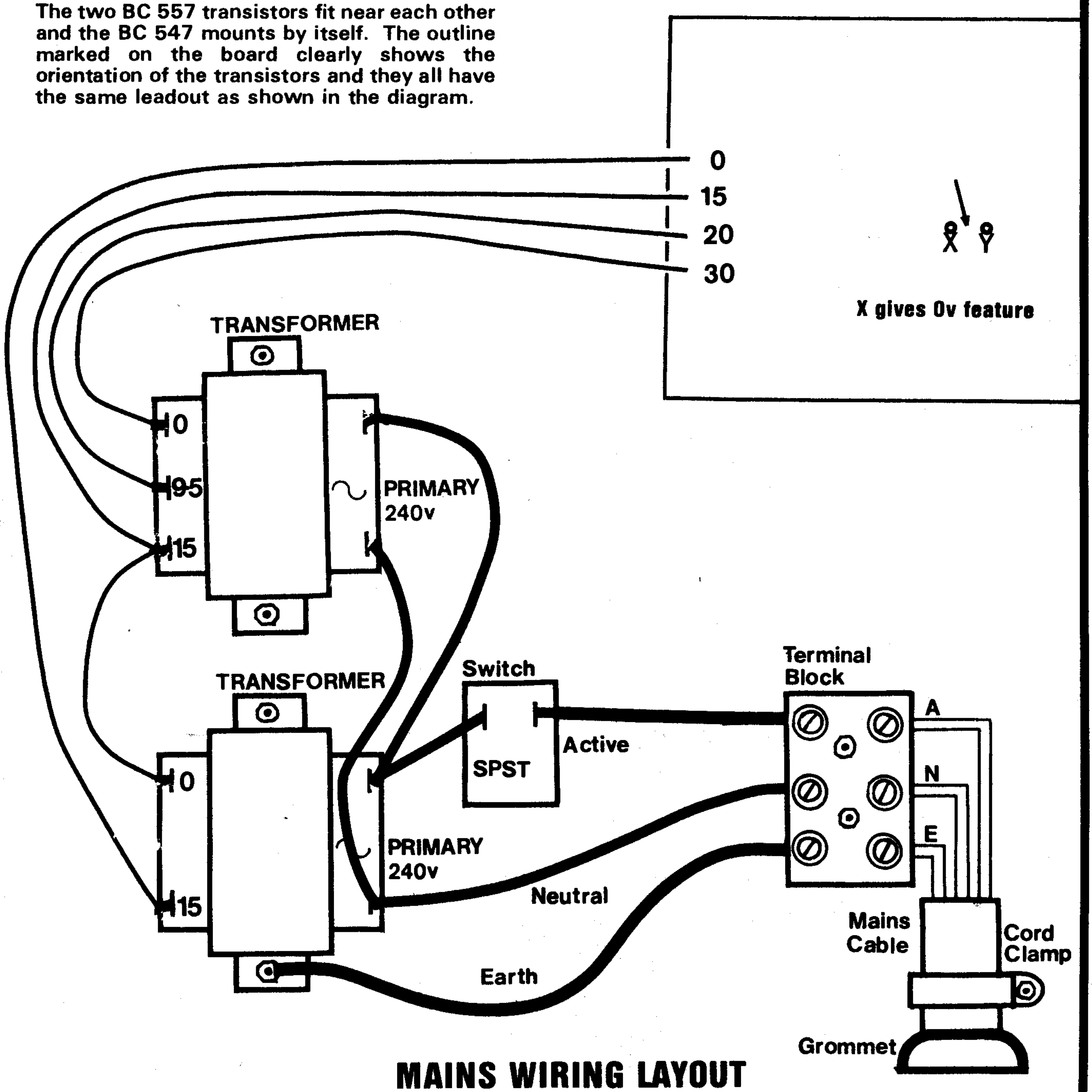
Make sure you understand which end of the diode is the cathode before starting. This also applies to the electrolytics. We have identified the positive on the layout but the negative is usually identified on the body of the component. The metal side of the regulator faces towards the edge of the board so that it can be bolted onto a heatsink via a nut and bolt and a little thermal heat compound.

The two BC 557 transistors fit near each other and the BC 547 mounts by itself. The outline marked on the board clearly shows the orientation of the transistors and they all have the same leadout as shown in the diagram.

The two light emitting diodes must be inserted as shown as they are a polarised component and will not work in the reverse direction.

Now we are ready for assembly. Use fine or very fine solder for the joints and tin the soldering iron ready for the start.

How you start or where you start is not critical but the most important point is to make a very neat job.



MAINS WIRING LAYOUT

SELECTING X or Y

The board contains 3 jumper wires and these are clearly shown on the overlay. On top of this, in the centre of the board, is a selection link. It is actually a jumper which enables you to select the 0v to 15v mode using the X hole or 5v to 15v using the Y hole. We suggest using the X hole to give the fully variable output. This makes a total of 4 jumper links.

Next add the 10 diodes. The white band painted around one end is the cathode and it goes upwards or to the left on the board. Push the leads through the holes so that the diode touches the board. This will improve the heatsinking through the copper tracks on the board. Bend the leads slightly apart so that the diode stays in position, ready for soldering. Solder the leads and cut the surplus flush with the board. Continue with the rest of the diodes.
Eleven resistors are needed for the circuit. They are fitted one at a time, soldered and the leads trimmed flush.

The mini trim pot can be any value from 1k to 10k and the best type to buy has thin leads which will fit down the 1mm holes.

The 100pf cap is the next part to be soldered to the board.

The two light emitting diodes are fitted so that the cathode lead (the short lead) fits down the hole identified with a line.

The two 10mfd electrolytics and 100n capacitors are the next components to be fitted to the board.

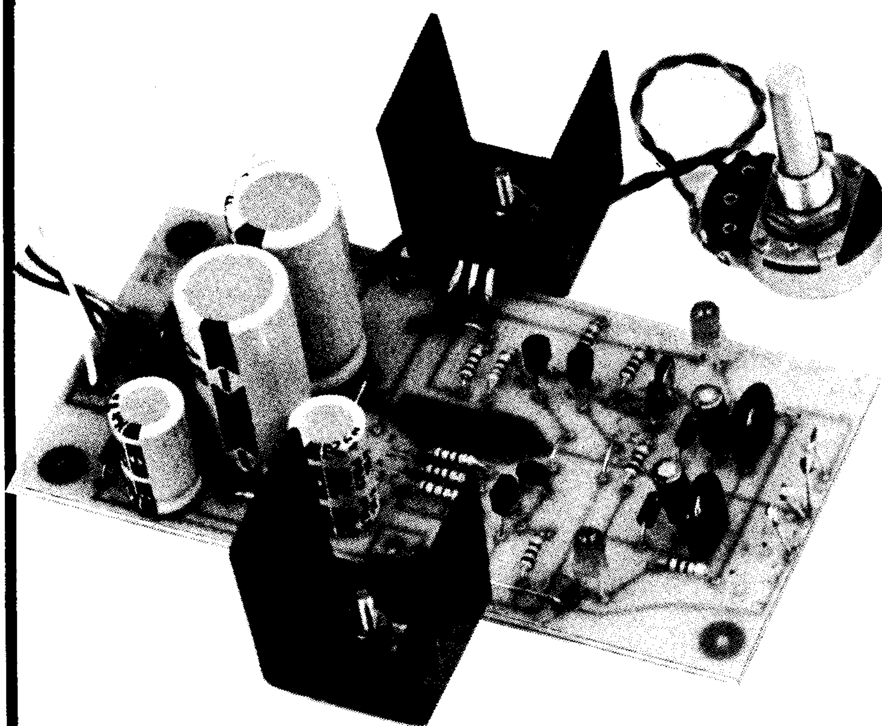
The 7805 and 7905 are soldered in position with the metal face near the edge of the board.

The two 2200mfd electrolytics are now added to the board, make sure the positive lead goes down the correct hole. Remember that the negative lead is identified on the body of the electrolytic.

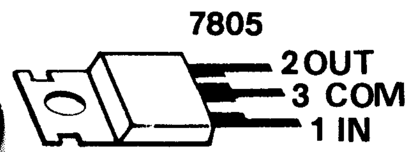
The three 470mfd electrolytics are the last components to be added to the board.

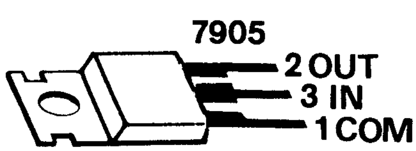
Cut short lengths of coloured hook-up wire for the 4 input leads, 2 leads for the 500R pot and 3 leads for the output lines. These should be long enough to allow the board to be taken out of the case for servicing, if needed.

The remainder of the construction will vary from constructor to constructor and the easiest way to present the wiring is via a diagram. We have shown the layout for a pair of 2155's as the 6672 transformer will be a direct connection.



REGULATOR PIN-OUTS





5v 1.5 AMP

For the special requirement of 5v at 1.5 amps, this power supply project requires a small modification.

You must firstly be aware that the dual voltage facility will not be available nor will the variable voltage be easily obtainable. We are catering for the situation where exactly 5v is required for a TTL project or other similar needs.

The 1.5 amp current situation is only recommended for short periods of time as the 6672 transformer is only designed for 1 amp loading.

To enable the bridge rectifiers to handle 1.5 amps, it will be necessary to parallel up two of the diodes to cater for the higher current.

You can use two extra diodes out of your parts-box or remove the two outside diodes from the board as they will not be needed. There are the diodes which supply the voltage to the transistor stages.

Refer to the diagram for the position of the two diodes which must be paralleled up. The two added diodes can be soldered to the copper side of the board or the top side as shown in the diagram. Either way you will need to take advantage of the heatsink and the leads must be keps as short as possible so that the heat can be effectively dissipated.

Keeping the diodes cool is most important when the maximum current is required.

The main electrolytic can also be increased to give added smooting but this not as important as attending to the diodes.
Remember the regulator must be allowed to have at least 3v across it to maintain regulation. The regulator must also be perfectly heatsinked with thermal grease and mounted on a large heatsink to capable of supplying the necessary wattage.

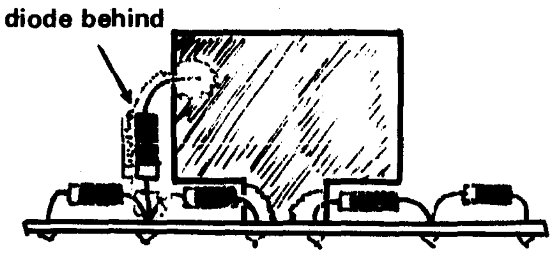
HEATSINKING THE DIODES

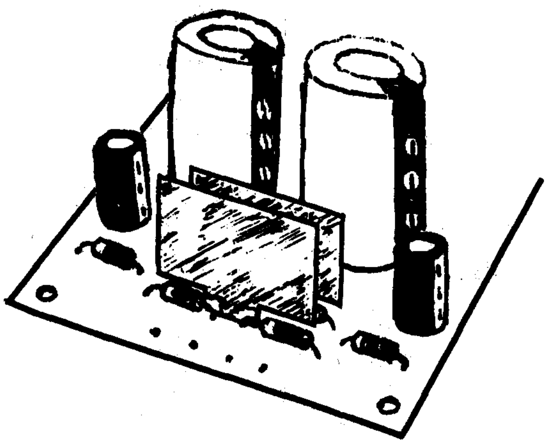
For any current requirements above 1 amp, you will find the power diodes will be fairly warm to touch after a few minutes of operation.
This is because they will be dissipating nearly 1 watt of power per diode because the voltage drop across the junction rises to nearly .9v as the current rises to about 1 amp.

If you intend to operate the power supply for long periods of time at a farily high current level, the diodes must be heatsinked. This will not only lengthen their life but also prevent the possibility of one or more of them becoming damaged.

The best heatsink is a small piece of sheet-metal cut from the lid of a can. This is soldered directly to the leads of two diodes as shown in the diagram. You will see that this is not creating a short circuit as the copper track under the board also joins these two diodes.

Another piece of sheet-metal can be used to heatsink the other two diodes and these can be stood upright and parallel to each other. To prevent them from shorting against each other, a small piece of insulating tape can be stuck on the inside of each fin. The diodes will love you for the attention they receive and if your are quick at soldering, you will not damage them.





HEATSINKING THE DIODES

HEATSINKING THE REGULATORS

The photos of this project show heat fins on the two regulators. These heat fins are only intended as run-up components, to test the supply and get it operating. Each fin will only dissipate 5 to 6 watts and this well below the expected dissipation when the supply is delivering 1 amp.

To act as a suitable heat sink, I looked around for something everyone would have in their parts box. I found it in the form of a lid from a jiffy box which is the same size as the PC board for this project. This enables it to match perfectly with the holes in the PC board.

Only the positive regulator needs an additional radiating fin as the current required from the negative line will be small and the mini heat fin will be capable of dissipating any heat generated by the 7905.

To enable the sheet of aluminium to attach to the PC board, it is necessary to make a cut 1 cm from each end and cut into the sheet 1 cm to make a tab containing the corner hole. This tab is bent to 90° and the same is done with the hole in the other end. The strip of aluminium between the two tabs is bent to 90° in the opposite direction. See the sketch.
Sit the aluminium sheet on the PC board and make sure the holes align. Next mark the aluminium with a scriber through the mounting hole of the regulator and drill this hole in the sheet to take a short 4BA nut and bolt.
After the hole has been drilled, use a flat file to remove the filings and prepare the face of the aluminium near the hole to meet the surface of the regulator. Do this on a flat surface and repeat on the other side. The mini heat fin will be used in conjunction with the aluminium sheet to prevent a hot spot from developing near the regulator. The aluminium sheet itself is not thick enough to transfer the heat quickly from the face of the regulator and an additional thickness is required. Thus the need for the heat fin.

Prepare the surface of the heat fin with the flat file and sit it onto the aluminium sheet to observe their flatness. You should not be able to see any light through the crack . . the closer the fit the better.

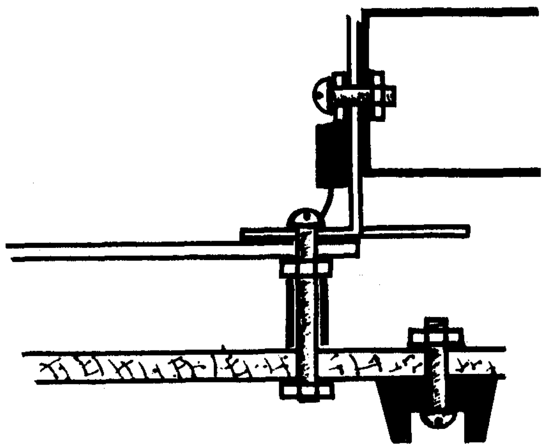
Attach the aluminium sheet to the PC board with two nuts and bolts. Smear a very small amount of thermal compound on the four conducting faces. Clamp the regulator and mini heat fin to the aluminium and screw them up tight so that the thermal compound squeezes out of the cracks.

You have now produced the cheapest and best heat fin for 10 watts dissipation. It will probably dissipate more heat but 10 watts is about the maximum we are producing.

It is an interesting exercise to load up the power supply and feel the heat fin after 10 or 20 minutes of operation. With 10 watts of heat being generated the fin is quite hot but still able to be touched. You can sense the temperature the regulator the temperature is so high that you might wonder how the regulator can still be operating.

Without the mini heat fin you run the risk of the regulator overheating and shutting down. So it forms a very important part of the cooling.

We have only added a large heat fin to the positive line as the negative rail will be supplying only a very small current under normal operation. If you find the heat fin on the negative rail gets too hot, you can add an additional radiator at a later date. Anything up to 150 milliamps on the negative line can be delivered without overheating the regulator.



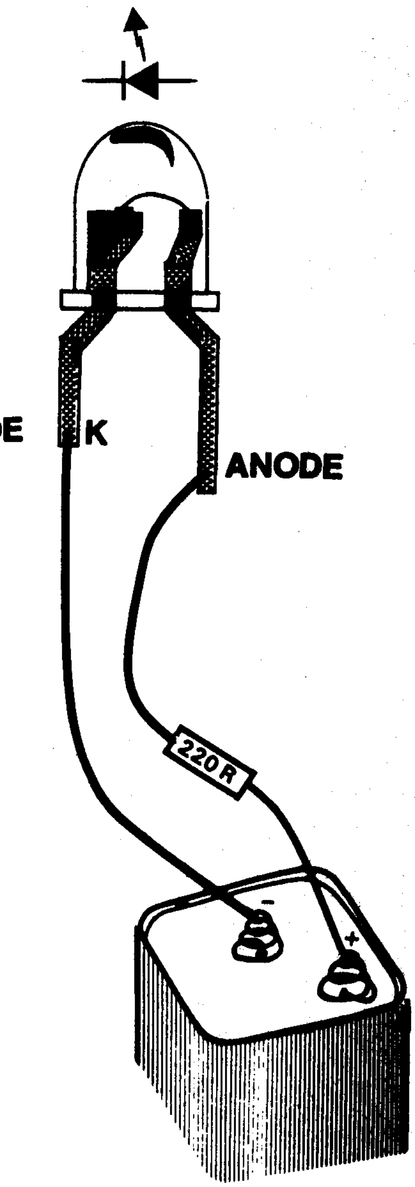
One possible arrangment for the regulator heatsink and mini fin is shown in the diagram.
The PC board is attached to the wooden base via a long screw and sleeve. This screw also holds the heatsink in position. Note the heatsink is bent to 90° at its base to produce a foot which can be clamped to the board.

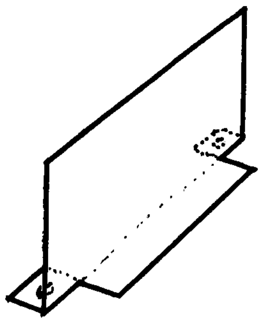
IDENTIFYING THE LEDs

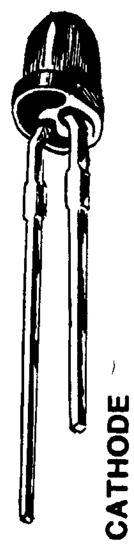
IDENTIFYING THE LEDs.

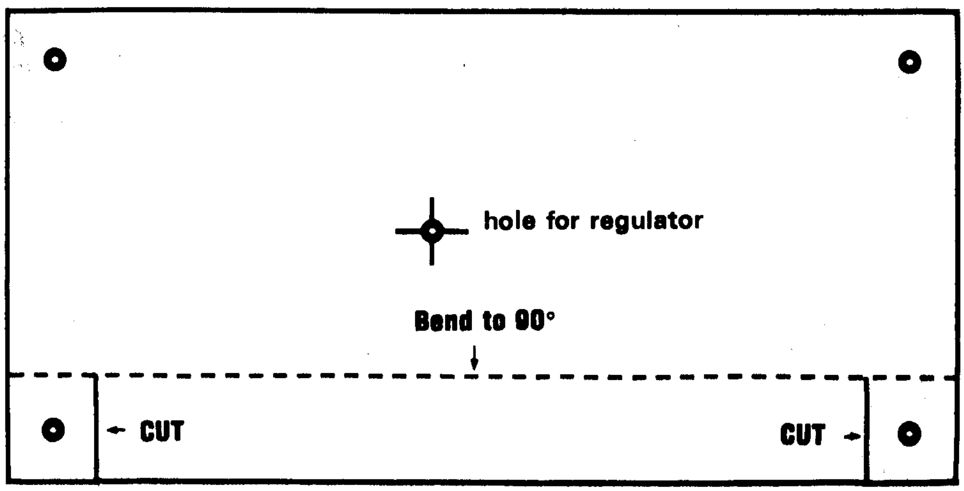
The two light emitting diodes are mounted on the board in opposite directions. There is no intentional reason for this, just circuit layout. If you are using new LEDs, one lead will be longer than the other. The short lead is the cathode. Otherwise, if the leads are the same length, you will have to use one of two other methods for determining the cathode.
Mind you, they won't be damaged if you insert them into the board around the wrong way. It's just nice to get things right the first time.

When soldering the LEDs to the board, do it quickly and keep the leads long. LEDs can be very easily destroyed by heat. But before soldering them, look into the body of the LED. You will se one lead is larger than the other. This is the cathode. The fine piece of wire joining the two leads is not a filament, it connects the leads to the light emitting crystal. The other methode of determining the cathode is to look for a flat on the side of the LED. It indicates the cathode.









SUBSTITUTES

The components in this project are really not as critical as you think. You will be pleasantly surprised when I tell you the range of values you can use.
I will start at one end of the board and work across.

The diodes can be any voltage from 100v to 1000 volts. Only the current rating is important. They must be capable of handling at least 1 amp. The 4 diodes in the bridge are the critical ones. They will be required to pass the current when the supply is operating. The other diodes pass a smaller current or are only used for short periods of time.

The two 470mfd electrolytics which smooth the supply for the transistors are mounted near the set of bridge diodes. They can be 100mfd, 220mfd or 330mfd, without affecting the performance.
The 470mfd producing the negative voltage cannot be reduced in value without affecting the negative supply to the regulator.

The 2200mfd main reservoir capacitor can be reduced to 1000mfd if the higher value is not available. If only one 2200mfd is available, it should be used for the positive section.

The 7805 can be replaced with a 7806 or 7808 or even a 7812, however it may not be possible to reduce the voltage to zero with the 12v type.

The 7905 can be substituted with a 7906 or 7908 and the same conditions apply. It will not be possible to get lower than 2v more than its rating.

The 220R resistor can be substituted with a 180R 270R or 330R.
The 4k7 can have any value from 2k2 to 47k.
The two 680R resistors can be changed to 560R or 820R.
The 1k5 can be 1k2 or 1k8.
Any of the 10k resistors can be changed to 8k2, 12k or 15k. The only important point is the two resistors in the voltage divider network must be the same value so that the mini trim pot has a chance of trimming their value exactly. The mini trim pot can be any value between 1k and 10k, however the higher value will make adjustment more sensitive.

The two 1k current limiting resistors for the light emitting diodes can be 820R, 1k2 or 1k5.

The 10mfd electrolytic in the output can be 4.7mfd or 22mfd without any effect on the performance.

It boils down to any of the components can be changed to a value higher or lower than the suggested value and the only components which are critical are the two resistors in the voltage divider network.

IDENTIFYING THE TRANSISTORS

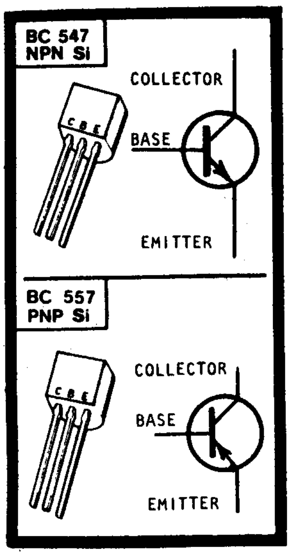
Both PNP and NPN transitors are used in the tracking circuit.

They both look exactly the same and have the same pin connections.
But you cannot swap them over or used them around the wrong way and expect the circuit to work. Because it won't.
Use only NEW transistors as they are so cheap to buy that it is not worth spoiling the project for a few cents. We use BC 547's and BC 557's for nearly all our projects and it will be worth while stocking up on a quantity of them if there is a saving for bulk lots.

HOW TO TEST UNMARKED TRANSISTORS.

If the numbers have rubbed off your transistors, they can be identified by using the transistor tester on the LOGIC DESIGNER or you can use a multimeter and the instructions on DATA SHEET 2.

When clipping to the leads, rmember that they are normally bent to form a tripod and this identifies the collector, base and emitter. If the leads are in-line, you will need to go by the flat on the transistor to pick out the collector. The holes in the board form a triangle and the outline of the body is also shown, so the transistor can only be inserted one-way-round.



THE MAINS

It cannot be too strongly stressed. This is a mains operated project and a great deal of caution is required when connecting any type of project to the mains. It is so easy to make a mistake in the wiring of something simple. Something like wiring a plug top, switch or socket. If this happens on the mains side of things, the result could be lethal.

And we always tend to make the greatest number of mistakes when something looks easy.

It's a simple truth. The easier the job, the more chance of making a mistake. This is because we electronics experts are complex human beings and when we are required to handle a simple operation, we let out guard down. And what happens. We do it wrong.

Whenever I do something simple I get it checked. And quite often I have made the simplest of mistakes.

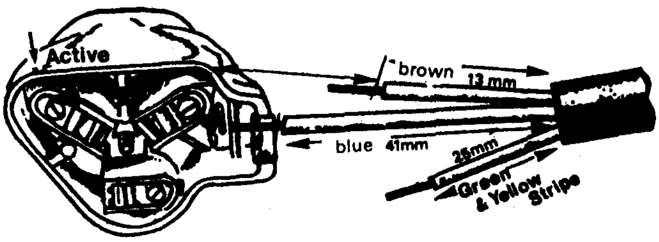
So the lesson is there.

This project requires checking.

It is absolutely essential to get the finished project checked by someone else. Anyone else. As long as it is not yourself doing the checking. You cannot proof your own work.

The plug and lead should be firmly connected via the wiring and the outside cover of the plug-top attached to the base. This is only simple intelligence but it is amazing how often the cover is left loose. The outside cover of the flex should enter the plug-top as far as possible so that no wires are visible.

WIRING A PIGGY-BACK PLUG



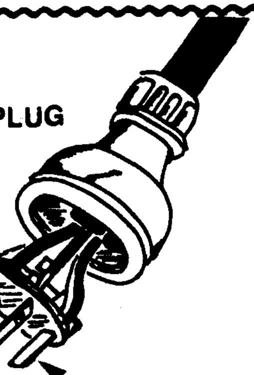
It is best to use a piggy-back plug so you can still use the mains outlet for your soldering iron or other lead.

A good cord anchorage will be needed where the cord enters the cabinet so that the lead does not pull on any of the wiring inside the case. The hole for the flex should also be large enough so that the lead does not rub against the metal hole in the case.

WIRING A PLUG-TOP

This simple operation looks fool-proof but needs concentration to avoid a mistake, because the new colour-code for the flex is so confusing.

Begin with the GREEN or YELLOW striped lead to the EARTH or plated pin. The RED or BROWN goes to the left-hand ACTIVE pin and the BLUE or BLACK to the right-hand or NEUTRAL pin.



Ideally an electrical engineer or technician should look over the mains side of the wiring to see if everything is tight, neat and well-insulated. But if this is not available, a teacher, parent, adult friend will be willing to follow the layout and tug at the flex and wiring.

Every part of the mains wiring should be insulated to the touch and no exposed wires should be visible. Only a screwdriver should be able to undo the screws on the terminal block.

Preferably the cord anchorage should be mounted inside so that the hole is prevented from chaffing the cord.

Provide plenty of flex for the project because you never know where it will be used. A long lead is always appreciated.

RUNNING-UP THE POWER SUPPLY

This secion deals with loading the power supply to rated current to check all components.

When all the parts have been mounted on the board, the next stage is to load the output so that about 1 amp is drawn from both the positive and negative rails. This lets you know the power supply is operating and will supply a current. More than that we are not testing.
We are only performing a run-up proceedure and the current will not be required for more than about 10 seconds.

The easiest way to load the supply is to connect two 10 watt resistors to the output. These can be either 8R2 or 10R and you will need a voltmeter to set the voltage fairly accurately.

Lay the project on the bench as shown in the photo and have a multimeter ready to make a very quick voltage adjustment.
Turn the supply ON and set the voltage to 8.2 volts for the 8R2 resistor or 10 volts for the 10 ohm resistor. By simple ohms law we have created a current flow of 1 amp in the output.

We are now going to make a very approximate statement. The power loss in the load is equal to the loss in each of the regulators. This is due to approximately half the voltage dropping across the regulators and half across the load. This means they will heat up to about the same temperature given the inference that the volume of the load is about the same as the regulator and heat fin.

In any case the run-up procedure calls for the supply to be turned on for 10 seconds and then switched OFF. Feel the temperature of each load resistor. They should feel about the same. Feel the two heat fins. They should also be about the same temperature.

This is only a simple test to test the outputs under load.
If you keep the power applied for more than 30 seconds you will burn your fingers on the load resistor, just as I did now. The heat fin will also get extremely hot and may exceed the normal limits of the regulator.

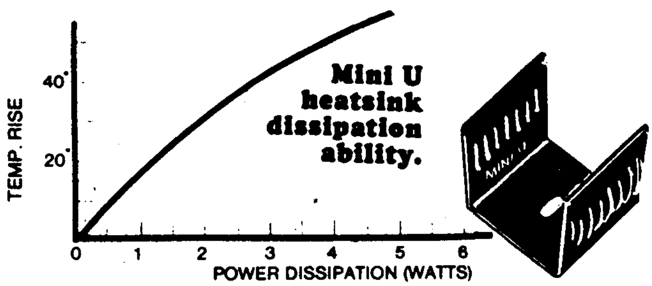
If any of the four heat-dissipators remains cold, you will need a multimeter to find out what is going wrong.
When you are testing the section containing the cold component(s), do not let the other section get too hot. Disconnect the load from the working stage and concentrate on the faulty section.

Firstly check the input voltage to the regulator, then the 'jacked-up' voltage on the GND connection. Finally measure the output voltage. If the input voltage is missing, it could be two faulty diodes, a dry joint or a break in the PC track. If the GND voltage is zero, the output will be only 5 volts and this could account for the cool resistor. If the GND terminal is jacked-up, and the output is zero, the regulator could be open.

If the regulator is getting very hot and the load is cold, it will be a short across the output, but not a short inside the regulator as it is a series-pass device and the shorting current must flow to earth. You can also refer to the notes dealing with a damaged power supply, later in the project.

Hopefully you will not meet any of these problems and the 4 heat-dissipating components will be nice and hot.

You are now ready for the next stage which is mounting the board and transformer in a case, cabinet, box or chassis.



CALIBRATING THE OUTPUT

The diagram opposite shows a suitable layout for the front panel. The scale for the output voltage is created with the aid of a multimeter set to a suitable voltage range.

Connect the voltmeter across the output terminals and turn the pot to read 5v on the multimeter. Mark the position on the scale.

Increase to 10v and again mark the position. The highest voltage will be about 15v and this is also marked on the dial.

You can include a panel meter to monitor the output voltage if you wish however the output voltage is generally not too critical and a scale around the pot is usually sufficient. You can use a multimeter to read the exact voltage as required.

COMPLETING THE POWER SUPPLY

Although we have said the layout of the case will be left up to you, I think a few pointers on good layout and design will be in order.

To pass a good design award, you will need to consider three aspects:

1. Perfect safety and insulation from the mains.
2. Good ventilation.
3. Good layout for the front panel.

First and foremost is the safety aspect.
If you are going to fabricate your own case, you have a choice of three materials: METAL, WOOD or PLASTIC. You can either use one material throughout or combine two or more together.
Sheet aluminium is the easiest metal to work with however wood is most versatile and the most pleasing when completed. Plastic will be the most difficult to work with and the most expensive to buy as sheet PVC or perspex is very expensive when bought in cut sizes.

You may like the idea of making wooden ends and bending up the base and front panel from a piece of sheet aluminium. The back and top can be made from another sheet. This will give a join at the top front edge and to avoid any joining problems you could make the base and back in one section and the front and top in another section. This will take the join to the back of the cabinet.
When using metal for the base and front, all the terminals, PC board and LEDs have to insulated from the metal. The use of wood throughout will eliminate this.

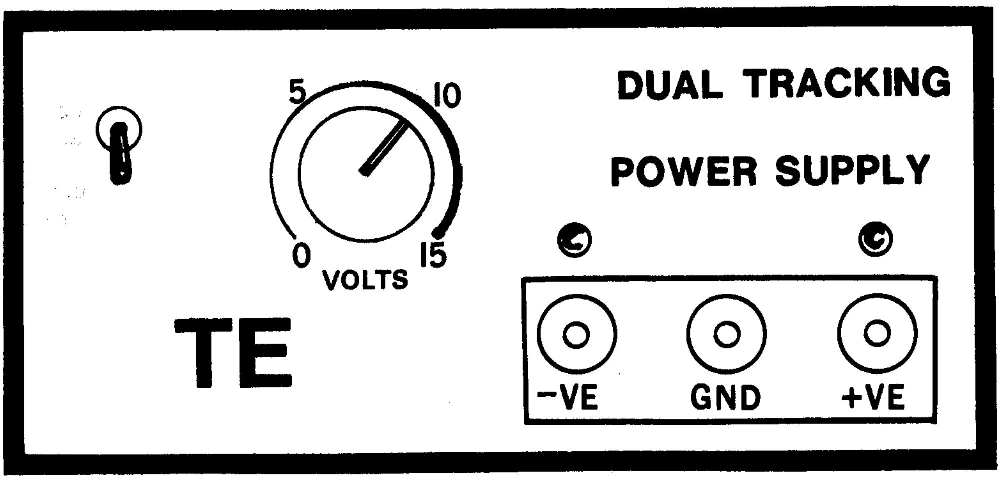
Any way you do it, it will come out cheaper than the over-priced cabinets on the market.

The most important design feature of a good cabinet is ventilation. The transformer and heatsink must have a draft of air passing through the base and out the top or near the top. Without this, the parts will gradually heat up and heat up until they finally cook. It may take a few hours but it will eventually occur and in the process you are exposing the case to the risk of fire.

Apart from an abundance of holes, you will need to have clearance between transformer and case of at least 1 cm and the top of the heatfin should be 1 cm from any case-work. The top of the heat-fin can be bent over to create this clearance if necessary. The PC board should be mounted on stand-offs so that the track-work does not short against the base.

Depending on your selection of one or two transformers, the internal layout should closely follow the layout diagram. It should be kept neat and tidy with the mains side of the project well away from the secondary voltages. No exposed leads should be visible and no connections to the mains available to the touch. Even a piece of electrical tape or a plastic sleeve is sufficient to prevent accidental contact. This is absolutely essential as some constructors say "I made it, so I can fiddle with it." and they become a little blase.

When laying out the front panel, centralise the controls and terminals to make them look appealing. Use the appropriate colour for each output and make sure it is adequately labelled. Don't forget, someone else may be using your supply one day and an unlabelled supply may lead to an accident.



A SUGGESTED FRONT PANEL LAYOUT

USING THE POWER SUPPLY

I shouldn't need to give you too many ideas about the uses for the power supply. By now you will be rearing to put it into action - even before it's complete.

I can only say it will be the next best investment after buying a set of nicad batteries. Not only will it last longer than rechargeable batteries but you won't have the worry of a project dying in the middle of development for want of a little juice.

The only thing I must say to you is: do not short the positive terminal to the negative terminal, "to see if the power supply is working." This is the most dangerous act of all for a dual supply and even beats shorting an output to earth.

When you short the positive to negative you are actually trying to make the output go below zero voltage and this will turn on an isolating diode inside the regulator. Without the two output protection diodes you would damage both regulators. But these diodes are not all that robust. With the high current and the added zap of an electrolytic, one of them could be damaged and go 'short'.
A tiny 10 ohm resistor connected on the outside of the supply to the negative terminal will prevent any likelihood of this happening and the low voltage drop across it, when drawing a few hundred milliamps, will be small.

Before connecting the output leads to any project or even a set of nicad cells, it is most important that the polarity of the leads be correct.

The output terminals should be colour coded with red for the positive output, green for the common line (or earth line) and white for the negative rail.

Use leads of a similar colour code and also matching alligator clips.

With the power supply switched off, connect the zero volt line (the common or earth line) and then the positive line. Switch the power supply ON.

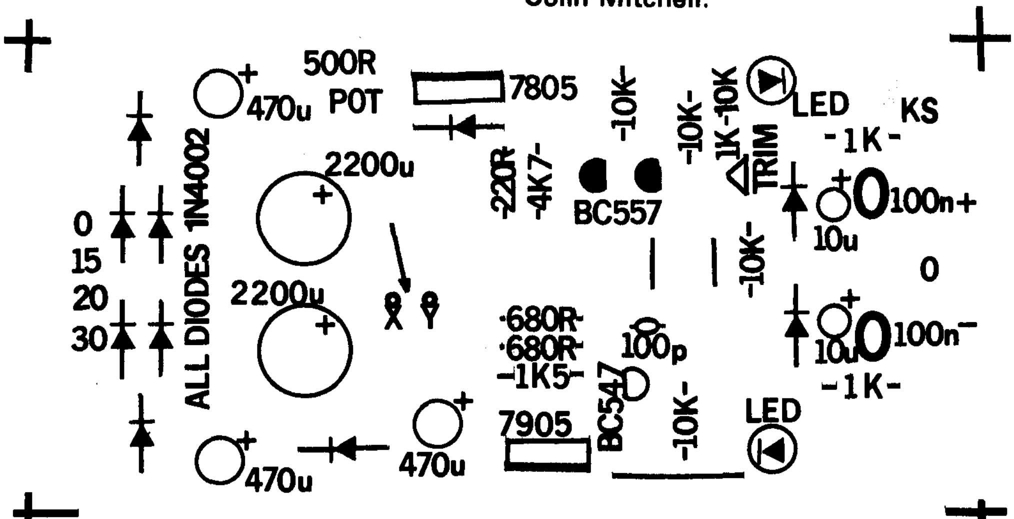
If you think a project you are powering may have a short circuit or consume too much power, the only protection device I have used with success is a small 10 ohm resistor of ¼ watt rating, in one of the lines. It doesn't matter which line contains the resistor, so long as it is well away from any other components as it may go up in smoke and catch fire.

If a short is present or occurs, the resistor will go up in smoke. If the project takes a heavy current, the resistor will go up in smoke quickly. The resitor may be wasted but it serves as a very handy indicator. It could save a number of expensive components from being damaged.

The only way of testing the power supply for the 1 amp current flow is via an 8R2 or 10 ohm wire wound resistor capable of dissipating 10 watts. An ammeter across the output will not indicate the correct current flow because it will create a short on the output and the regulator will shut down.
If you only have low wattage resistors to do the testing, a 10 ohm ¼ watt resistor can be flicked across the output while holding the body of the resistor in your fingers. About ½ second after touching the output terminals, the heat from the resistor will reach your fingers and you will know power is available!

All the best with the project and remember NO SHORTING!

Colin Mitchell.

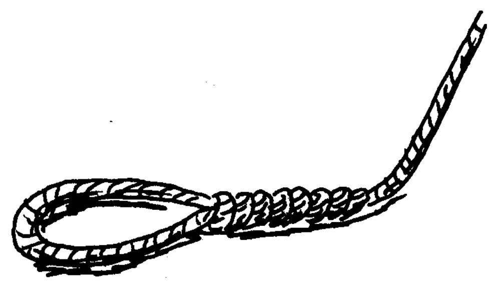


HANGMAN

AN OLD GAME - UPDATED.

This is a HYBRID circuit - meaning it is composed of two different species. We have combined transistors with IC's to achieve an update of an old game. The complexity of the circuit comes from the repetition of the transistor stages. Due to the number of biasing resistors required it is strongly suggested that you use a PC board. Not only has the layout of the board been carefully designed to make it look symmetrical when completed but also allows the project to go together so much easier. The boards are printed with an overlay and will fit directly on top of a Zippy box so the whole game looks professional.

THIS PROJECT COVERS 7 BUILDING BLOCKS AND MAKES A VERY INTERESTING GAME FOR TWO PLAYERS.



Everybody likes re-discovering something they did years ago. Here's a game we all played at school. Possibly under the name of HANG THE BUTCHER. The game is quite simple. One player thinks of a word and writes down the number of letters in that word in the form of boxes or dashes. The object of the game is for the opponent to suggest letters of the alphabet, and if they are correct, are placed on the dashes in the correct order so that the word gradually appears.

To make the game more interesting, a side issue is introduced which effectively counts the number of incorrect guesses. Each time an incorrect letter is suggested, a systematic framework is created with straight lines in the form of a gallows. A stick man, representing a person being hung, completes the diagram.

The game is concluded when the correct word is created or the stick man is completed, whichever comes first.

This is an electronic version of that game. The stick man and gallows are made with 15 LEDs and each time a TOUCH PLATE is touched, one more section of the cartoon is illuminated.

The last LEDs to be lit are 14 and 15, which represent the feet of the man. When these LEDs are at full brightness, the 8th LED begins to flash, indicating the man is 'HANGED'.

The game can be played in two ways. The 'normal' way involves the secret word and using the hangman to count the incorrect letters. The other suggestion is to take it in turns illuminating the LEDs until the flashing LED is set into oscillation.

The player creating the first sign of continued flashing is the winner.
In either game, you will have lots of fun. Especially in a darkened room where the full effect of the LEDs will be appreciated.

In either game you will have lots of fun. Especially in a darkened room where the full effect of the LEDs will be produced.

HOW THE CIRCUIT WORKS

7 BLOCKS

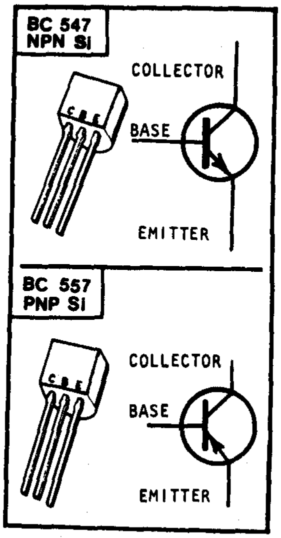
The HANGMAN game consists of 7 main building blocks. These are shown in the block diagram and are identified as follows:

1. 2Hz oscillator with voltage trip.
2. 2 KHz multivibrator
3. Voltage doubling
4. Staircase voltage detector
5. ½ second debounce
6. 1/10th second "one shot".
7. Shut down.

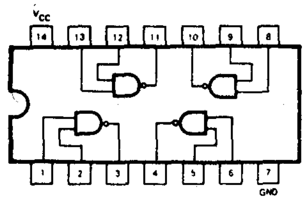
When the power is applied, the only building block to come into operation is the 2 KHz multivibrator, block 2. It is made up of gates c and d of IC2 and feeds the push-pull buffer consisting of Q11 and Q12 to charge the 100mfd electrolytic. The oscillator runs at a fairly high frequency and this reduces the size of the coupling capacitor. This building block is called a VOLTAGE DOUBLER and the voltage appearing at the output terminal is very close to double the 9v supply minus the voltage drops across the two diodes. Under no-load conditions this voltage appears at the output as 14v. We call this BOOST and we have labelled it 12v BOOST because it reduces to 12 volts under full-load conditions.

The mechanics of the voltage doubling circuit are very easy to follow. The multivibrator c and d produces a square wave which is fed to the bases of the two complementary transistors. When one transistor turned hard on, the other is full off. For the first cycle, the output gate c is LOW and the BC 557 is turned ON. The negative end of the 22mfd is taken to the negative rail and charges quickly via the top IN 4002 diode to 7.5v. At the same time the 100mfd electrolytic is charging to 7.6v via the two diodes. When the multivibrator swings HIGH, the top BC 547 transistor turns ON and the BC 557 turns off. The negative end of the 22mfd is now brought to the positive rail and its stored 7.5v will be added to that of the 100mfd electrolytic to bring the total voltage up to 15.2 volts minus .7v drop across the lower diode. In fact the voltage drop across the diodes have a double effect on reducing the voltage since they are used for each part of the voltage doubling action. They account for nearly 3v drop. We must also include the collector-emitter voltage drop of each transistor as this reduced the maximum voltage available on the 22mfd boosting electrolytic. Thus the resulting voltage out of the doubler is considerably less than you would expect. All these diode and transistor voltage drops are constant for any voltage doubler and would obviously be less noticeable when using higher voltages. This arrangement is capable of delivering 15 to 20 milliamps and since it does not have a very good regulation, the voltage under load drops to about 11 or 12 volts. This is just enough to illuminate LEDs 14 and 15 in the staircase circuit.

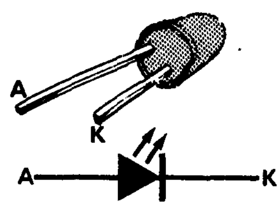
PIN OUTS:



CD4011

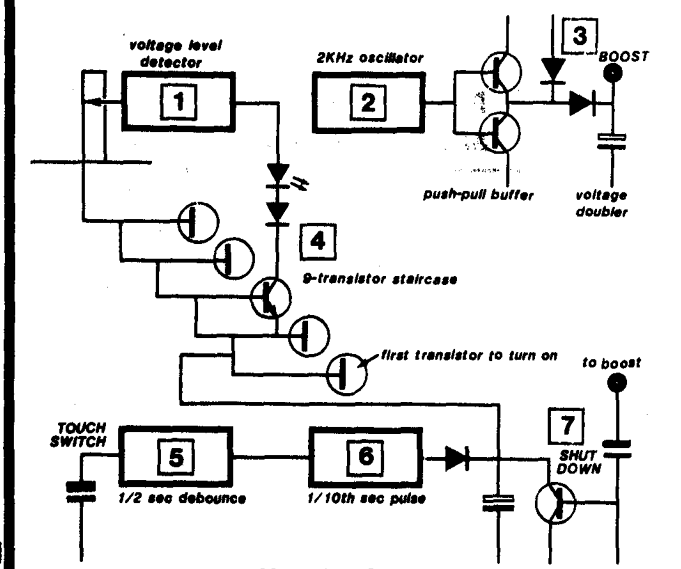


QUAD 2-INPUT NAND GATE

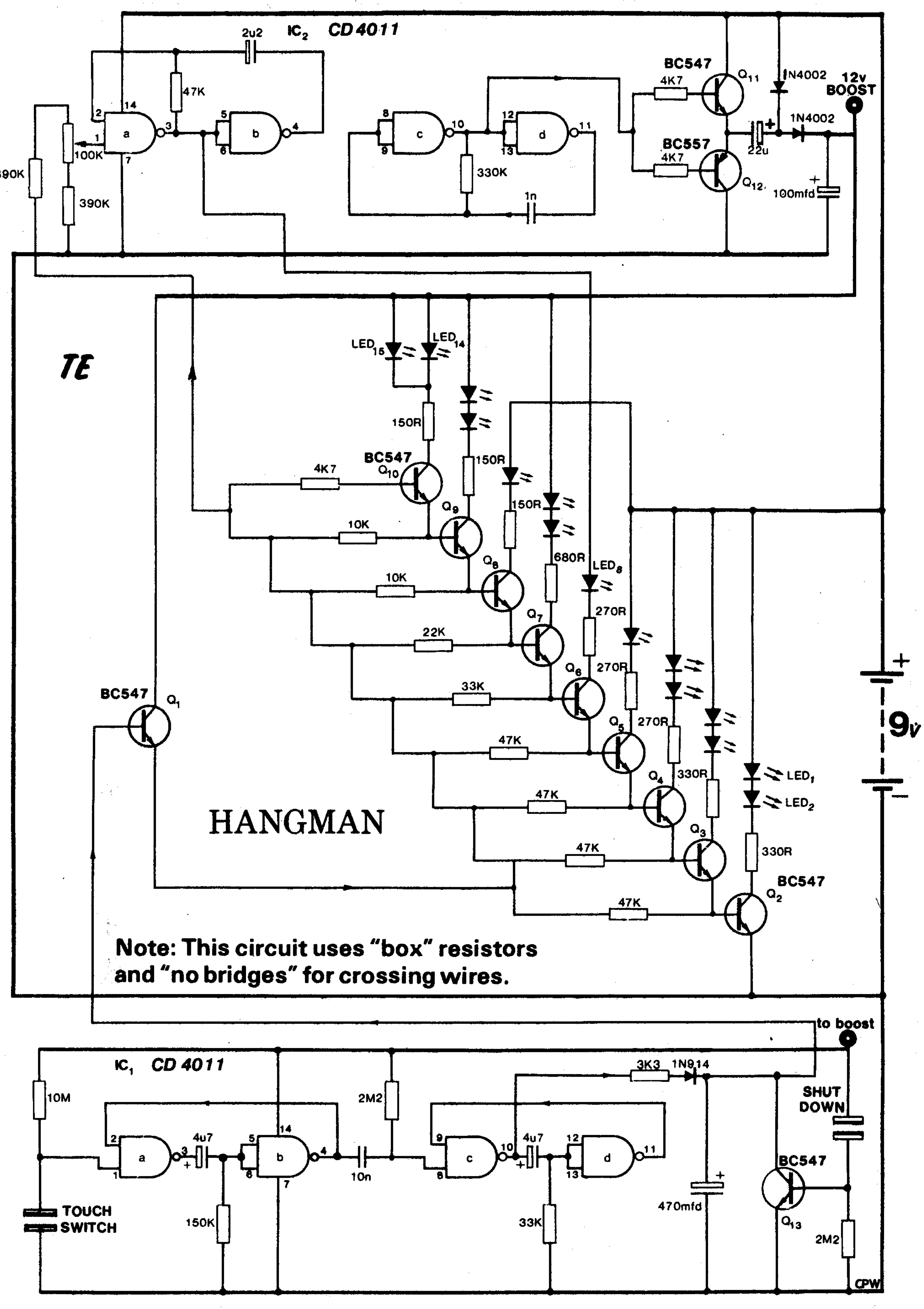


LEDs 14 and 15 are positioned as the feet of the man being hung and are controlled by transistor Q10. The reason for providing a voltage doubler circuit is two-fold. It introduces a new building block into our "library" and adds interest to the project while providing an economical way of producing the necessary higher voltage rather than using a 12v battery.

When a player places his finger on the TOUCH SWITCH, the first monostable multivibrator consisting of gates a and b in IC_1 will be triggered into its unstable state and produce



HANGMAN BLOCK DIAGRAM


IC2 CD4011
2u2
47K
100K
390K
390K
330K
1n
BC547
Q11
4K7
IN4002
12v BOOST
BC557
Q12
4K7
22u
1N4002
100mfd
TE
LED15
LED14
150R
BC547
Q10
4K7
150R
150R
10K
10K
22K
33K
47K
47K
47K
47K
680R
LED8
270R
270R
270R
330R
330R
LED1
LED2
BC547
Q2
BC547
Q1
9v
HANGMAN
Note: This circuit uses "box" resistors
and "no bridges" for crossing wires.
to boost
IC1 CD 4011
10M
2M2
3K3
1N914
4u7
4u7
10n
150K
33K
470mfd
SHUT DOWN
BC547
Q13
2M2
TOUCH SWITCH
CPW

HOW THE CIRCUIT WORKS

a spike through the 10n capacitor to operate the second monostable. The output pin 10 is normally HIGH. It will go LOW for 1/10 second then return HIGH. It will remain in this state until your finger is removed and re-applied. The time delay of the first monostable has been carefully chosen to be longer than the second so that the circuit is fully debounced and can only be triggered every half-second. The second monostable has a time delay completely independent of the first and produces a short pulse which charges the 470mfd electrolytic via the 3k3 resistor and diode. The purpose of the diode is to prevent discharging of the electrolytic once the monostable has fallen back into its stable state and produced a LOW on pin 10.

The charging of the electrolytic is exponential and each pulse from pin 10 produces sufficient energy to raise the voltage of the 470mfd approximately .75v to turn on one transistor at a time in the staircase. At the bottom of the staircase, one pulse will be sufficient to illuminate one of these steps but as they increase towards the top, more than one pulse will be required. Transistor Q1 is connected as an emitter-follower, the load being the base resistors in the staircase. The voltage across this load will be directly proportional to the voltage on the electrolytic (i.e. the base voltage) minus the .75v base-emitter voltage drop. The main purpose of the transistor is to separate the 470mfd electrolytic from the load of the staircase. If we were to remove this transistor the circuit would function as before except that the load of the 9 staircase transistors would tend to discharge the electrolytic rather quickly.

So, in effect, the emitter-follower transistor is providing an impedance matching arrangement to reduce the drain on the electrolytic, to a value about one-hundredth of a directly-coupled version.

The base resistors have been chosen according to the voltage they will be required to drop. Many factors influence the actual value selected for each base resistor as the impedance of the circuit changes with rising voltage and this alters the conditions considerably.

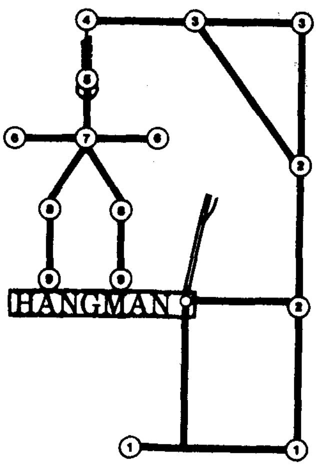
The first transistor in the staircase is Q2. It will turn on when its base voltage is .75v higher than the emitter voltage. The second transistor, instead of being connected to the ground, is connected to the base of the first transistor. It too will turn on when its base voltage is .75v higher than its emitter. This means the incoming voltage will need to be .75v + .75v or 1.5v before it will be fully turned on. This reasoning continues up the staircase so that the top transistor will require 6.75v to be turned on.

Extending back, the 470mfd electrolytic requires 6.75 + .75v or 7.5v and the output of the IC .6v higher again to account for the diode drop. The IC needs an even higher output voltage to be able to supply a charging current through the 3k3 current limit resistor. To be on the safe side, the supply rail should be 7.5v + .6v + 2v = 10.1v. The need for a high supply voltage is even more important when illuminating the top LEDs 14 and 15. So much of the supply has been lost in the base-emitter junctions that 11.5v is the absolute minimum voltage if we expect to get adequate brightness from the top LEDs. This will allow us just 4v for the dropper resistor and LEDs.

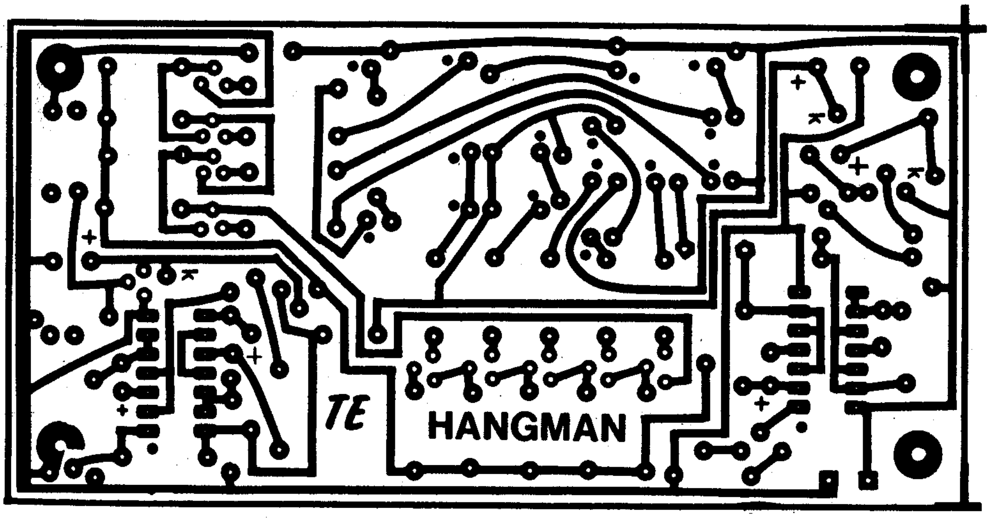
You will notice the power to the 15 LEDs comes from 3 different sources. LEDs 1 to 7 and 11 are supplied directly from the 9v supply. LED 8 derives its supply from a slow cycling oscillator. This is made up of gates a and b of IC2. It forms a gated oscillator with pin 3 normally HIGH. The oscillator is triggered via pin 1. When it detects approximately half supply voltage through the high impedance resistor network comprising the 390k, 100k pot and 390k resistor, it will flash LED 8 at about 2 cycles per second.

LEDs 9 and 10 are series LEDs and need to be driven from a source slightly higher than 9v. To achieve this we must take them to the BOOST rail. LED 7, being a single LED will just operate from the 9v rail. The remaining 4 LEDs need 12v to operate. LEDs 14 and 15 are paralleled together so that they can attain full brightness from the 12v rail.

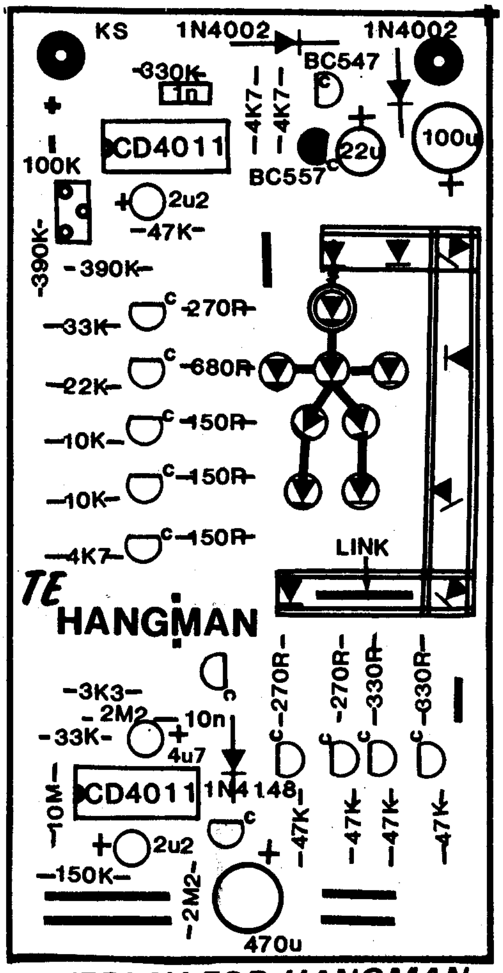
Resetting the game is accomplished by discharging the 470mfd electrolytic via Q13. This transistor is normally biased in the off condition with the 2M2 base-emitter resistor. When you touch the SHUT DOWN wires with your finger, a small forward bias is applied to the transistor and it turns ON to bleed the electrolytic. The light-emitting diodes will gradually turn off as the voltage on the electrolytic falls. When all the LEDs are off the quiescent current for the game is only about 100 microamps. This is so small that no on/off switch is required and even small penlite cells will last their normal shelf life.



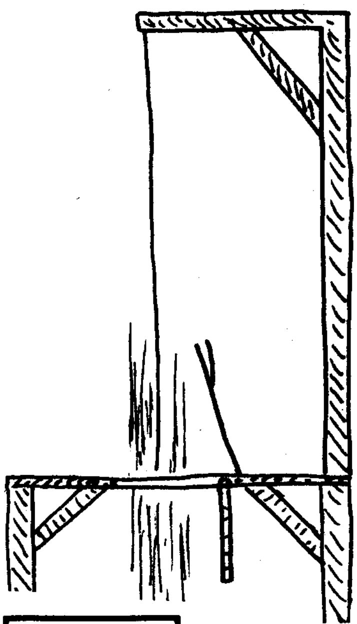
LEDS TURN ON IN THIS ORDER

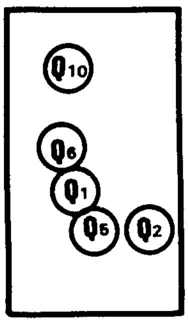


PC LAYOUT



OVERLAY FOR HANGMAN





An overlay makes construction so easy. It will take the best part of an hour to assemble the HANGMAN, even with all the parts ready at your fingertips. See the cover photo if you are not sure where any of the components are placed. Use only DURACELL's to power this game. Ordinary 'dry cells' will not last very long as their voltage soon falls to 8v or less for a 9v battery. At this voltage the doubling circuit will be incapable of supplying sufficient voltage to light LEDs 14 and 15.

HANGMAN PARTS

R1	"	10M
R2	"	150K
R3	"	2M2
R4	"	33K
R5	"	3K3
R6	"	2M2
R7	"	47K
R8	"	330R
R9	"	47K
R10	"	330R
R11	"	47K
R12	"	270R
R13	"	47K
R14	"	270R
R15	"	33K
R16	"	270R
R17	"	22K
R18	"	680R
R19	"	10K
R20	"	150R
R21	"	10K
R22	"	150R
R23	"	4K7
R24	"	150R
R25	"	390K
R26	"	390K
R27	"	47K
R28	"	330K
R29	"	4K7
R30	"	4K7
C1	electrolytic	4.7mfd 16v
C2	capacitor	10n 100v
C3	electrolytic	4.7mfd 16v
C4	"	470mfd 16v
C5	"	22mfd 16v
C6	capacitor	1n 100v
C7	electrolytic	22mfd 16v
C8	"	100mfd 16v
Q1 - Q11	Transistor	BC 547 (11 off)
Q12	"	BC 557 (1 off)
Q13	"	BC 547 (1 off)
IC1	Quad Nand	CD 4011
IC2	" "	CD 4011

LED 1 - 15 3mm Red Leds. BUY 20 LEDS.
1 - 100k mini trimpot
1 - Diode, 1N914 or 1N4148
2 - Diodes 1N4002
battery snap
9v battery
HANGMAN PC BOARD

CONSTRUCTION

Examine the printed circuit board for any holes which may be filled-in with solder. This happens during manufacture, when the copper tracks are being tinned. Clean these out with a soldering iron and needle or a length of copper wire. Whether you use a ready-made printed circuit board or a home made board, some form of printed construction is essential for a circuit of this complexity.

Follow the usual method of fitting the parts by firstly soldering the resistors in position. Refer to the layout for their positions. They can be inserted either-way-round but it's best to fit them so that the colour bands can be easily read when you are checking them later. Some boards have been silk-screened with an overlay of the parts. This will assist enormously with construction and show you which side of the board to mount the components as well as their exact positions.

Next fit the jumper wires including those making up the touch switches. You can use tinned copper wire or the ends of resistors. Do not use plain copper wire as its appearance will deteriorate and become tarnished after a period of time.

The 100k mini trim pot, 3 diodes and 2 capacitors are fitted next. The diodes must be inserted with their cathode end as shown. This is identified on the component by a band. In the case of the small signal diode, it may have a blue, red or black band, depending on the type supplied. The 5 electrolytics must be inserted with the positive end as marked on the board. The positive lead is the long lead so don't cut them off until they have been inserted and soldered.

Next fit the 15 light-emitting diodes. These are very temperature sensitive and will be destroyed if allowed to get too hot during soldering. Hold them in position during soldering with your fingers to prevent them getting too hot. The cathode all face one direction and are marked on the board with a dot. This is the short lead and if you look into the body of the LED you will see it is the largest portion inside the LED.

Lastly insert the semiconductors most vulnerable to damage. The 13 transistors are inserted as shown in the layout diagram. You will notice a uniformity about the way they are inserted and this should prevent too many mistakes. Finally the two chips are fitted to the board. Locate pin 1 on the chip via the dimple and align it up with pin 1 on the board as marked with a dot. Double check this against the layout before soldering as you will not be able to remove the IC after it is soldered.

IF IT DOESN'T WORK

If your unit fails to operate for some unknown reason, it can be diagnosed with the aid of a simple piece of test equipment. This is a LED and 330R resistor connected to two jumper leads. A multimeter will also be handy but not essential. Connect the cathode of the test LED to ground and use the resistor lead as a probe. Switch on the game and test these points on IC2 for a HIGH: Pins 3, 5, 6, 10, 11, 12, 13 and 14.

Pins 10, 11, 12, and 13 will be supplying the BOOST circuit. Notice the LED will light up very brightly when connected to the positive of the 100mfd electrolytic. Also connect to the common emitter terminal of the transistors for a HIGH. Once you have BOOST, IC1 will be obtaining its power and will be ready for test. Before checking the second IC, test each of the LEDs by taking the flying lead of the resistor to each of the resistors connected to the collectors. The test LED will light up as well as a single or pair of LEDs on the game. If only 1 LED of a pair lights up as well as the test LED, you will know that one LED has shorted during assembly due to soldering. If no LEDs light up, one of the LEDs is open circuit, and if only the test LED lights up, both LEDs are shorted. If one section of LEDs is not being turned on, the transistor could be getting incorrect biasing or suffering from a base-emitter short.

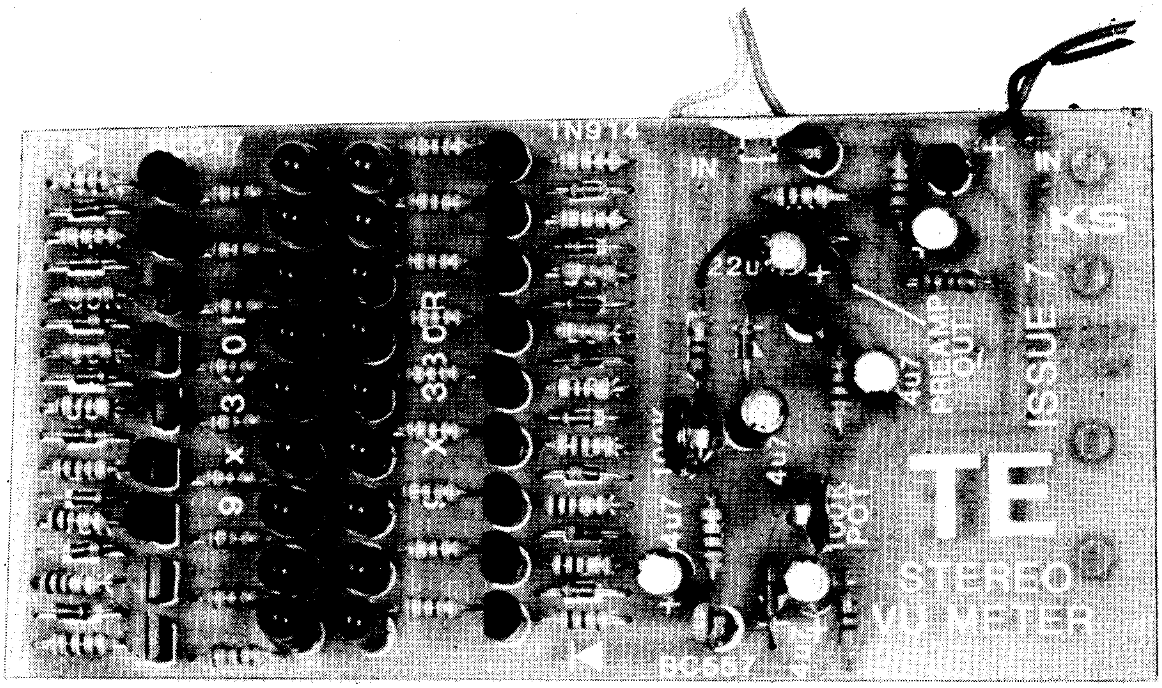
The 470mfd electrolytic can be charged externally via a 10k resistor connected to the positive of the battery. This will sequentially turn on the LEDs. Once the LEDs are alight, they should stay illuminated for 5 minutes or so without the voltage on the electrolytic draining away. Any gradual decline will indicate a leakage path. Remove the shut-down transistor and the staircase base resistors starting from Q10, to locate the fault. Q1 should also be checked.

The second 1/10 second TOUCH SWITCH delay output is detected on pin 4. It is normally HIGH and goes low for 1/0 second. Pin 10 is the pulse output and will flash briefly when gates c and d are pulsed via the 10n capacitor. You will be able to detect this pulse on the test LED.

If LEDs 14 and 15 do not illuminate, the battery will be slightly LOW. The circuit is fairly critical on voltage. The supply cannot deviate more than 1 volt either side of 9v.

Use this same reasoning to investigate any other of the blocks you feel are not operating satisfactorily. Once you have run over the whole circuit, connect up the battery firmly and mount the board on a Zippy box. The board will accept 2 small screws to finish the job. All that is left now is find an opponent and meet your challenge.

STEREO VU METER



The completed unit in close-up. The overlay makes construction easy. Make sure the left-hand row of resistors starts with 47k at the bottom and 4k7 at the top. I saw one unit with the whole row reversed. It made very little difference to the performance of the unit, but it was not quite as sensitive as the correct version.

Does your amplifier have a level indicator?
Have you always envied the fancy amps with LED level bar graphs?

Would you like to build your own STEREO LEVEL INDICATOR?

Pine no more, it's here. Our feature project this issue will suit all audio buffs. It is a STEREO LED LEVEL METER. It's the cheapest and best bar graph display available and best of all, it uses readily available components.

You only need a handful of LEDs,22 transistors some resistors, diodes and a set of electros - it doesn't require any chips.

You may be wondering why we didn't choose the LM 3914 or LM 3915 bar-graph LED driver chips. The reason is simple. We learnt our lesson from the Mini Frequency Counter book. In this project we used a relatively novel chip, the CD 4026. And after releasing 10,000 copies of the magazine, with printed circuit boards attached, the chip became almost unobtainable in Australia. This proved to us that many of the readers were making the Mini frequency Project. Now, a chip such as the LM 3914 is scarce at the best of times. Can you imagine what would happen if we used it in a project? Ninety per cent of the readers would miss out. This means we must confine our projects to readily available components and avoid rare items, no matter how inviting they look.

We compared a LED level meter using the chip with our unit and the difference was negligible. Both had the same quick response-time and about the same readout values on the line of LEDs for the same input signal. But the big difference is in the cost of construction. By using transistors, you will save $4 over the cost of two chips. If you don't mind the additional time required to fit the extra components, the $4 is a valuable saving and by using discrete components, you can build it from parts you may already have in stock.

THE CIRCUIT

The circuit basically consists of two identical channels feeding two rows of LEDs. A high-gain bootstrap front-end is also provided to allow the board to be coupled to an inbuilt speaker/microphone which will give a mono readout of the sounds being picked up.

A mini trim pot is provided to set the sensitivity. This makes the project completely portable and it can be used as a SOUND LEVEL meter in a disco or other noisy situation. To give a readout in dB it would require calibration. The simplest method of calibration would be to compare it with a commercial unit and give each LED a value in dB.

By providing another bootstrap circuit, a portable stereo sound detector could be made. This could compare sound level in different parts of a room or compare the relative outputs from 2 speakers.

As designed, the stereo section needs to be wired into the output of each channel, across the speaker terminals, and the unit mounted in some prominant place for an eye-catching display.

Before you begin. Lay the components on the work-bench in a position relative to that on the PC board. Some of the parts have the same value, such as the 330R resistors. These should be positioned on the board with their tolerance bands all round the same way. Separate the two BC 557's from the other two transistors and be sure you can identify the 22mfd electrolytic from the 4.7mfd.

The board looks deceptively simple because most of the components are placed in rows. It will take at least one and half hours to construct the project and the most important facet throughout construction is to create a neat appearance. This means aligning each component with the next and keeping the heights all the same.Otherwise the neat appearance will be destroyed.

The suggested method of construction is to start with the resistors and diodes. These should be inserted alternately as required by the board so that you have maximum room when placing them in position.

Slow and sure wins the race. It is best to insert the parts one at a time and push them firmly onto the board. Nothing looks worse than a mass of floating components, some high, some bent this way, others bent the other way. Once you push them onto the board, bend their wires outwards so that the component is held in position. Turn the board over and solder the two connections quickly. Check that the component has not shifted then snip the two wires. Continue down each row, taking one item at a time.

If you find that you are closing over some of the holes with solder when you are soldering, I suggest you only tack-solder the leads and wait for the other component to be inserted, before finishing the joint. Tack-soldering is very fast and requires almost no solder. This prevents the solder flowing over other lands and filling up the holes.

You may have noticed that the two channels are a mirror-image of one-another. This means the cathodes of the LEDs face outwards and before inserting each LED you should look into their opaque body to make sure they are being inserted correctly. In our prototype, the two top LEDs of each channel are a different colour, mainly to add interest to the display. You may choose to add another colour for the bottom two or three LEDs and produce an even more exotic display.

The driver transistor for each channel and the bootstrap circuit fits onto the right-hand end of the board. All the component values are identified on the overaly and the two BC 557's are shown as 'filled in', whereas the BC 547's are open 'D's'..

There are no jumpers on the board. However we have made provision for for connecting the bootstrap circuit to either one or both channels via one or two links. These links are taken from the 'pre-amp OUT' point to either the left hand channel or the right hand channel.

PARTS LIST:

18 - 330R
2 - 470R
1 - 1k
1 - 2k2
5 - 4k7
5 - 10k
2 - 22k
2 - 33k
4 - 39k
4 - 47k

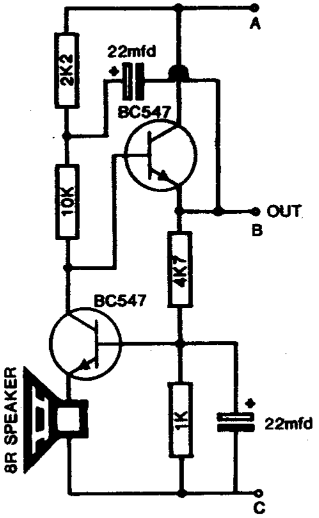
4 - 4.7mfd 16v PC
2 - 22mfd 16c PC
20 - BC 547
2 - BC 557

18 - IN 914 or IN 4148

14 - 5mm red LEDs
4 - 5mm green LEDs

2 - 100k mini trim pot
1 - battery snap
1 - speaker 8 ohm

1 - VU METER PC board



The BOOTSTRAP circuit connects to the LED bar graph via A, B, and C. Only one BOOTSTRAP circuit is provided on the board. It is capable of driving both bar graphs in a mono mode. For stereo readout, via a bootstrap circuit, you will need to build another bootstrap. This will give a portable STEREO SOUND LEVEL INDICATOR.

HOW THE CIRCUIT WORKS

The VU METER consists of 3 sections:
1. BOOTSTRAP CIRCUIT
2. BUFFER TRANSISTOR
3. STAIRCASE VOLTAGE DETECTOR

The first section should not be new to you. We have presented the BOOT-STRAP circuit in a number of previous projects. It is very successful at allowing a dynamic microphone in the form of a 2¼in speaker to detect small sounds and have them amplified sufficiently to be fed into a normal amplifier.
The BOOTSTRAP is rather unique in its operation. It uses 2 directly coupled NPN transistors wired in a similar mode to cascade to give an enormous gain. In our prototype we measured this to be about 1,000 times!

In the quiescent condition, the transistors in the bootstrap circuit are slightly turned on. This means they will accept a few millivolts from the speaker and turn the circuit on harder or turn it off. During idling conditions 2 millivolts is developed across the speaker due its resistance of 8 ohms.

Take the case where the speaker produces 2 millivolts which is in phase with the quiescent voltage and this will turn the transistor Q1, slightly off. The collector voltage will rise and in doing so, take the base of the emitter-follower Q2 with it. Under normal circumstances, the collector voltage would rise about .2v, for Q1. This will make the emitter voltage of the emitter follower rise .2v (which is normal for an emitter follower). Now the top 22mfd electrolytic will transfer this .2v to the join of the 10k and 2k2 resistors. Since Q1 is turned off, the .2v rise will appear on the base of the top transistor and turn it ON further and cause its emitter to rise another .2v . This action feeds back into the base via the electrolytic and the emitter rises to slightly less than rail voltage. Here it will stop due to the collector-emitter voltage drop preventing it going any further. At this stage the emitter has risen from 3.5v to 8.5v and the join of the 10k and 2k2 risen from 8.5v to 12.5v. Yes, that's right! The junction of the two resistors rises to greater than the supply voltage. The capacitor cannot hold its charge for ever and some of it is bled off via the 2k2 resistor. This reduces the base voltage and the transistor begins its downward excurtion.

I have taken the extreme case. If the first transistor does not turn on to quite the same extent, the emitter-follower will rise until the loss from the top electrolytic prevents the transistor from rising any more, and it begins to fall. The lower 22mfd prevents this swing from appearing on the base of Q1. It acts as a damper.

The output from the BOOTSTRAP can be as high as 2v p-p and this is ample to drive the buffer stage. In fact the signal needs to be attenuated by a pot so that the range can be set according to the amplitude of the input signal.

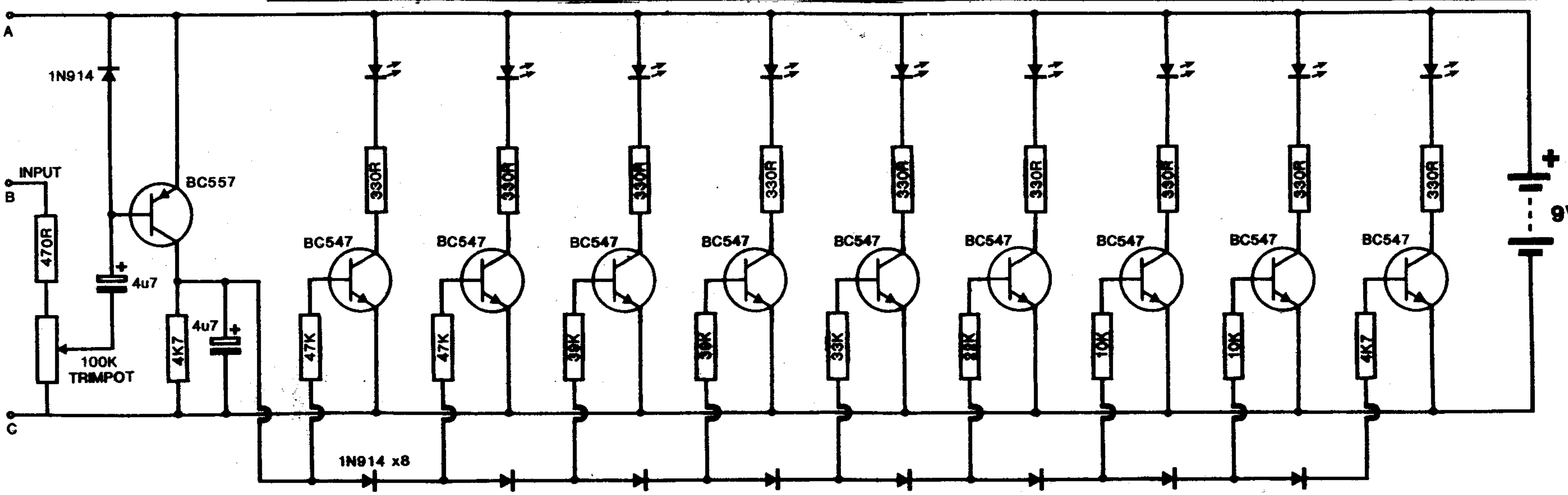
The 470R resistor in series with the pot is only needed when the VU meter is connected di rectly across speaker lines.

The BC 557 is not an emitter follower. Don't get confused. It is wired as a normal common emitter stage for a PNP transistor. Thus it will provide a high gain in this situation. The AC voltage appearing out of the 100k trim pot will pass through the 4u7 electrolytic and become rectified by the 1N 914 diode. With no signal present, the voltage on the base will be 9v. As the input signal increases, the voltage on the base will drop to 8.35v and this is sufficient to turn the transistor ON fully.

The voltage on the collector will range between 0v and 8.5v. This voltage is stored in the lower 4u7 electrolytic and applied to the chain of 8 diodes. The 4u7 dictates the decay rate and gives the LEVEL METER its rapid attack, slow decay characteristic and allows even brief peaks to be detected. To reduce the decay time you can increase the electrolytic to 22 mfd and this will keep the LEDs illuminated for a longer period, similar to the commercial units.

Between each diode is a high value resistor. As the voltage rises to about .6v, the first transistor turns ON. At this stage the voltage on the cathode of the first diode is 0v since the .6v has been dropped across it.

The voltage needs to rise to about 1.2v before the second transistor turns ON. This continues down the line with each transistor turning ON at its allotted voltage level.
The set of 330R resistors limit the current through the LEDs to a safe value and the base resistors serve as a voltage dropper so that the base will not be forced to go higher than .6v.
The number of transistors which can be operated in this staircase arrangement is limited to the battery voltage available since each transistor and diode will take .6v from the voltage supplied by the BC 557 buffer transistor.



TESTING

To test the stereo VU meter, connect the two links as shown on the board and connect the dynamic microphone (speaker). Solder a battery snap to the board and connect a 9v battery. This project is now a self-contained level meter and will give a dual readout of the sound detected by the speaker. We are using a small speaker as a microphone as we have had a great deal of success with its sensitivity. No calibration is required. You only need to position the pick-up (spkr) near a radio or stereo which is playing at normal listening level and adjust the sensitivity controls. These are the two 100k mini trim pots in the buffer stage. First you must set each one so that the top led is just illuminated when a loud passage is being received. Then you need to trim the two displays so that they produce equal readouts for the same information.

FAULT FINDING

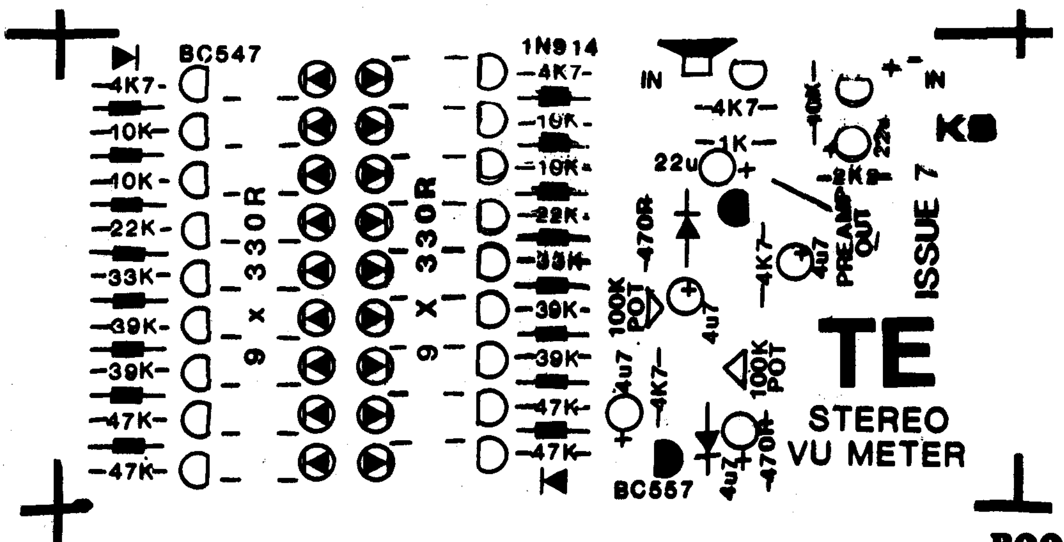
Since each channel is identical, you will be able to reference off one channel to repair the other.

The chain of transistors are all DC coupled and you can test their operation by using a 10k resistor connected to a set of jumper leads. Connect one jumper to the positive of the battery and touch the other onto the cathode of the lowest diode in the staircase. You cannot do any damage to any component when probing around either channel and I suggest you take this opportunity of seeing the effect of a turn-on voltage when applied to a set of transistors.

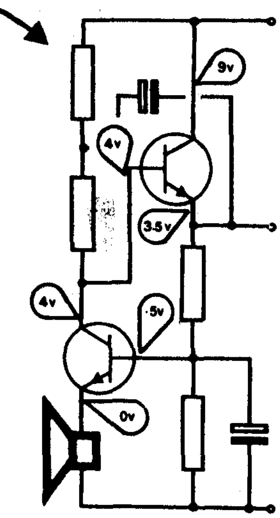
When the 10k resistor is touched on the cathodes, almost all the LEDs should light up. By moving the resistor up the chain, the top LED will light. This will show the channel to be functioning and you should test the other channel for the same effect. If one LED fails to light, you may have a base-emitter short in one of the transistors or the LED itself may be faulty. If anyLEDs above number 6 fail to light, one of the diodes may be open or a dry joint may be the cause.

If you have trouble gatting one channel to function, you can use the components from the other channel as test pieces. This is the great advantage of having two identical channels. But by using parts from the good channel, you could finish up with two bung channels. That's the risk you take.

The buffer transistor can now be tested by connecting the 10k to earth and touching the other end on to the base of the BC 557 transistor. This will illuminate one complete row of LEDs. The remainder of the circuit is AC coupled via the 4u7 electrolytic.
Only the DC conditions of the bootstrap section can be tested with simple equipment. Use a multimeter to detect voltages similar to those given here:



THE OVERLAY: Eventually you will be able to construct our projects using only the overlay. If you are unsure, follow through the notes.

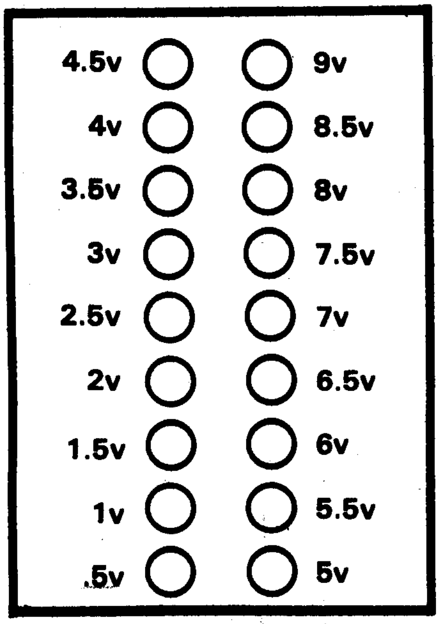


BOOTSTRAP VOLTAGES

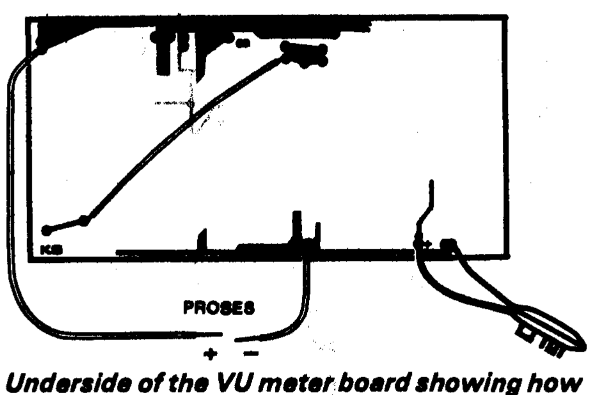
Both transistors will be turned on very slightly and because it is a very high gain circuit, you cannot remove one transistor and hope to get a smaller amplification. It will completely fail to work.

0 - 9v LED VOLTMETER

If you already have a VU meter, maybe you would like to build this LED VOLTMETER. It uses the VU METER board and requires only those components driving the two rows of LEDs.



SCALE FOR LED VOLTMETER



Underside of the VU meter board showing how to cascade the two rows of LEDs.
The positive and negative probe leads are shown connected to the board. You will also need a 6v or 9v battery to supply power to light the LEDs.

It is often handy to have a voltmeter on the work bench to test low voltages without tying up the multimeter.

The STEREO VU METER can be converted into a 0 - 9v voltmeter very easily.

Since the staircase of transistors are all DC coupled, they can be used to display voltages from .5v to 9v.
To make the LED VOLTMETER easy to read, we have allocated each LED with a voltage increment of .5v. This proved to be on the safe side by less than .25v for a full-scale reading and was quite accurate through the range.

Now you can test your battery packs and lantern batteries under a slight load and get an accurate assessment of their voltage. This voltmeter has a medium sensitivity equal to 10k ohms per volt and draws only 1 ma from the voltage being tested. You will still need to have a 6v or 9v battery to drive the row of LEDs, but this circuit will only draw power when the LEDs are showing a reading. It will shut down to almost zero and conserve the battery after a measurement has been made.
To produce a range of 0 to 9v in .5v increments, we are using the two rows of LEDs in cascade. This means the end of the first row is connected to the beginning of the second.

Refer to the layout diagram for the jumper link. You will not require any of the bootstrap section or the buffer transistors for this application and they can be removed from the board. If you wish to remove only the minimum of components, both the 557 transistors, the 4k7 load resistrs and the 4.7mfd storage electrolytics should be removed. The remainder will not inhibit tha performance of the circuit.

No extra parts are required except two lengths of flex to act as test leads. Use a red lead for the positive end a black lead for the negative. When testing a voltage, it will not damage the meter if the leads are reversed. The LED display will just fail to light.

TE Clock

Ideal for your workshop or beside your bed, this illuminated clock can be read by day or night. It will help you run to time.

Our cover feature is a clock. A simple every-day timepiece. Something we think very little about. Most of us have a clock in at least two rooms of the house and at least one of these will be digital. If not, you will be particularly interested in our project. If you already have a digital clock, you will appreciate the advantage of an illuminated dial.

One hundred years ago, the purchase of a timepiece was a great decision. Miniature clocks and watches were enormously expensive and necessitated careful decision as to the most suitable model and the relative costs.

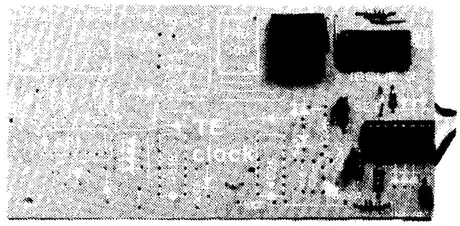
Clocks have always been an item of great beauty. If you have ever wandered through the clock section of a museum you will appreciate the aesthetic designs which have been incorporated into the face and even the workings of many clocks and watches.

This is the only mechanical product I can recall where the workings have been specifically designed to be ornate as well as being fully functional.

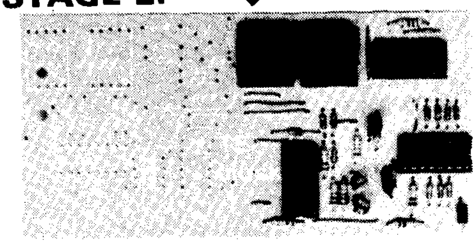
From the beginning of time, man has strived to produce a more accurate time keeper. From the simple time-candle, to the atomic clock, we have seen the introduction of one improvement after another. As you are possibly aware, the biggest threat to accurate time keeping has always been temperature. The variation between hot and cold expands and contracts all materials such that a pendulum will increase in length on a hot day with the effect that it will swing at a slower rate. Some form of temperature compensation is required which will alter the distributed mass of the pendulum and keep its centre of gravity constant. Once this was achieved, the next major breakthrough was the balance wheel. This enabled pendulums to be incorporated into pocket watches where the pendulum needed to be compact and placed in any position and still function. Again, temperature compensation had to be included in the form of two bonded metals on the rim to expand or contract the arms of the balance wheel. Static and dynamic balancing was also an essential part of providing accuracy and for this a tiny set of screws was placed on the rim of the balance wheel. Needle point bearings and jewels were added to reduce friction and for 30 years the Swiss engineers had the watch market sewn up.

The clock is constructed in 4 stages. Each stage adds a display to the board. After completing each stage with its associated driving components, the clock is tested. This will reduce the chance of a mistake and make troubleshooting very easy. . . .if required.

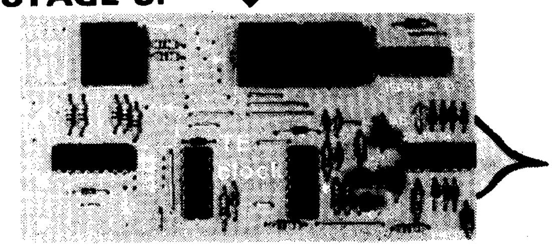
STAGE 1:



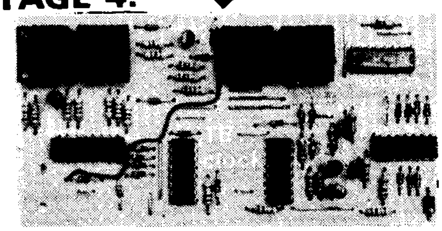
STAGE 2:



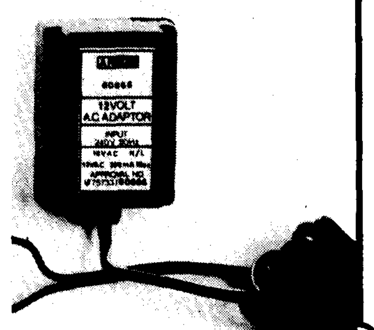
STAGE 3:



STAGE 4:



An AC plug pack of 100 - 300mA rating powers the clock. Don't use a DC plug pack as we need the AC to trigger the 4040 counter chip. The ripple from the DC plug pack would not be sufficient.



A partial threat came with the introduction of electronic watches using a tuning fork principle and an enormous chain of gears to divide down the vibration of the resonating arm.
It was not until the introduction of digital electronics and its miniaturization due to the US space program, that the wind-up watch market crashed.

Every-one likes new and potentially more-accurate devices. The consumer abandoned his old watch for one of these new electronic wonders. Initially they were LED displays and needed a switch to illuminate the dial. This worked well for a while but if your required to know the time more than once per day, the battery life was severely limited.

The introduction LCD displays and a one year battery life cured that. You could now get a watch having an accuracy of .1 second per month, for less than a wind-up style. Additionally the watch would run for about a year on one set of batteries and had an extra feature of day, date and alarm. Some even boasted dual time zones and stop-watch facilities, all at a price below that of 10 years ago.

It's no wonder digital watches took off.

For a construction project, there is one obstacle to producing a digital watch or clock. Most of the chips used in these products are designed especially for a partcular function and no technical knowledge is to be gained by soldering a single chip into a circuit.
On the other hand, if we produce a clock circuit using a purely digital approach, the number of chips required make construction very costly.

So some form of compromise has to be made. The readout can still be digital but the method by which some of the digital stages operate will have to be simplified with a few tricks. By bending the rules a little, we can produce a digital clock with as few as 5 chips. And this is what we have done. We have used transistors for some of the operations.

Before constructing this project, you must have completed at least three other projects from TE or other magazines. There are two main reasons for this.
1. It will ensure only those capable of constructing a projectof this complexity do so,
2. It will spread out the construction of the clock over a longer period of time to allow everyone to buy the components. This is important as some of the IC's are in relatively short supply.

It does not matter which three projects you have constructed however the inclusion of the LOGIC DESIGNER will be an advantage since it will be used in the testing of each stage.

USING CD 4033's

If you experience difficulty obtaining CD 4026 IC's, a replacement in the form of CD 4033 can be used.

The only modification to the board is at pin 14. It must be taken LOW to prevent the display showing a figure 8 at all times. Pin 14 is the LAMP TEST pin and when it is taken HIGH, it presents an output on all segments to test if they are all operating.

The other difference between the chips is pin 3. The 4033 provides ripple blanking at pin 3 which can be used to suppress unwanted zero's at the beginning of decimal numbers. With suppression, a number such as 00.05 would be shown as 0.05. This line is held HIGH for normal operation as is the display enable of the 4026. Therefore no modification is needed to the PC board.

In place of the Display Enable out, pin 4 is Ripple Blanking out. AS the display enable out is not connected in the project, no modification is needed at this pin.

PARTS LIST

1 - 220R
7 - 470R
1 - 2k7
5 - 10k
6 - 100k
1 - 330k or 33k (value not critical)
1 - 470k
2 - 1M

2 - 10n greencap
1 - 1000mfd 25v electro

1 - 1N 4001 diode
22 - 1N 914 diode

3 - BC 547 transistors
1 - BC 557 transistor

4 - FND 500 display

2 - CD 4026 (or CD 4033 with mod)
1 - CD 4040 binary counter IC
1 - CD 4511 display driver IC
1 - CD 4518 dual BCD counter IC

tinned copper wire
Plug pack 9v AC 200mA
TE CLOCK PC BOARD

BEFORE ASSEMBLY

Before you add any components to the board, it must be inspected. This isn't a 5 second glance-over. It will take at least 5 minutes to fully inspect both sides of the board and remedy any slight imperfections. You will need a sharp knife and a drill which is capable of accepting a fine twist bit such as a number 60 drill or a .85mm, .9mm .95mm or 1mm bit.

Refer to the layout pattern for the correct position of the copper tracks. Make sure none of the tracks touch each other or even come near to each other. If two tracks short out,you will damage one of the chips, especially if it is a display driver. Cut between the tracks with the knife so that the gap is enlarged.

Next you should check for any undrilled holes. Every pad should be drilled and even one or two holes are required on the bus rails. If you need to drill a hole, make sure the land is free of excess solder to prevent the drill wandering off centre. Use only a sharp drill as a blunt one will lift the land off the board. If this happens, don't worry, when you are assembling the project, the component lead can be bent over and soldered to the remaining part of the track.

Finally inspect the overlay for any missed printing.
This inspection should be done at least twice. It is so easy to miss a fault when you are not familiar with a situation.

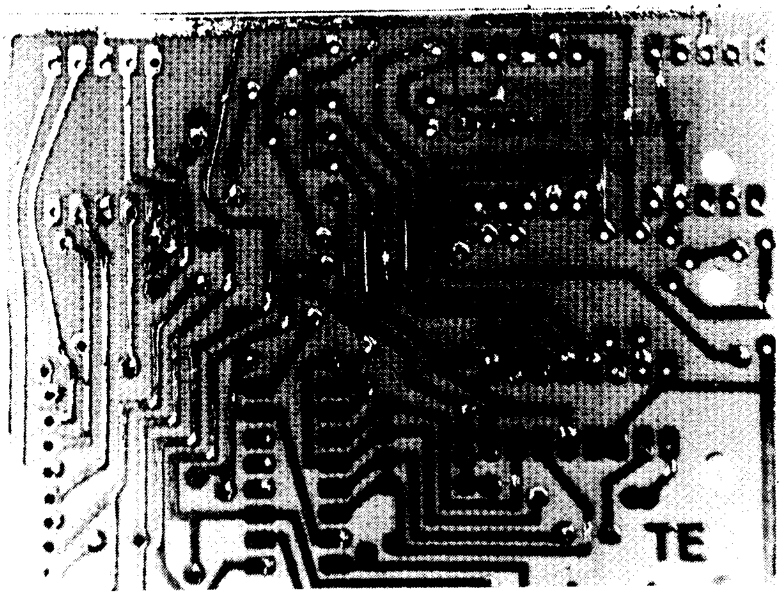
The photos show three common faults on PC boards:
1.Undrilled holes,
2.Touching lands.
3.Tracks missing.

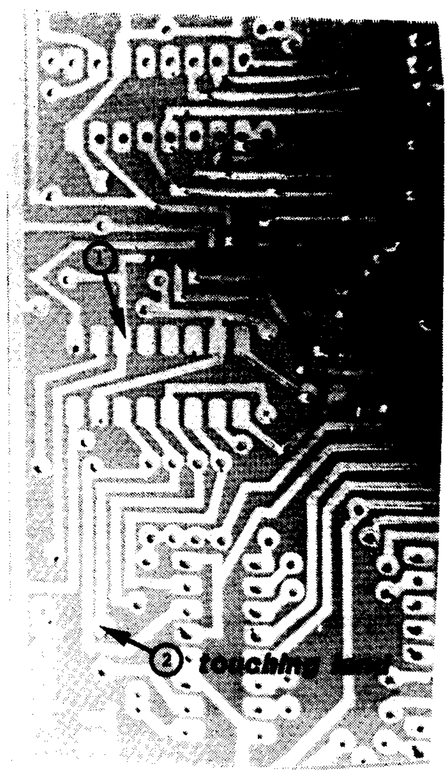
SOLDERING

Look at your soldering iron. Is it a temperature controlled Weller or a Scope TC 60? Is it a Dick Smith Special or an Ellistronics cheapie or an Altronics $10 special? This is about the only range of soldering irons I will accept for the construction of our projects. If you are using greater than 35 watts or an instant heat iron such as Scope or Birko or Pronto, they can be thrown out immediately. We don't want our projects ruined with a plummers soldering iron or a frizzling hot instant iron. They absolutely butcher the boards and damage the parts in the process. You cannot possibly expect the project to work properly after plastering it together with a cumbersome tool.

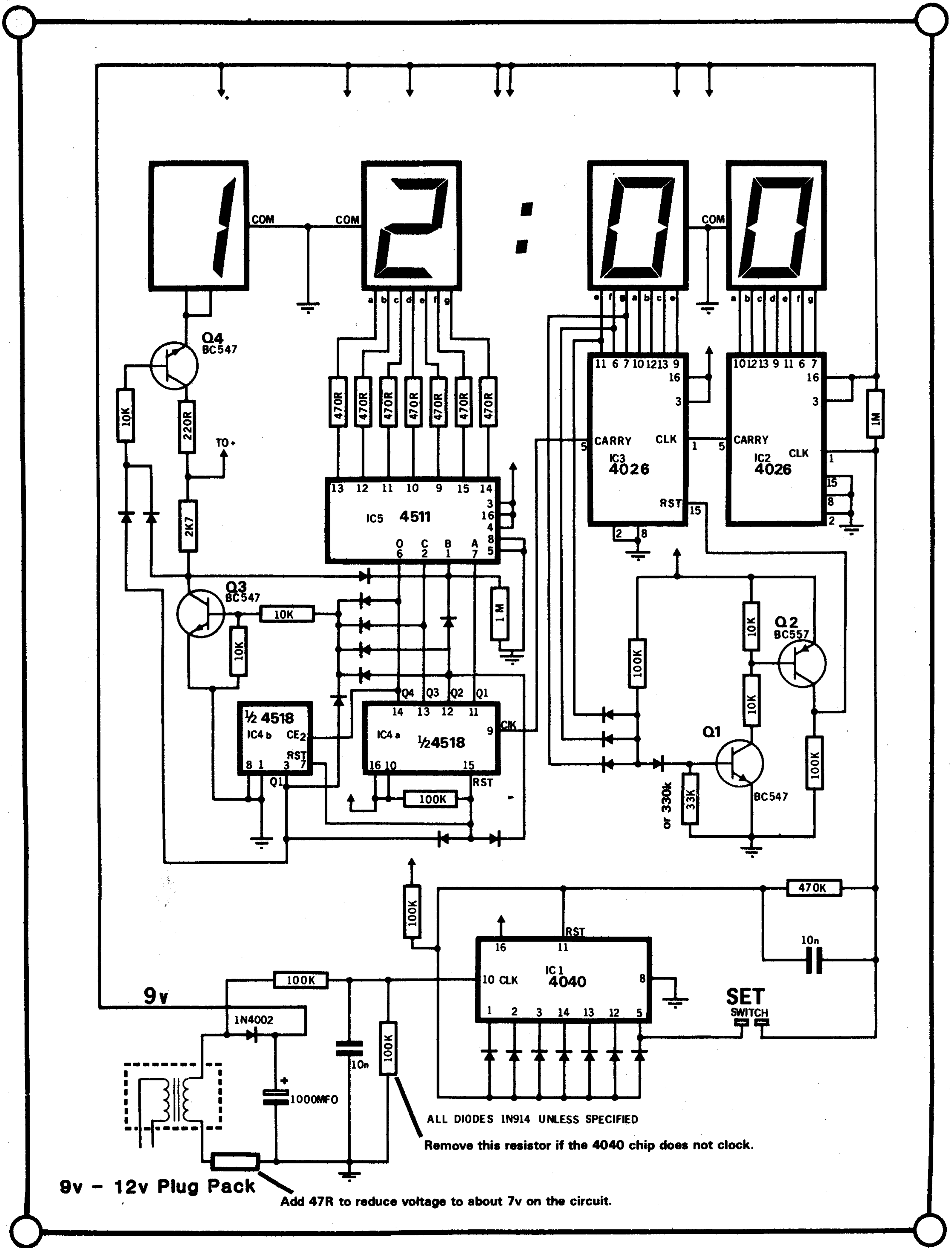
The second point of concern is the type of solder you are using. We use only .71mm resin cored solder and with no additional skill from you, the constructor, the projects looks 100% better than using thick awkward solder. The solder joints come up cleaner, brighter and very little resin is left behind. I think you will be quite surprised at the improvement when using fine solder, and especially so for the clock.
1mm solder is also available in handy rolls and will perform an equally satisfactory job.

So please, please don't make a mess and then turn around and blame TE for a bad design because we know the clock circuit works and will be a valuable time piece in your workshop.





Three common PC faults.


COM
COM
COM
Q4
BC547
10K
220R
TO+
2K7
Q3
BC547
10K
10K
470R
470R
470R
470R
470R
470R
470R
13 12 11 10 9 15 14
IC5 4511
3
16
4
8
5
O C B A
6 2 1 7
1M
½ 4518
IC4 b
CE2
RST
8 1 3 7
Q1
Q4 Q3 Q2 Q1
14 13 12 11
IC4 a
½4518
9
CLK
16 10
15
RST
100K
11 6 7 10 12 13 9
16
3
CARRY
CLK
5
1
IC3
4026
RST
15
2 8
10 12 13 9 11 6 7
16
3
5
CARRY
IC2
CLK
4026
1
15
8
2
1M
10K
Q2
BC557
100K
10K
Q1
or 330k
33K
BC547
100K
470K
100K
RST
16
11
10n
10 CLK
IC1
4040
8
1 2 3 14 13 12 5
SET
SWITCH
9v
100K
1N4002
10n
100K
1000MFD
ALL DIODES 1N914 UNLESS SPECIFIED
Remove this resistor if the 4040 chip does not clock.
9v – 12v Plug Pack
Add 47R to reduce voltage to about 7v on the circuit.

HOW THE CIRCUIT WORKS

The clock circuit derives its accuracy by dividing the 50 cycles per second mains frequency into a digital readout. The SEC maintains a prescribed number of cycles per year and this is a fairly accurate method of driving a clock. After all, most AC clocks use a synchronous motor to drive the hands and they have been accepted as being accurate for most businesses and factories.

The input to the clock appears at the lower left hand corner in the form of a power transformer. The secondary should be about 9 to 15v AC at 100mA to 300mA. The AC waveform is passed through a voltage dividing network made up of two 100k resistors and the signal appears at the input of IC1, a CD 4040 binary counter IC. This counter has been decoded with 7 diodes to count to 3,000 and reset. These diodes form an AND gate and when all the 7 outputs are HIGH, the reset line will go HIGH due to the 100k pull-up resistor.

This occurs once every minute and this pulse is also passed to IC2, a CD 4026 decade divider and display driver chip. The 470k and 1M resistors provide a slight positive voltage on the input of the CD 4026 so that the chip will clock on each minute pulse. The 4026 is a display driver and each clock pulse will appear on the FND 500 display as MINUTES. The output of the chip is pin 5 and this is connected to the input of a second CD 4026. As you are fully aware, a clock displays 0 to 59 minutes and the second display is required to count 0 to 5.

To produce this count and reset sequence, we have diode detected segments e, f and g of the display. These three segments are characteristic to the figure 6 and when these lines go HIGH, the AND gate is allowed to go HIGH via the 100k pull-up resistor to turn on the BC 547 transistor. This in turn activates the inverter transistor, a BC 557, to provide the reset pin with a HIGH. The reason for the double inversion is due to the low voltage appearing at the output of the drive lines. This voltage would not be sufficient to reset the chip. The diode in the base line of the BC 547 transistor eliminates the slight voltage present at the anode ends of the AND gate, which would have the effect of turning on the BC 547 transistor. The output of IC3 passes to the clock pin of a BCD counter, a CD 4518 dual BCD counter IC.

This next stage uses a little magic. As you know, a clock is required to count to 12. If you study the counting sequence for the hours display, you will see it is required to display: 1,2,3,4,5,6,7,8,9,0,1,2,1,2,3,4,5,etc In the middle of this sequence we have an un-inviting 1,2,1,2, sequence. To produce this we have had to produce a few little tricks. The 4518 counter has BCD outputs. These have been diode gated into a BC 547 transistor so that when any of the 5 diode lines is HIGH, the transistor is turned on and does not feed IC 5 via the diode to pin 1.

IC4a counts normally from 1 to 8 when, at this stage, Q4 receives a HIGH. This HIGH is passed to the seond half of the chip which has its clock pin connected to earth and the clock enable pin connected to Q4 of IC 4a. By wiring the counter in this manner, it will increment on the falling edge of the waveform.

This situation occurs when the counter IC 4a passes from 9 to 0 and the resulting figures on the display show 10. The next incomming clock pulse produces an 11 on the display and everything up to now is quite normal. When the next clock pulse arrives at counter 'a', pin 12 goes HIGH and this resets the counter to zero via the diode between pins 15 and 12. This action can take place due to the pull up resistor providing a HIGH, because pin 3 is also HIGH and the diode between pins 3 and 15 will allow this rise to occur. Both counters are now set to zero and this removes the HIGHs from the 5 diode OR gate so that the BC 547 transistor is switched off. Its collector voltage rises due to the presence of the 2k7 resistor and this artificially puts a 12 on the screen via the diode into pin 1 of the 4518. It will also turn on the top BC 547 by supplying voltage into the base circuit via the diode and 10k resistor.

The next clock pulse to arrive will turn off this arrangement and display a 1 on the screen....exactly what we want.

A fast-forward feature is provided on the clock for setting the time. This is shown in the lower right hand corner. The SET SWITCH is really two pieces of wire which are touched with your finger to allow the 3Hz signal appearing at pin 5 of the CD 4040 to pass to the first CD 4026. This produces the slow clocking feature. If you touch the two wires very lightly, you will place the 4026 in its intermediate zone and this will produce a very rapid clocking of the displays, similar to fast-forward. By using this feature, you can set any time on the clock in a matter of moments.

The power supply for the clock is provided by a diode and 1000mfd electrolytic. This electro is mounted under the board to keep the board neat.

The only other 4 components which need mentioning are resistors. The two 10k base resistors on the BC 547 provide a load for the OR gate so that the voltage drops to zero when all lines are LOW.
The 1M resistor on pin 1 of the 4511 decks the input pin since it has two OR gate diodes feeding the pin and this will create a floating situation which must be tied low.

The 220R resistor feeding the hours digit is a voltage dropper for the '1' segments.

BEFORE ASSEMBLY

Before starting construction, sort out all the components in an orderly layout. Keep the integrated circuits wrapped up but make sure you have the correct types. Check for the letters FND 500 on the displays or look into the red diffusing screen and locate the decimal point. Pick out the BC 557 and the 1N 4001 diode. Tick off all the resistors against the parts list and keep the pen handy as you will need it to tick off each stage as you complete it.

(   ) Layout all the components neatly with all the resistors facing the one direction.

( ) Check that all the components are in the kit, or if you have purchased them separately, make sure that all in front of you.

(   ) Arrange for an AC power supply of about 12v AC to be available. The best choice is an AC plug pack as it is completely insulated. A small power transformer such as 2155 or 2851, properly insulated in a plastic box with power lead attached and earthed, will be quite suitable.

(   ) Collect up all the necessary tools, such as side cutters, pliers, small screwdriver, soldering iron, solder, iron stand.

(   ) Check the PC board for undrilled holes, blocked holes, broken lands or touching lands. The clock board is possibly the best PC we have produced and should not have any faults. But when you produce many thousands of PC boards, one or two faults invariably creep in. Make sure you have a good idea of where the parts go before starting.

(   ) Decide if you are going to use sockets for this project. This is always a very wise precaution. We haven't used them or supplied them in the kits as it adds to the cost and tends to make the completed project a little more bulky. If you have any doubts about your ability at soldering you should use sockets for the four chips.

CONSTRUCTION

The construcion of the clock is separated into 4 stages. Each stage adds one display to the readout. Upon completion of each stage, the circuit is thoroughly tested. The power is applied and the clock is allowed to run.

This is the first stage:

Refer to the photo and the overlay for the position of the parts for this stage.

( ) Fit the 1N 4001 diode. The bar or line on the diode goes "down".

(   ) Fit the IC socket for the 4040 so that the cutout on the socket (indicating pin 1) is near the edge of the board.

( ) Fit the three 100k resistors. Push them onto the board and splay their leads outwards so that they stay in position while soldering.

(   ) Fit the 470k resistor.

( ) Fit the two 10n capacitors. Push them close to the board and solder their leads.

( ) Make the SET SWITCH by creating a loop with tinned copper wire 2cm high. Solder it in position, then cut the top of the loop to create a small gap.

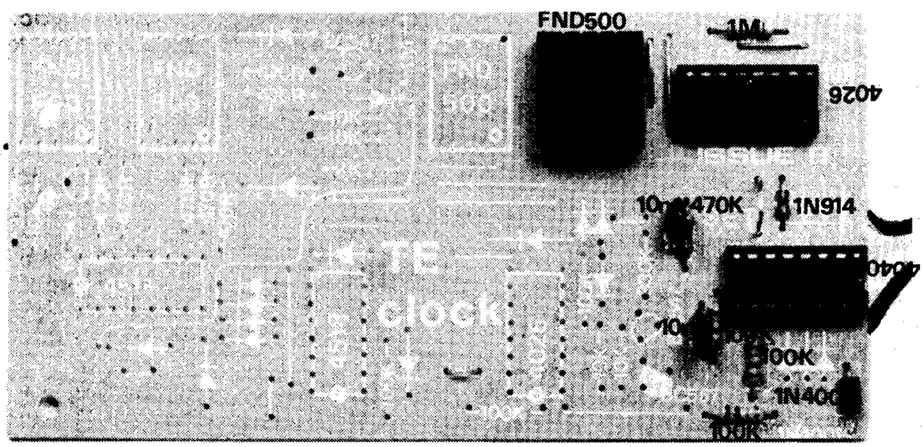
( ) Fit the Q4 diode, a 1N 914 diode. This will give a 3Hz clock pulse to the first 7-segment display.

( ) Fit the top 4026 IC socket. This chip has pin 1 near the KS symbol,

(   ) Fit the top 1M resistor.

(   ) Fit the link next to the 1M resistor.

(   ) Fit the 2 links between the minutes FND 500 and the 4026 socket.



( ) Fit the minutes FND 500, making sure the decimal point covers the dot on the layout.

( ) Fit a short link near the centre of the board towards the bottom, to supply the positive rail voltage.

( ) Fit the 1000mfd electrolytic. This is mounted on the underside of the board. Locate the square solder land beneath the 4040 socket for the positive lead and the thick land beneath the row of gating diodes for the negative lead. No holes are needed for the electrolytic. The leads solder onto the copper tracks.

( ) Fit the 4040 IC with pin 1 near the edge of the board. The numbers on the chip may be upside-down.

( ) Fit the 4026 with pin 1 near the KS sign. Do not go by the numbers on the chip, look for the indent at the end of the IC for pin 1 identification.

( ) Solder either lead of the AC plug pack to the large land beneath the row of diodes and the other lead to the earth rail, which is beneath the 1N 914 diode.
Add a 47R resistor or two 22R resistors to keep the voltage on the clock circuit to about 7v.

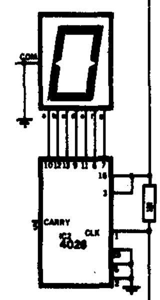
( ) Switch on. The display will clock at approximately 3 numbers per second.

( ) Once the minutes display clocks at 3Hz, you add the remaining gating diodes to create a MINUTES readout.

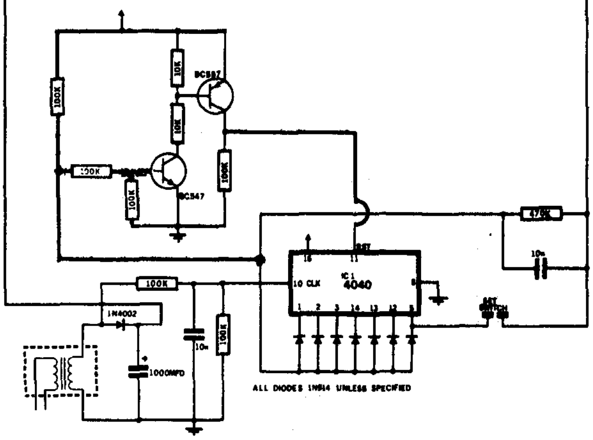
( ) Add the remaining 6 gating diodes, type 1N 914.

( ) Switch the clock on and wait for the display to change. Start timing the clock with your watch. Let the clock run for a few minutes and see if the display changes at minute intervals.

A 4020 chip can replace the 4040 in the clock circuit providing pin 4 of the 4020 connects to the diode which was previously conntected to pin 2. And pin 7 of the 4020 connects to the diode which was previously connected to pin 3.



This is the circuit modification:



THE CIRCUIT DIAGRAM FOR THE FIRST STAGE

If the 4040 does not clock at exactly 60 second intervals, the fault will lie in the self-resetting of the chip.

To produce a very reliable reset arrangement, a two-transistor circuit can be constructed on a piece of matrix board and fitted beneath the main board.

The circuit for this modification is shown above and has been found to cure all the stubborn problems.

THE ELECTROLYTIC IS SOLDERED TO THE UNDERSIDE OF THE BOARD AS SHOWN IN THE PHOTO.

QUESTION:
You have two clocks. One does not work at all. The other loses about 1 minute per day. Which clock is the more accurate?

ANSWER:
The non-working clock. It tells the correct time twice per day. The other clock is only accurate every two years!

IF IT DOESN'T WORK:

Connect the AC terminals of the LOGIC DESIGNER to the AC input of the clock in the following manner: Connect the ground terminal of the LOGIC DESIGNER to the earth of the clock and the top AC terminal of the Logic Designer connects to the anode of the 1N 4002 diode on the clock.

Connect a jumper to the clock pin of the 4024 on the Logic Designer via a 100k "stopper" resistor to be used as a test lead.

If the clock does not light up, check the link near the centre of the board for continuity. You can use one of the buffer transistors to check this. Check the voltage at pin 16 of the CD 4026.

( ) Check the 50Hz signal at the anode of the 1N 4002 diode with the clock input of the 4024. This binary counter should fill very quickly at 50Hz.

( ) Check for 50Hz at the clock input of the 4040 (pin10) with the input lead of the 4024 counter.

( ) Check for 25Hz at pin 10 with the test lead of the 4024.

( ) For the first section of the project, the 4040 produces a 3Hz signal via output Q4. Detect this signal on the 4024 binary counter.

( ) Check the anode end of the gating diode with a buffer lead. The 4024 may not detect this pulse as it does not rise high enough to clock the counter.

With the last two steps you have proven that the signal emerges from the 4040 and passes through the gating diode.

( ) Check the 3Hz signal at pin 1 of the CD 4026. The output pins 6,7,9,10,11,12,and 13 can be detected with a buffer lead. If the display fails to light up, check the soldering to all the display pins, the insertion of the display and the earth line.

STAGE 2:

This stage will create the minutes display reading 10's, 20's, 30's, 40's and 50's. The display is required to reset after 5 and not show the figure 6. To achieve this, we need a couple of transistors and some biasing components. After completing this stage, half the board will be complete.

( ) Fit the carry-out link at the top of the board, near the letters KS (if this has not alrady been done).

( ) Fit the lower 4026 socket. Make sure the pin 1 identification faces downwards.

( ) Fit the 4 1N 914 diodes.

( ) Fit 2 x 100k, 2 x 10k, 1 x 330k resistors.

( ) Fit 6 links to this section of the board as shown.

( ) Fit the BC 547 transistor.

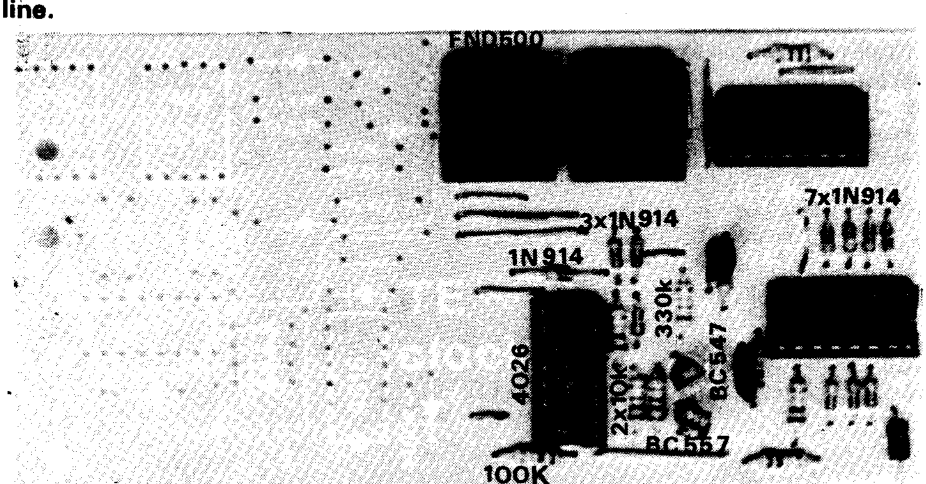
( ) Fit the BC 557 transistor.

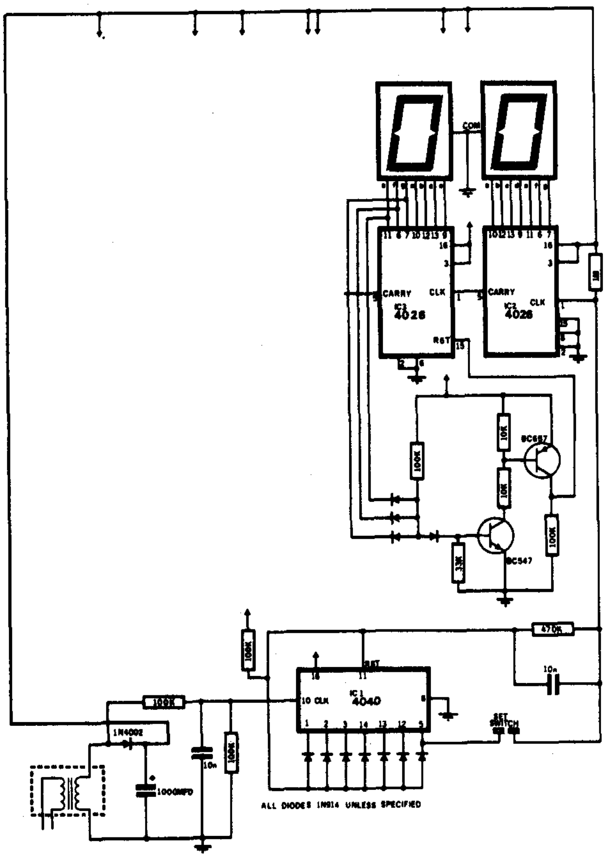
( ) Fit the FND 500 display.

( ) Insert the CD 4026 IC into the socket so that pin 1 is near the edge of the board.

Check all soldering for shorts between connections and make sure no leads are unsoldered.

Connect the power and join the SET SWITCH wire(s) together with a clip. The minutes display will count to 59 then reset.





THE CIRCUIT DIAGRAM FOR THE SECOND STAGE

IF IT DOESN'T WORK:

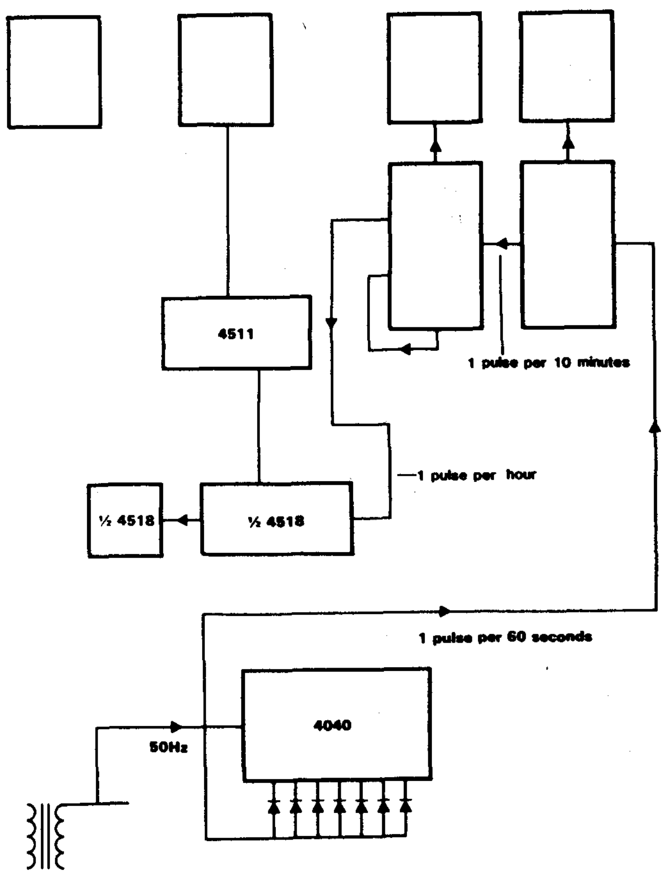
If the display is passing 5 and showing 6,7,8,9, the reset circuit is not operating. Firstly check the BC 547 and BC 557 transistors for correct insertion. At rest, the collector of the BC 547 should be very nearly rail voltage and zero voltage on the collector of the BC 557.

Check the operation of this circuit by powering the clock and take a 10k jumper lead from the positive rail to pin 15 of IC 3. If this resets the chip, take the 10k resistor from earth to the base of the BC 557. Next take a 10k resistor from rail to the base of the BC 547. These three tests should reset the chip.
To check the AND gate, remove one end of the three diodes and switch the clock on. If the display does not reset, the base diode is open or the 100k resistor is faulty.

If the display does not advance past zero, the reset line is remaining on. Remove the BC 557 and operate the clock. If the display remains on zero, check the 100k resistor in the reset line. Take pin 15 to earth with a jumper. The 4026 should now count. If not, the chip is faulty.

Place a jumper from the anode of the diodes to earth. If the display counts, a leakage voltage is present at this point. The base diode is designed to remove it and may be faulty. Replacve the base diode. If the display fails to change, check the voltage on the 100k resistor. It should be about .5v. If it is 1v or higher, check the cathode voltage of the three diodes. It should be less than .5v for a non-lit segment. You can put two diodes in series in the base line to remove the voltage from the AND gate. Short the 33k resistor with a jumper lead. If the display fails to change, replace the BC 547 and BC 557.

SIGNAL PATH



STAGE 3:

This section will add parts to drive the hours display. (It will not produce the complete 1 - 12 readout).

( ) Fit the seven 470R dropper resistors.

( ) Fit the two IC sockets. Make sure you know which end identifies pin 1.

( ) Fit the 1N 914 diode at the bottom left hand corner of the board, below the 4511 IC.

( ) Fit the 100k near the 4518 IC holder.

( ) Fit the 1N 914 diode next to this resistor.

( ) Fit the FND 500 display.

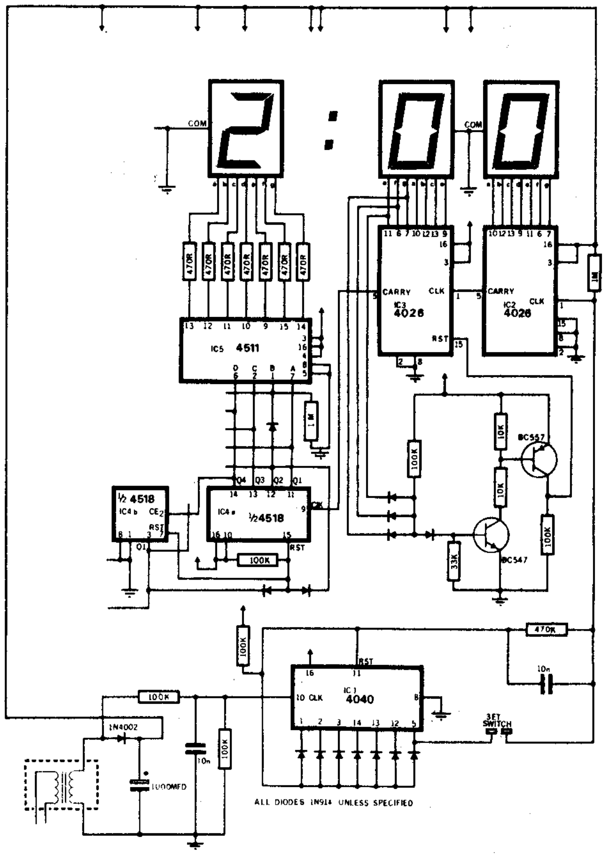
( ) Fit the 5 jumper links as shown.

( ) Fit the 4518 IC with pin 1 downwards as shown.

( ) Fit the 4511 with pin 1 near the 4 large staple holes.

( ) Check all the solder connection and see that all parts have been inserted as shown on the photograph.

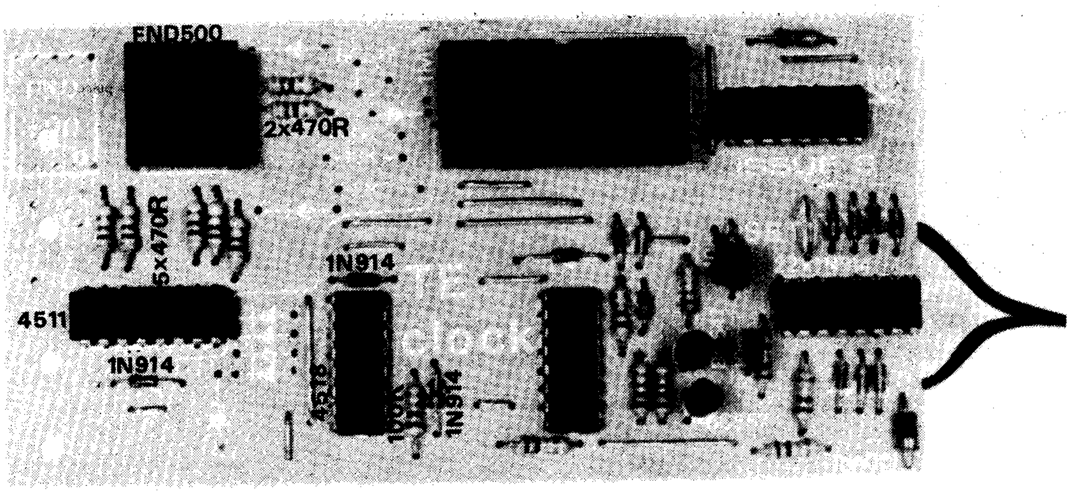
( ) Switch on the supply and note the illumination of the three displays.
They may not come on with all segments lit as the chips should be reset or clocked through one complete cycle. Allow the clock to fast forward by placing your finger very lightly on the SET SWITCH. The readout should clock 0 - 9 and this will indicate that all segments are operating.



THE CIRCUIT DIAGRAM FOR THE THIRD STAGE.

IF IT DOESN'T WORK:

If one segment fails to light up, check the 470R resistor for a dry joint or the display for a faulty segment. You can check each segment via a jumber lead and 1k resistor.
If the display fails to clock, check the voltage on pin 15 of IC 4. Deck pin 15 and re-test.



STAGE 4:

Stage 4 adds the special gating required to obtain a 1-12 readout.

(   ) Fit the long jumper wire running from between the displays to the bottom left hand corner of the board.

(   ) Fit the eight 1N 914 diodes.

( ) Fit six resistors: 1 x 220R, 3 x 10k, 1 x 2k7, 1 x 1M.

(   ) Fit the FND 500 display.

(   ) Fit 2 x BC 547 transistors.

FINAL TESTING

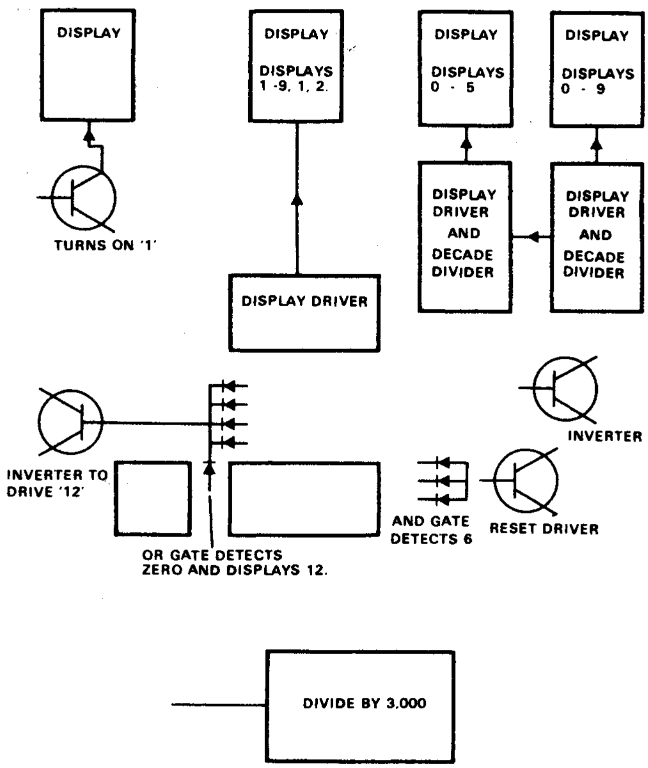
TESING Q3 and Q4.

Transistor Q3 is designed to force a 12 on the screen when the two counters are recording 00. By decking the base voltage with a jumper lead, a '1' will appear on the first display and the number in the second display will be increased by two if it is a 1,4,5,7,8, or 9. Otherwise numbers such as 2, 3 and 6 will not alter.

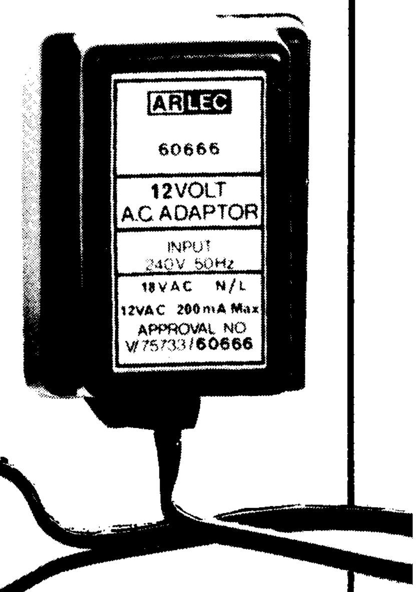
If a '1' does not appear on the first display, check transistor Q4, by placing a 10k resistor on jumber leads to positive and touch the base lead. If a '1' does not appear, replace the transistor. If it does appear, check the diode on the OR gate and the 10k resistor.

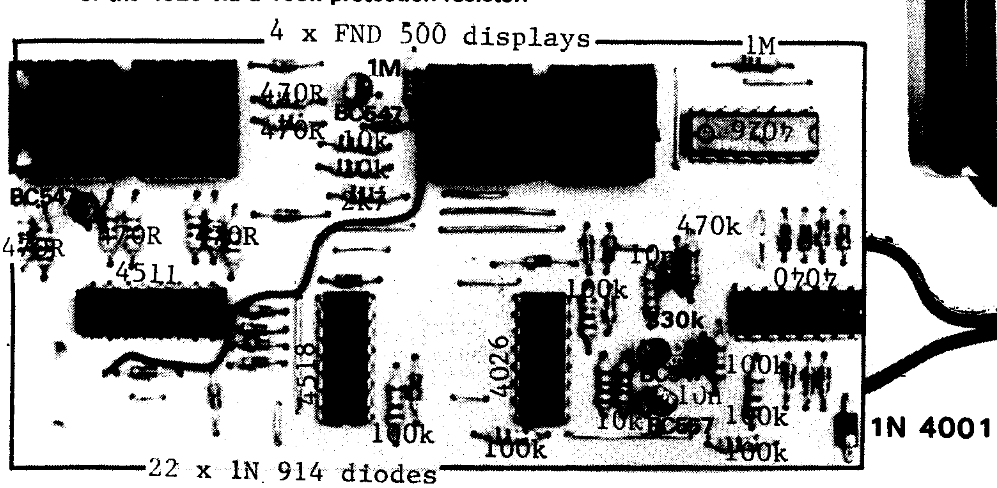
If the number in the second display does not increment as explained, check the gating diode between the collector and pin 1 of IC 5.

THE FUNCTION OF EACH ITEM



The clock can be fast-forwarded via one or two push switches in place of the SET SWITCH. The slow-forward switch is connected in place of the SET SWITCH and a fast-forward switch taken from the 50Hz to the input of the 4026 via a 100k protection resistor.





THE COMPLETED CLOCK

SOME COMMON DIODE FAULTS:

FAULT: The display does not show 10 or 11. It jumps from 9 to 12.

REMEDY: The diode between pins 12 and 15 is open or faulty.

FAULT: The display jumps from 1 to 12 to 1 to 12.

REMEDY: Diode between pin 3 and 7 is open or faulty.

FAULT: Display shows 9, 12, 11, 12, 1, 2, 3, etc.

REMEDY: Diode from pin 3 to AND GATE faulty or open.

FAULT: Display shows 1,2,3,4,5,6,7,1?,9,10, 11,12 1 etc.

REMEDY: Diode from pin 14 open of faulty.

FAULT: Display shows: 7,8,9,10,11,12,1,2,3 16,5,6,7,8 etc.

REMEDY: Diode from pin 13 faulty or open.

FAULT: Display shows 1,12,3,4,5,6,7,8,9, 010,11,12,1 etc.

REMEDY: Diode from pin 12 faulty or open.

FAULT: Display shows 12,13,2,3,4,5,6,7,8,9, 10,11,1 etc.

REMEDY: Diode from pin 11 faulty or open.

HELP, IT DOESN'T WORK!

We have covered some interesting and unusual faults. It could be that your unit develops other faults due to a combination of two or more components failing at the same time. I do not want to scare you, but to fix your own project will be the best instruction you can get.

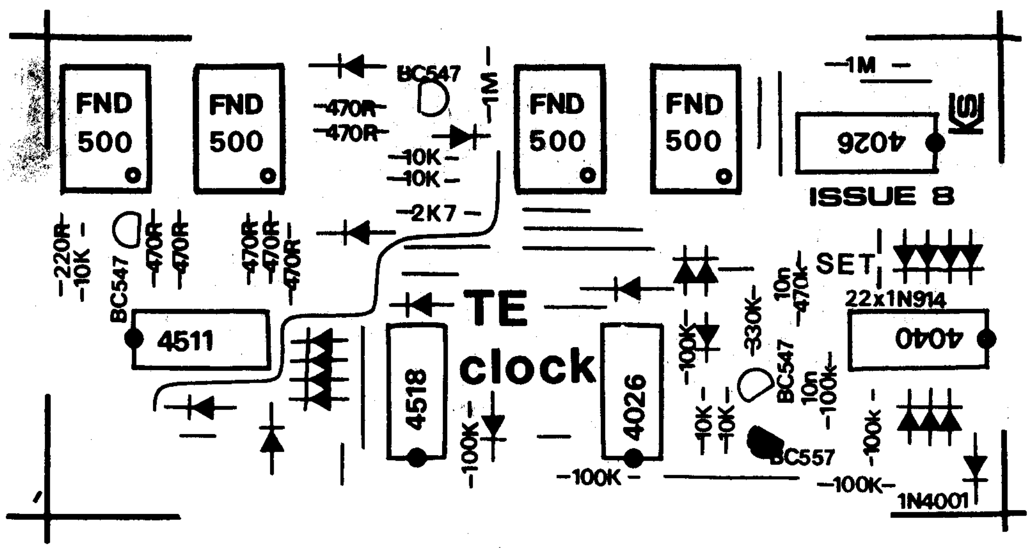
Approach the problem logically and try to narrow the fault down to a small part of the circuit. You will need a multimeter and either a Logic Probe or the LOGIC DESIGNER. Quite seriously, I hope your project does contain a small fault. After you find the fault and fix it, you will say "Gee, that TE clock project is really great". Take it from me, I know; I learnt the hard way. No-one showed me how to fix faults, and once you solve it yourself, you NEVER forget.

We receive a few letters every week from readers who solve a problem or two in their project and the feeling of achievement is really great.

If, however, you have tried to find the fault and it eludes you, we offer a repair service for the clock at a fixed fee. To be accepted for repair, the project must be completed on the PC board produced by us and must contain IC sockets for all the IC's. The soldering must be neat and all parts must be as per the overlay.

You do not have to send the power transformer but the project must be well packed in a jiffy bag with plently of foam styrene to protect the board.
Mark your jiffy bag:
CLOCK PROJECT,
Box 486,
Cheltenham, 3192.

Enclose your name and address and the $9.00 fee.



FAULT FINDING THE CLOCK

Some troubleshooting hints:

Our clock project produced a few problems for some constructors. A reliable modification appears on P 68 however here are some other solutions which have varying success.

THE 4040 COUNTER

Most of the problems with the clock occured in the first section. They revolved around the 4040 IC. In our prototype we used a CD 4040 or MC 14040 and no troubles were experienced. Our next batch of chips were Fairchild f4040 and this is when the trouble started. In the circuit diagram, the counter chip is designed to reset itself via its own inputs. . . when a combination of 7 outputs go HIGH. With the arrangement of self-resetting, the reset pulse is very short. Normally this arrangement would not be used but to conserve components, we have used it.

Unfortunately this short pulse was not long enough to fully reset all the internal flip flops in the Fairchild device.

This means that some of the stages had a 'count' still in them after the resetting operation. On the next cycle, the counter would need say 2,500 pulses instead of the full 3,000 to fill the counter before resetting. This meant it took less than 60 seconds to create a full cycle. The result was some of the projects were gaining time. Constructors rang to say the minutes display changed every 45 seconds or so.

Two solutions are possible: You can change the 4040 for a Motorola type or carry out a circuit modification.

WE have produced a mod to increase the pulse length (or effectiveness) to the reset pin.

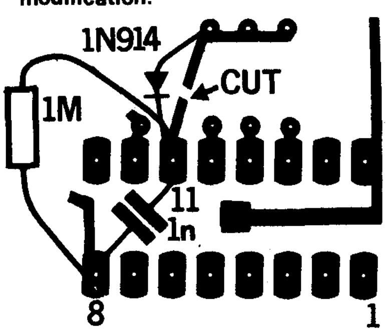
This mod was introduced when M. Warren and R. Martin had exactly the same fault. The minutes display did not clock at 60 seconds intervals.

So they brought their project to us for help.

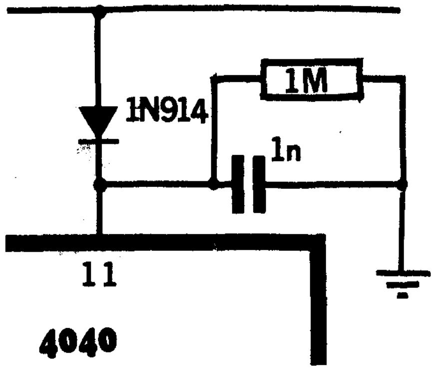
This is what we did:

We provided a diode in the reset line of the 4040 and a storage capacitor from pin 11 (the reset pin) to earth. This would lengthen the reset pulse and enable all the flip flops to reset.

The following diagram shows how to cut the track and fit the parts for this modification.



Mounting the components under the 4040 IC.



The modified circuit diagram. Don't use f4040 chips.

FAST-FORWARD TOUCH SWITCH

The fast and slow forward setting feature was a little vague to some readers. Let me explain it more fully: Basically we are using the 3Hz output pin for the slow forward and by touching your finger on the two touch wires, the signal passes through the low resistance of your finger to give the 3 cycles per second clock rate.

By lightly touching these two wires, a voltage divider is set up between your finger and the 1M pull-up resistor.

This action may bring the input of the IC into its linear region and cause it to clock rapidly. This becomes our fast-forward feature.

But this idea did not always work. Sometimes, due to chip characteristics, and sometimes due to individual finger resistance, the 2 clock rates could not be guaranteed.

The solution is to use two push switches. One from pin 5 and one from pin 10 of the 4040 IC.

✶✶✶✶✶

The fault with Mr Pauloff's clock was in the minutes display. It clocked 1,3,6,9,2,5,8,etc.

At each minute, the display would change from 1 to 3 to 6 to 9 etc. The minutes IC (IC2) was receiving noise pulses which clocked the chip rapidly. The most likely cause of trouble was in the slow clock pulse. Possibly it was due to the rising edge of of the clock being too slow and the 4026 was picking up noise pulses between the LOW and HIGH states.

The circuit components determining the condition of the clock pin of the 4026 and the 1M and 470k voltage dividing resistors. They provide a 'set' on the clock pin and this may be too high for the chip. Try swaping the two IC's or reduce the 470k or 10n capacitor.

Another unusual fault was this clock sequence: 9, 10, 11, 2, 3, 4, 5, etc. The 12 and 1 failed to appear.

This fault is due to the reset resistor between pins 10 and 15 of IC4a, being too HIGH in value. At 100k, it did not allow the voltage on the reset pin to rise enough to reset the two halves of the 4518.

The reason for this lies in the voltage divider network made up of the 100k resistor and the 10k resistor feeding the base of Q3. The voltage across the two diodes in this line (as well as the voltage drop across the base-emitter junction) was only just high enough to reset the IC. In some cases, the diodes drops were less and the resulting reset voltage was insufficient.

This resistor can be reduced to 47k or even lower to obtain reliable resetting.

4026's GETTING TOO HOT

The display driving capability of the 4026's varies enormously from one make to another. Some chips remain relatively cool throughout the construction testing and running of the clock, while others get fiercely hot. A simple method of reducing this temperature rise is to add a 10 ohm or 22 ohm resistor in each line of the 12v AC transformer. When the first stage of the clock has been completed, the voltage from the plug pack is about 10v and this is too high for the 4026. As more stages are completed, the voltage is down to about 7v due to poor regulation of the transformer. When the clock is complete, the supply is about 7v and the 4026's have a better chance of dissipating the heat. Even for a completed clock, a 10 ohm input resistor will reduce the heat generated by the driver chips.

✶✶✶✶✶✶✶

The clock is a very worthwhile project to construct. Because it is presented as a number of separate blocks, the operation of each of the divider stages can be understood. The CD 4026 chips are in plentiful supply at the moment as they have been imported in larger quantities due to the demand. If you are considering building a complex project, this one should be on your list.

We have plans for a CALENDAR to be connected to the clock to give a DAY, DATE readout for a whole year. See future issues of Talking Electronics for this.

The overall feedback from the Clock has been 'complete success'. Bill Street from Billco Electronics constructed his unit in a project box and is sitting on the sales counter as the main time reference. "It keeps perfect time" he said, when I called in to see his newly opened shop in Dandenong. Three other readers wrote in to say that their clock worked 'first go'.

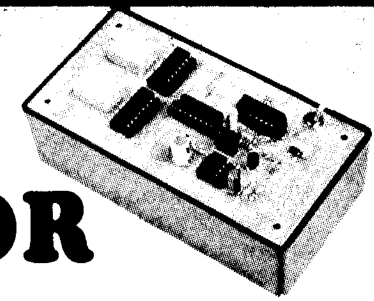
If you have any problems with this or any other project, just drop us a note or phone for advise. We will be pleased to assist.

We prefer you to ring for a couple of minutes with each small fault and phone back to say the remedy worked, rather than store up a lot of problems and hit us with them all at once. We don't even mind if you ring 4 or 5 times with different problems on the same project - we know how it is . . . we sometimes have problems too!

— Colin.

LOTTO NUMBER SELECTOR

By Ken Stone.



USE OUR LOTTO SELECTOR TO WIN A FORTUNE!

This project is a real winner - in more ways than one. When you understand how the circuit works, it is really 4 projects in one. And it has two modes of operation: MANUAL and AUTOMATIC.

Everyone likes a little flutter. The recent introduction of so many outlets in competition for the gambling dollar is positive proof of this. Every week the total prize pool for these games rises and this alone must draw in many new customers.
The possibility of winning something for nothing lures even the most cautious person into buying a ticket.
Nothing has been more successful than LOTTO. The concept of choosing your own numbers is brilliant. It has fooled the greater percentage of punters into thinking they are closer to picking a winner by this method, than buying a pre-numbered ticket.
Although nothing could be further from the truth, no amount of explanation will deter the avid invester from his weekly punt.
So, rather than being against them, we have decided to join forces and produce our contribution to selecting a winning combination. . . we have called our electronic number predictor:
LOTTO NUMBER SELECTOR.

This is our cover project. It will almost certainly create a fortune for someone and provide lots of fun in construction and operation.
Our circuit is a real gem. It looks simple but lurking within the 5 chips are a number of interesting building blocks.
The most significant feature of the circuit is the absence of 14 display resistors. Both the 4511's are display drivers and under normal conditions, a set of dropper resistors would be required. We have designed our circuit to eliminate them.
At the other end of the electronics ladder we have used a single-pole switch with a centre off to provide 2 functions.
To achieve ths we have had to insert the switch in the negative line. All these features are fully explained in the following pages.
For now, lets look at some of the mis-conceptions of chance.

★ LOTTO or POOLS selector - to help you win a fortune!
★ Single or Dual dice for games such as Monopoly.
★ Percentile Dice for war games or other strategy games.
★ As a random number generator for pure amusement.

THE EFFECT

When the power switch is turned on (to either MANUAL or AUTOMATIC), the two displays will show two figure '8's'. These will gradually slow down to a flicker and numbers will start to flick onto the displays. This will slow-down even further until double numbers can be identified. Finally, a random number will remain on the screen.

A BRIEF SUMMARY OF HOW THE CIRCUIT WORKS

When the switch is turned on, a Schmitt trigger oscillator supplies a 10kHz signal to a 4518 chip. This is a divide-by-ten counter with 2 separate stages. The output of these is in binary and these 4 lines of binary are fed to individual display drivers.
The numbers appearing on the two displays are randomly generated due to another slow cycling oscillator providing the halt condition. Between each number appearing on the screen, the high speed oscillator is generating up to 40 clock counts.

TWO-UP!

Take a simple penny (we will have to convert to a 20¢ coin for the younger readers however a penny has much more feeling and authenticity when it comes to gambling). A penny was used anywhere from a cricket field to the bar in a hotel for decision making. It provided answers to complex questions such as "who will shout next", "who bats first" or "who pays the taxi fare".
The chance of a coin landing heads is 50%. Thus it is obvious to everyone as being a fair way of solving a dispute.

A die or dice is also used in decision making and the chance of scoring a high number is one-in-six.

These are easy figures, However if I asked about the probability of selecting 6 numbers from a total of 40, most people would give an answer which would be so far from reality that they would be astonished.

Very few people understand high numbers. As proof, try a friend with this simple problem. Take a sheet of paper and tear it in half. Place the two halves on top of each other and tear the stack again. Repeat the operation 20 times. How high do you think the pile of paper will be? If I said it would reach the moon, would you be impressed? Such is the enormity of multiplication.

Because it seems so utterly impossible to create an enormous possibility with just 40 insignificant numbers, LOTTO has taken off from its very inception, and never looked back. Chance and probability is a fascinating mathematical study. One which can engross a dedicated mathematics student for his entire life.

The pseudo study of probability has been the downfall of many a punter as everyone thinks he or she is a good predictor.

Without the correct mathematical data, the casual backing of 'hunches' or 'certs' will eventually bring the novice to bankruptcy.

It is only by using probability correctly that you will increase your winnings. However the gain rate is only 2 to 5 per cent and few people are happy with low margins. They want big wins and quickly!

Don't think I am encouraging this form of wager. Just because I have presented a Lotto Selector does not indicate my acceptance of gambling of any kind. And yet by stating that, I am a hypocrite. Life is a gamble. Running a business is a gamble. Even driving to work or buying a product is a gamble.

Gambling itself is not a danger. It is only the excess of gambling which leads to ruin.

So, away from the preaching.

If you are against any form of gambling, you can use the LOTTO project to play a number of harmless games.

The two readouts can be considered a dual die, in which the numbers 1 to 6 are used, and any other numbers are ignored.

Other games such as war games or MONOPOLY require percentile readout and both digits can be used.

On the other hand you can use it on a personal basis to guess the next number to appear. With the switch set to the automatic position, you can use the project as a guessing game.

This is even more dramatic in a darkened room where the display will give the best results. You can even use it as a sleep inducer and try to stay awake until the batteries run down!

HOW THE CIRCUIT WORKS

THE SCHMITT OSCILLATOR

The heart of the LOTTO SELECTOR is a free-running oscillator. This is made up of a Schmitt trigger between pins 13 and 12 of the 74c14. It oscillates at approximately 10kHz due to the value of the frequency-setting components: the 10n capacitor and 4k7 resistor.

The output of the oscillator has a very short duty cycle due to the presence of the 1N914 diode. This means the ON time for the output is very short compared with the OFF time. The charging time for the 10n capacitor is provided by the diode and because it has a very small voltage drop, the capacitor is charged very quickly.

When the capacitor charges to ⅔ of the rail voltage, the trigger changes state and the output goes LOW. The diode is not reverse biased and does not have any effect on the discharge of the capacitor. The discharge time is provided by the 4k7 resistor and these two components are the frequency-setting items. The discharge-time to charge-time is appproximately 25:1. This duty cycle will not affect the counting of the decade counter chip (4518) but is as essential part of a very clever design. More on this later.

The 10kHz signal is passed to the clock pin of one half of the 4518. This chip is a decade counter and will divide the incoming pulses by 10. It is designed to give a readout of the numbers 0 to 10 in binary form and this requires 4 output lines as shown in the diagram.

The highest priority line (pin 14) is then taken to the clock input of the second stage. The result is a counter capable of counting to 100.

Each of the outputs consist of 4 lines of binary information of a decimal number. Thus it is called BINARY CODED DECIMAL.

These outputs are passed to a 4511 display driver chip which is designed to convert the binary information to 7 lines of information to drive a 7-segment display. One chip is fully employed doing this because of the number of pins required.

Another feature of the 4511 is the LATCH or FREEZE capability.

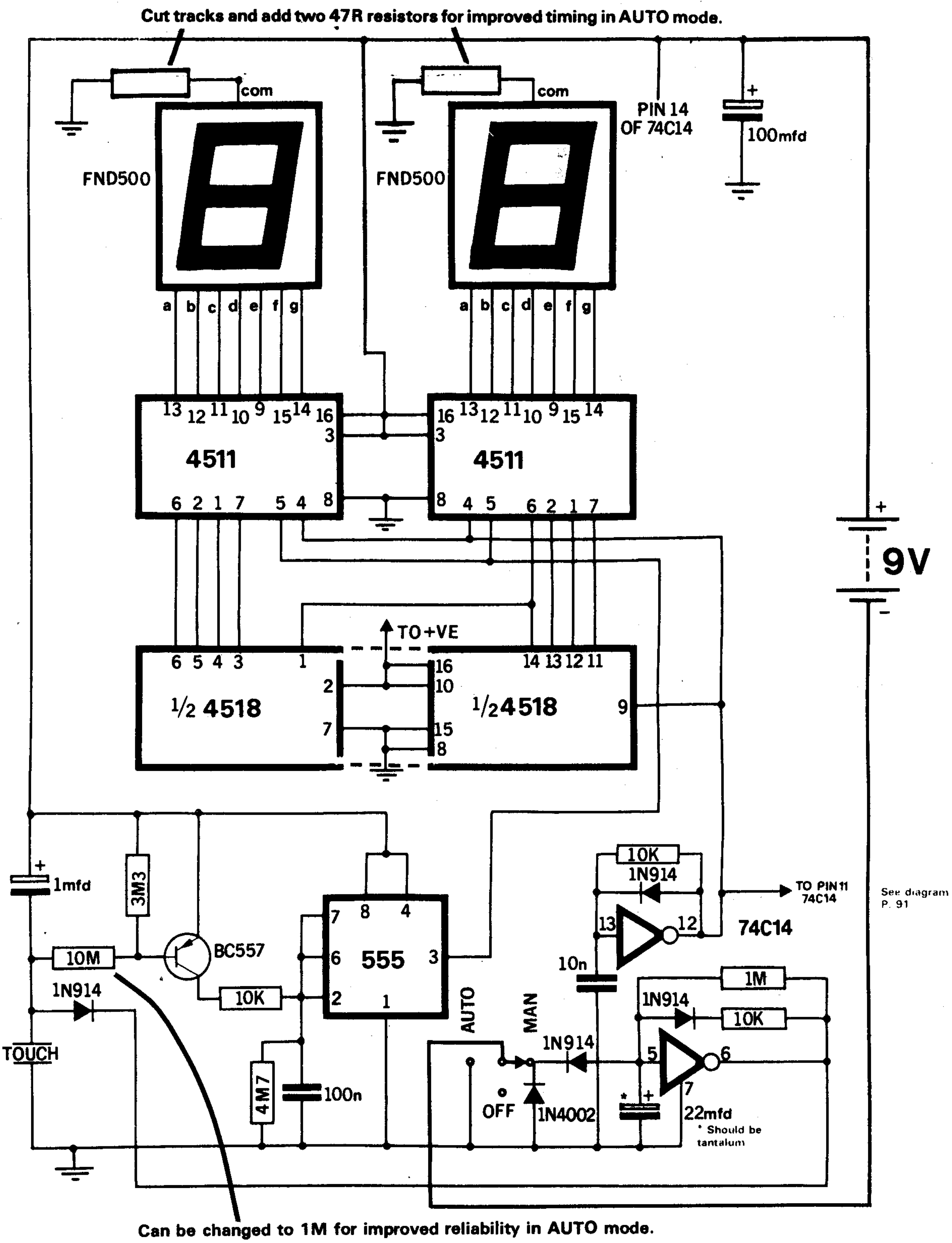
A number can be frozen on the display while another is being set-up on the input lines.

Two pins control this effect.

One pin (pin 4) is called the BLANKING INPUT pin. It has the effect of turning off all the segments when it is LOW. This means it is ACTIVE LOW. (It produces an effect when it is LOW).

The other pin (pin 5) is the LATCH ENABLE pin. It produces the freeze effect when it is HIGH because it enables the latch (opens the latch) when it is LOW.

This is what happens: If pin 4 is LOW, you will not be able to see anything on the display at all. When it is HIGH, the figure on the display will depend on the values of the incoming BCD lines and also the state of the LATCH ENABLE pin number 5.



The complete LOTTO circuit. The schematic diagram closely follows the layout on the PC board.
Remember: The 4518 is shown in two halves on the diagram.

. . .HOW THE CIRCUIT WORKS

If pin 5 is LOW, the numbers will change on the display according to the change in the incoming information. When it is HIGH, the display will freeze and the incoming information will not get through the latch circuit.

Now imagine the blanking pin being turned on for 4% of a cycle and off for 96%. This is occurring at 10,000 times a second, when the LOTTO project is switched on. Because the speed of this flashing is too fast for us to see, we think the display is on all the time. But electronics is faster than the eye. The display (made up of 7 light emitting diodes) will respond to this speed and they will turn on and off without being damaged.

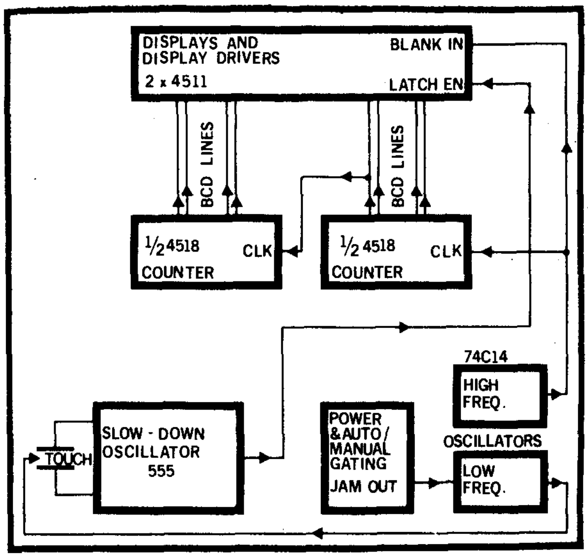
The reason for this clever circuit arrangement is very interesting. You will notice we have not included any display dropping resistors between the display and the driver chip. Normally we would require 7 resistors of about 470 ohm for voltage dropping to each display. And this circuit would need 14 resistors. We have eliminated them entirely.
How clever! This has saved parts, space and layout problems. The reason for choosing this method of operation is two-fold. We avoid the wasted power produced by display droppers and secondly, the display is allowed to operate at a more efficient level.

To achieve a good level of brightness, it is possible to drive the displays with 4 times the normal current for a very short period of time. This produces a bright display because the light emitting crystals have a higher efficiency at higher currents.

The next feature to look at is the number changing circuit.

This is accomplished by briefly bringing pin 5 LOW, then HIGH again. The new number appearing on the BCD lines will be frozen by the latches and displayed on the FND 500's. A 555 is used for the number changing operation because it can be designed in a circuit to have a very short pulse width, can be made to slow-down and is guaranteed to go LOW at the completion of its cycle.

The slow-down operation is accomplished by a BC 557 transistor. This transistor is being turned on by the charge (voltage) on a 1mfd electrolytic and the transistor is acting as a variable resistor. The transistor and 10k make up the charging resistance for the 100n capacitor. The discharging resistor has been eliminated and this means the LOW time for the cycle will be very short.



To start the slow-down oscillator functioning, the 1mfd electrolytic is charged slightly when you place your finger on the TOUCH WIRES. This voltage is passed to the base of the BC 557 transistor via the 10M resistor and a 'turn-on' condition occurs between base and emitter. The 3M3 resistor is a bleed resistor to slowly discharge the 1mfd electrolytic. This causes the 'effective resistance' between the emitter and collector to change and the 555 responds by changing its clock rate. The 4M7 across the 100n timing capacitor ensures the voltage on the 100n decays to zero and prevents the 555 from giving out random clock pulses once it has stopped.

The circuit is designed for AUTO or MANUAL operation.
In the manual position, the touch switch comes into operation and you can throw your own numbers by touching the TOUCH SWITCH.

In the AUTOMATIC position, The LOTTO SELECTOR will dial up a number, display it for a few seconds, then start counting again. This automatic control is governed by a long time delay created by the Schmitt Trigger between pins 5 and 6. Its repetition rate is controlled by the 1M resistor and 22mfd electrolytic Normally the output of the trigger inverter is HIGH. This causes the 22mfd electrolytic to charge up via the 1M resistor to ⅔ rail voltage. The trigger circuit changes state and the output goes LOW. When this happens, the 1mfd electrolytic in the slow-down circuit is charging up via the diode and this has the same effect as touching the TOUCH WIRES.
The 22mfd electrolytic discharges via the 1N 914 diode and the 10k resistor fairly quickly and this gives a brief pulse on the 1mfd electrolytic. When the voltage on the 22mfd electrolytic falls to ⅓ of the rail voltage, the output of the trigger inverter goes HIGH. The slow-down circuit comes into operation to give you a new number. This repeats itself ad infinitum.

Lastly a clever trick with the on-off switch.
So that the long delay timer does not come into operation when the Selector is set to Manual, we have held the voltage on pin 5 LOW via a diode. It may not be easy to see, but the 1N 4002 is positioned so that it supplies the rest of the circuit with power when switched to the MANUAL position. In this setting, the 1N 914 diode is connected to earth and pin 5 cannot rise more than .5v, thus the trigger circuit will not cycle. This arrangement could not be done in the positive line as the Schmitt trigger must be held LOW for its output to remain HIGH.

ASSEMBLY

All the components fit on the top side of the printed circuit board and each part is identified on the overlay. Your task is to learn about identifying the parts while constructing the project. So take your time.

Lay the components on the work-bench and make sure all the parts are present by referring to the parts list. Place the resistors around one way and grade them into ascending order. Don't make a mistake between the 1M and 10M. Check the difference between blue and green.
The first items to fit onto the board are the jumper links. These are made from tinned copper wire which has be straightened to remove any kinks. Start at one end of the board. Cut a length of wire and bend it into the form of a staple or 'U' shape. Fit it through the holes in the board and solder each end with a hot soldering iron. When soldering is completed, the jumper link should be touching the board and the wire should be straight. Complete the 12 links and two TOUCH WIRES with the same type of tinned copper wire.
Next components to mount are the diodes. These are sometimes hard to identify and an important point to note is the COLOUR of the band or bands is NOT the major identification. It is either the thickness of one of the end bands or its position at the end of the diode.

Take these examples:

A diode with a blue painted band indicates the cathode. Do not take any notice of the red lead inside the glass bead. This is purely the colour of the terminating cup for the crystal.
A diode with a thin brown line, a thin yellow line, a thin red line and a thick yellow line is identified by the thick yellow line being the cathode.

The 8 resistors are the next components to fit onto the board. The leads should be bent close to the end of the resistor but not so sharply that the lead is likely to break off. Insert the resistor until it touches the PC board. You should be able to hold your fingers on the resistor while you tack one end with the soldering iron. The other end can then be soldered and the first end re-soldered to create a perfect connection.
Double-check the value of each resistor before going on to the next to make sure a mistake has not been made.

PARTS LIST

2 - 47R (for modification)
3 - 10k
2 - 1M
1 - 3M3
1 - 4M7
1 - 10M

1 - 10n 100v greencap
1 - 100n 100v

1 - 1mfd 16v PC electro
1 - 22mfd 16v electro
1 - 100mfd 16v electro

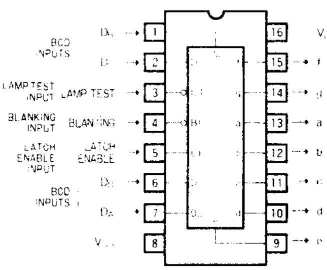
1 - BC 557 transistor

4 - 1N 914 diodes
1 - 1N 4002 diode

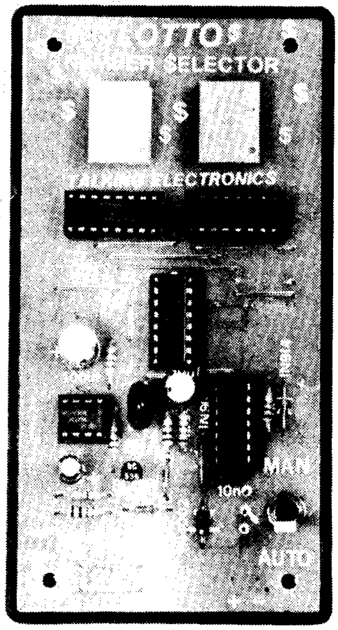
1 - 555 timer IC
1 - 74c14 (40014) IC
1 - 4518 dual BCD counter IC
2 - 4511 display driver IC's
2 - FND 500 displays
1 - 8 pin IC socket
1 - 14 pin IC socket
3 - 16 pin IC sockets

1 - ON-OFF-ON switch
1 - battery snap
tinned copper wire

LOTTO PC BOARD



4511BP

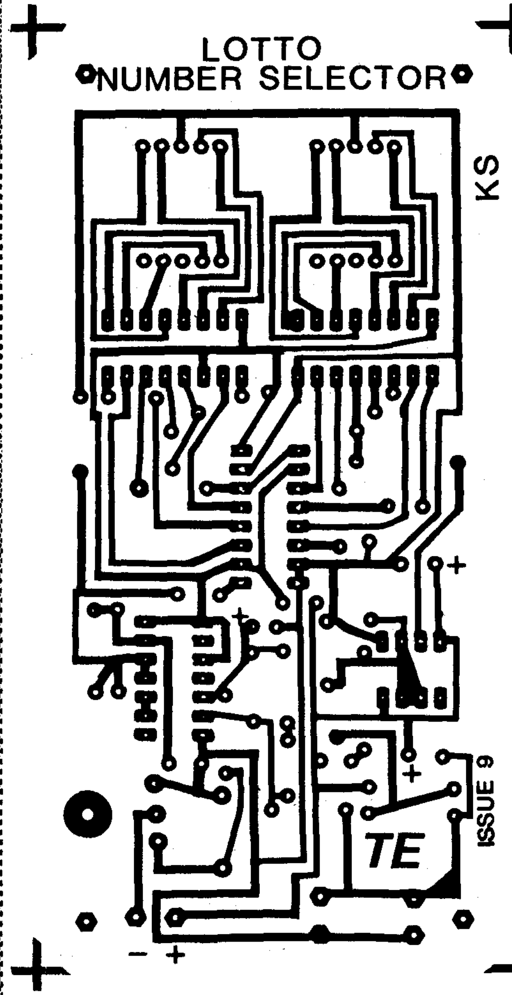


Complete LOTTO fits onto Zippy box.

Fit the 5 IC sockets so that the pin 1 identification on the sockets covers the dot on the PC board. This will make it easier to insert the chips around the correct way at the completion of the project. Solder each pin cleanly and swiftly, making sure the lands do not bridge with solder.
The two FND 500 displays are identified by the decimal point and these are soldered in position as shown in the photograph.
The 1N 4002 power diode is mounted so that the line on the diode case corresponds to the cathode lead on the overlay.
The 3 electrolytics are fitted so that the positive lead (the longest lead) fits down the marked hole. Take care when doing this because the marked lead on the case of the electrolytic is the negative lead.
The two greencap capacitors can be fitted around either way and are soldered in position so that they touch the board.
The BC 557 transistor fits directly into the 3 holes on the PC board.
Connect the ON-OFF-ON power switch through the hole in the PC board and use tinned copper wire to connect the 3 leads to the board.
A battery snap finishes the construction of the project.
All that is left to do is fit the 5 chips into their sockets with pin 1 covering the dot on the PC board.

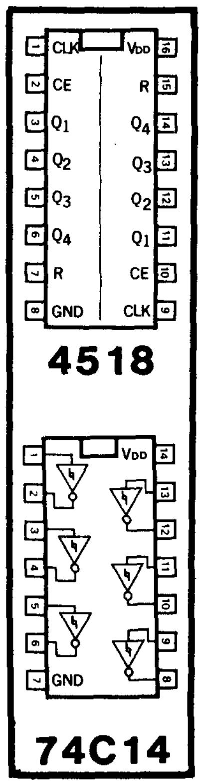
The LOTTO SELECTOR is now ready for run-up.

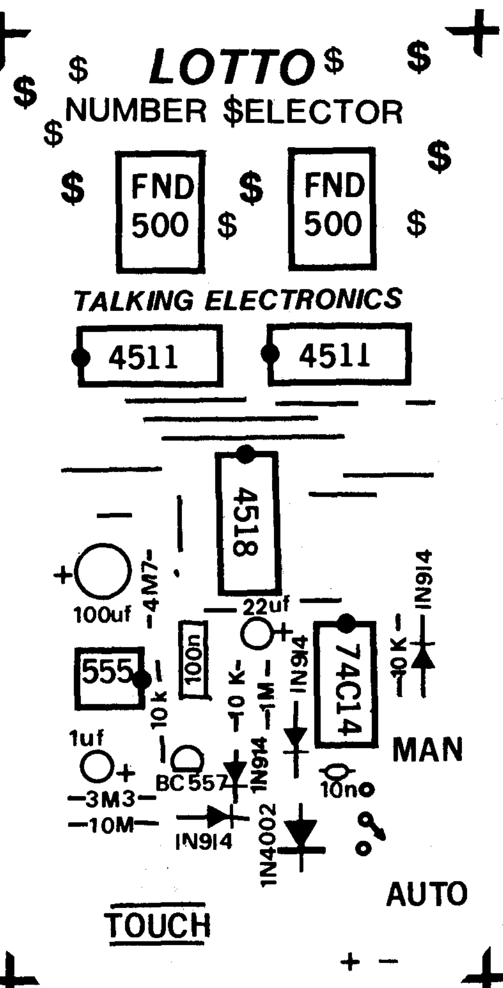
A 9v transistor battery or 6 penlite cells in a holder is recommended for the first trial run. Nicads can deliver a very high current and may cause damage if a short circuit is present.
After the selector is found to be working properly, a set of nicads can be used as the power source.



LOTTO PC PATTERN

PIN OUTS





OVERLAY WITH PARTS IDENTIFICATION

EXPERIMENTING WITH THE LOTTO SELECTOR

A number of features of the LOTTO SELECTOR can be examined with the aid of a LOGIC DESIGNER. This handy piece of equipment is covered on P 19 and is really an essential addition to your workshop.

If you have not built yet built the LOGIC DESIGNER, here is a run-down of its capabilities.

It is basically a digital man's CRO. It will let you know when a digital circuit is working correctly and will be much more use than a multimeter.

In some instances it will be more informative than a CRO since it will tell you when the amplitude of a pulse is sufficient to drive the input of a counter.

The LOGIC DESIGNER also has a division stage so that frequencies up to about 5kHz can be counted directly. A one-shot circuit produces individual pulses so you can slow down the operation of a circuit so that it can be understood. The Logic Designer lets you "see" into the workings of a circuit.

Here is how to connect the Logic Designer to the LOTTO project:

Set the Logic Designer to 9v, and supply the Lotto with a separate 9v, such as a battery.

To test the Lotto project you will need a common line from the Logic Designer and this is done by taking a jumper lead from the pin marked GND on the Designer, near the bridge diodes, and connecting to the negative on the Lotto at the centre of the switch.

Alternatively, the Lotto Selector can be connected directly to the Logic Designer between the positive take-off point (marked with a + at the top of the PC board) and ground GND. A test lead can now be taken from any of the inputs on the Logic Designer to the Lotto board.

The Logic Designer can be set to provide a divide-by-1280 by connecting the 'C' output of the 4026 to the clockinput of the 4024 binary counter. The clock input of the 4026 has a jumper lead attached and this is used to probe the output of the high-frequency oscillator (74c14)

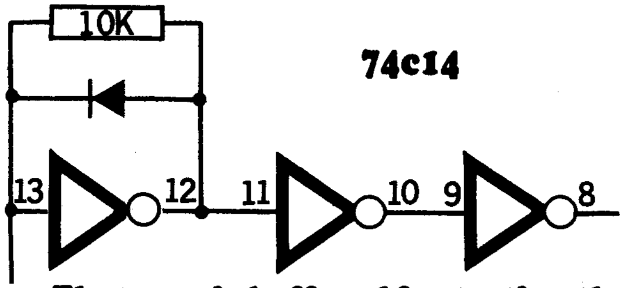
The 74c14 has been buffered so that you can read the output frequency. Do not probe pins 12 or 13 as this will kill the oscillator and the displays will light up like candles. Refer to the following diagram and probe pins 10, 9 or 8 to detect the presence of a high frequency waveform.

Place the probe lead on pins 11, 12, 13 and 14 of the 4518 and notice the different frequencies of the BCD lines. An even higher division is available from pins 3, 4, 5 and 6.

By using pin 6 and the 1024 division of the Logic Designer, you will be able to determine the frequency of the 74c14 oscillator.

Look at the binary counter outputs on the Logic Designer. The highest division is labelled '64' and this indicated that it illuminates after a count of 64. It remains on for 64 counts and thus a complete cycle for this LED is 128. We have connected the Logic Designer to represent a division of 1280 by connecting the decade divider (4026) to the input of the binary counter.

Further stages of division are provided by the chips on the Lotto Selector. Each half of the 4518 is a divide-by-ten making the total division 128,000.



The 74c14 is buffered for testing the 10kHz oscillator.

The high frequency oscillator will be running at 10kHz to 14kHz and you will be able to see this frequency being divided down to a point where you can actually count the cycles.

Attach a jumper lead to pin 6 and the 4518 and take this to the clock line of the 4026. The 'C' output of the 4026 is taken to the clock of the 4024. The count point is LED '64'. Start the timing operation when this LED comes on and this is counted as zero. The next time it comes ON, it is counted as 1. Keep counting until 10 and stop.

Use this formula to determine the frequency of the oscillator:

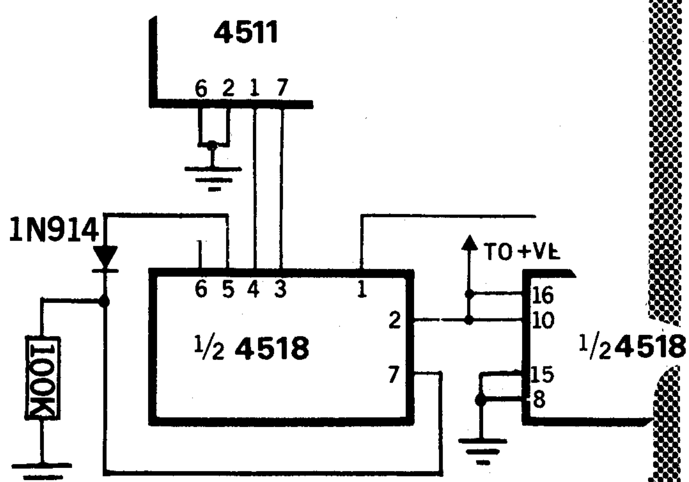
$$\text{Freq} = \frac{10}{\text{time in sec.}} \times 128{,}000 \text{ Hz}$$

The frequency of the slow-down oscillator can be detected on the Logic Designer by touching pin 3 of the 555 on the Lotto board with the jumper lead.

When you touch the TOUCH SWITCH, the Logic Designer display will tick over synchronously with the Lotto displays. The binary counter will show exactly how fast the 555 is clocking.

The Lotto Selector can be modified to count to 40 instead of 0-99. This requires only a small amount of track cutting between the 4518 and 4511 chips displaying the "tens" and the addition of a diode and resistor to create a reset condition on the count of 40. The double zero is read as 40.

The modified circuit diagram is shown below: ıge.



COUNT-TO-40 MODIFICATIONS

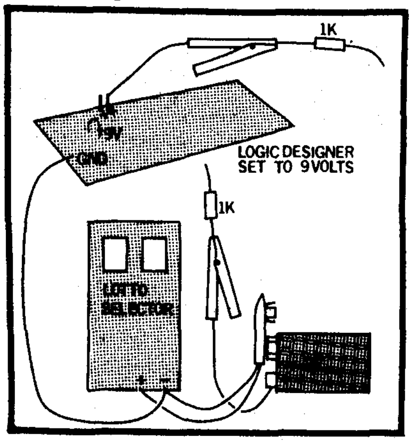
IF IT DOESN'T WORK:

If the LOTTO SELECTOR doesn't work, you will need a multimeter and the LOGIC DESIGNER to get into the signal side of the circuit.

The illumination of the displays will be the deciding factor on where to look.

IF THE DISPLAYS DO NOT LIGHT UP

Connect the negative terminal of the battery to the project and use a jumper lead connected to the positive terminal as a test lead. Place a 1k resistor in line with this test lead and switch the project on to MAN or AUTO. The Lotto project will not be powered by the battery for this test and, in fact, the circuit must not be operating for this part of the test as the outputs of the 4511 will short-out our test-voltage.



Touch the 1k resistor along the top row of pins for each of the 4511's (pins 9 to 16 and you will see each segment of the display light up in turn. If they do not light up, either the displays are faulty or the common earth lead is not connected to ground. You can also try these pins when the 4511's are removed. If the displays only light up when the chips are removed, the 4511's may be faulty or damaged.

If the displays check out ok with the 1k test resistor, but still fail to light, the fault may come from the BLANKING line. This is pin 4 of the 4511's. If this line is held LOW, the displays will not light.
You must be very careful when checking pin 4. You cannot let pin 4 float nor can you put a HIGH on pin 4 because the displays are directly coupled to the driver chips and may burn out if full rail voltage is applied to them. Pin 4 is normally receiving a very short pulse and only this short duty cycle can be duplicated for the correct operation of the set. A CRO will show you the mark-space ratio of the incoming pulse but in the absence of this piece of test gear, you can use the Logic Designer as described elsewhere in this article.

A failure of the Schmitt trigger oscillator between pins 13 and 12 of the 74c14 will cause the displays to remain unlit if pin 12 is LOW or if pin 13 is HIGH. Pin 13 may be touching pin 14 or receiving a leakage from the 9v rail. The 10k resistor may be faulty and failing to discharge the 10n capacitor. The Schmitt inverter may be damaged. If this is the case, you can use one of the unused inverters in the package.

IF THE DISPLAY LIGHTS UP LIKE A CANDLE:

If the displays light up far too brightly, the fault lies in the high frequency oscillator betweeen pins 13 and 12 of the 74c14. Turn the Lotto Selector off immediately and add a 100 ohm resistor in one line of the battery. You can now trace through and find the fault without burning out the displays. Use the Logic Designer to detect the frequency of this oscillator by connecting to either pins 8, 9 or 10. You will NOT be able to detect the pulse on the output of the oscillator (pin 12), so use the buffer stages provided. The displays will light up if the oscillator is jammed in the HIGH output mode. This means the input (pin 13) will be LOW and this could be due to a short in the 10n capacitor, a short between the leads or a leakage path to earth. It could also be the chip itself. Another possibility is the failure of both the 10k resistor and diode. This will result in the 10n capacitor failing to charge up.

Many of these possibilities are highly unlikely but it could be a fault in the soldering of the 10k resistor and diode which has left the charging line open. These things do happen. We have found hairline cracks in the PC linework, fine hairs of solder touching adjacent tracks and hard-to-detect dry joints. So don't be surprised if you find the trouble turns out be microscopic!

IF THE NUMBERS DON'T CHANGE

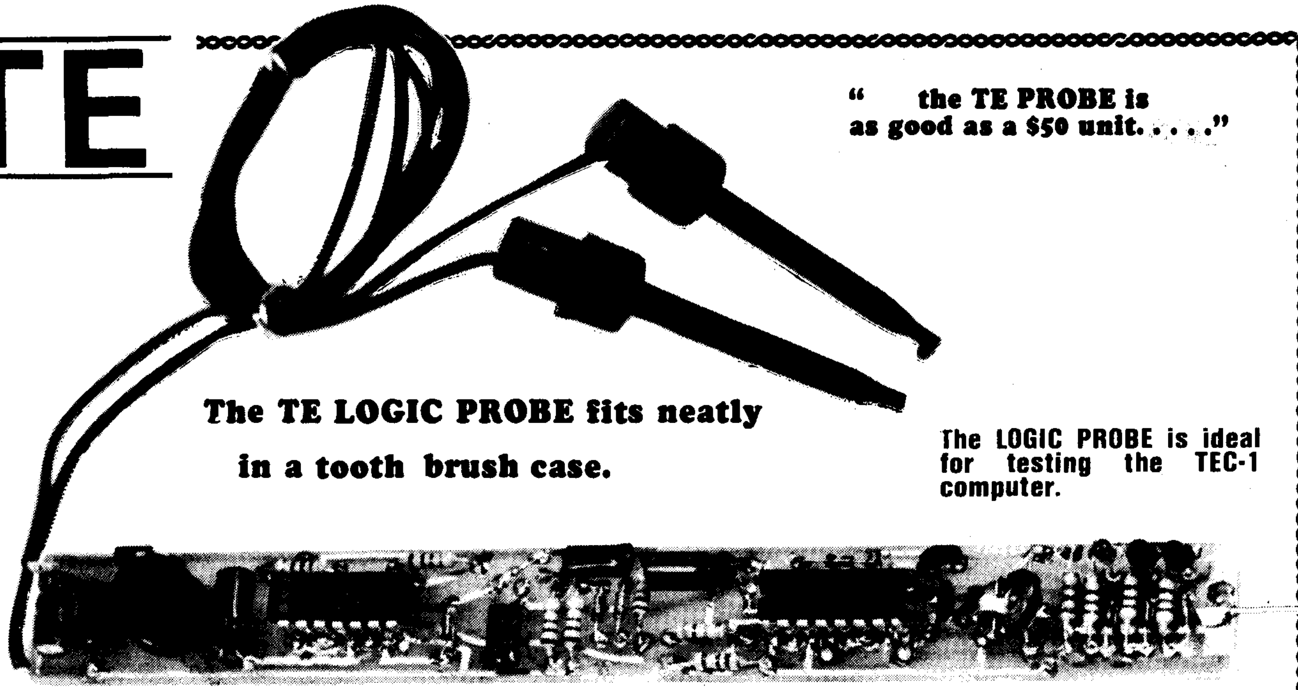
The numbers on the display change when the LATCH ENABLE pin 5 is LOW. The pulses from the 4518 can now pass through the display driver chip (4511) and alter the numbers. A HIGH on pin 5 will freeze the numbers. Both LATCH ENABLE pins are driven from the output of the 555 and the fault could lie within this oscillator. Test the operation of the 555 by placing a 100k resistor on jumper leads and connect one to the positive of the battery. Touch the other onto pin 7 of the 555. You should see the numbers change fairly rapidly. A 1M resistor will make the numbers change at a slower rate. If the numbers do not change, the fault will lie in the 100n capacitor being open (dry joint) or the 4M7 is the wrong value (take it out for this test). Pins 2 6 and 7 have a leakage path to earth or are touching earth. The 10k resistor is the wrong value or is touching earth. If the 1M resistor produces a change in the numbers on the display, try both ends of the 10k resistor: the effect should be the same. If not, the 10k could be open.
Next place a 1M resistor between the base of the BC 557 transistor and earth. This will turn the transistor on. If nothing happens, the transistor will be faulty. Try another PNP transistor. Make sure it is a PNP type!
Finally try both ends of the 10M resistor with a 100k resistor on jumper leads. If the TOUCH SWITCH side of the 10M resistor does not work, it may be an open resistor, the 1mfd electrolytic may be shorted to the positive line of the 1N 914 diode may be reversed or shorted.

IF THE DISPLAY DOESN'T STOP

If the display keeps ticking over and does not finally come to rest in the MANual position, it may be due to leakage in the BC 557 transistor, leakage across the TOUCH lines, or leakage in the 1N 914 diode. Remove the diode, lift one end of the 10M resistor, lift one end of the 3M3 resistor. If the ticking still occurs, the transistor will be leaky. Another possible fault is the 100n capacitor not fully discharging. The 4M7 is designed to carry out this operation. Try a 1M resistor across the 100n capacitor. Pins 2, 6 and 7 may have a leakage path to positive rail. Check your soldering and track work.
If the display doesn't stop in the AUTO mode, the fault will be due to the time delay circuit made up of the inverter between pins 5 and 6, the 22mfd electrolytic and the 1M resistor. You can use a multimeter to see when the output pin (6) is HIGH. It should remain HIGH to allow the numbers to gradually slow down. If it goes LOW, the numbers will speed up again.
To lengthen the time delay, the 1M resistor can be increased. But firstly try another 22mfd electrolytic as these electros require a "forming" voltage on them to produce their full capacitance. After a few charge-ups, the capacitance increases.
To increase the time delay, use a 1M5 resistor as the charging resistance. If the display keeps cycling in the MANual mode, the gating diode on pin 5 of the inverter will be open or have a dry joint. If the display does not cycle and produce a new number in the AUTO mode, the delay timer is not operating. Check the output pin 6 with a multimeter for a change from HIGH to LOW after 10 or 20 seconds. If this does not occur, the timer is not operating. The fault may be due to leakage within the 22mfd being higher than the charging current and consequently it never reaches its ⅔ Vcc value. The 1M resistor may be the wrong value (say 10M), the 1N 4002 diode may be leaky (remove it and see if the problem is cured) or the chip itself may be faulty.

I hope you don't have any insurmountable problems with the LOTTO project but it would be nice to have a small problem and need to use either a multimeter or the LOGIC DESIGNER to locate the fault.

TE LOGIC PROBE



A LOGIC PROBE is possibly the most important of all pieces of test equipment for the digital designer.

Comparatively speaking, it is equivalent to an electrician's multimeter or an audio technician's CRO.

You may be wondering why we have not described a Logic Probe before. Well, basically, there are two reasons. Firstly we have not had the need for it and secondly the complexity of the circuit has not fitted the scheme of things.

But the time has now come for its application. This is our computer issue and to build a computer without having access to a PROBE is like taking the car on a long journey without a spare tyre.

In fact, with the computer project, the circuit is tested at various stages with the probe to make sure it is functioning correctly.

So, to be without this piece of test equipment is digital suicide.

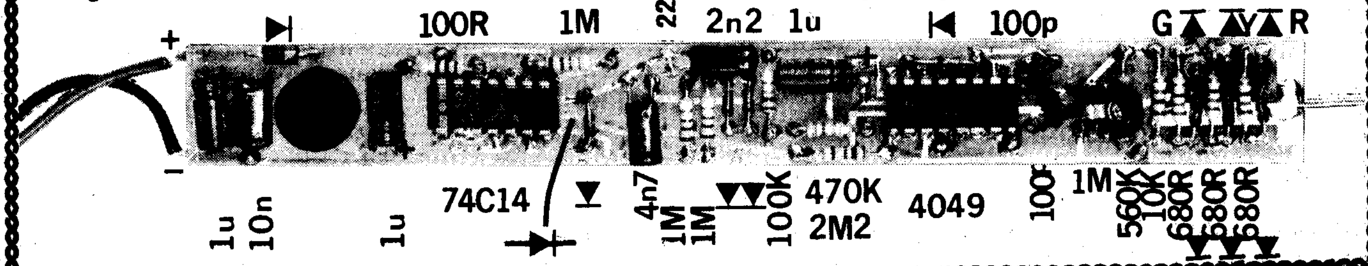
As with all our projects, this probe is the culmination of many hours investigation and diagnosis of the various equivalent products on the market and we have come up with the best design ever.

The probe has both visual and audible indication of HIGH, LOW and PULSE and if you have ever used a probe for trouble-shooting, you will know the advantage of an audible signal.

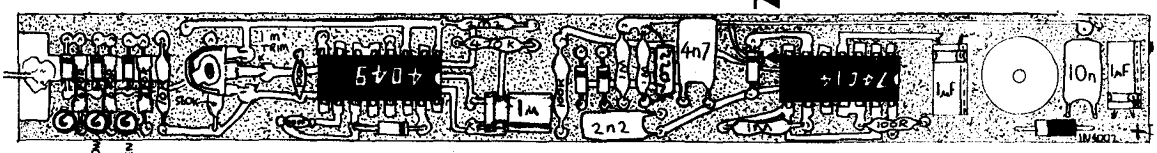
Whenever you are probing around the pins of an IC, it is absolutely essential than you do not take your eyes off the probe, even for an instant.

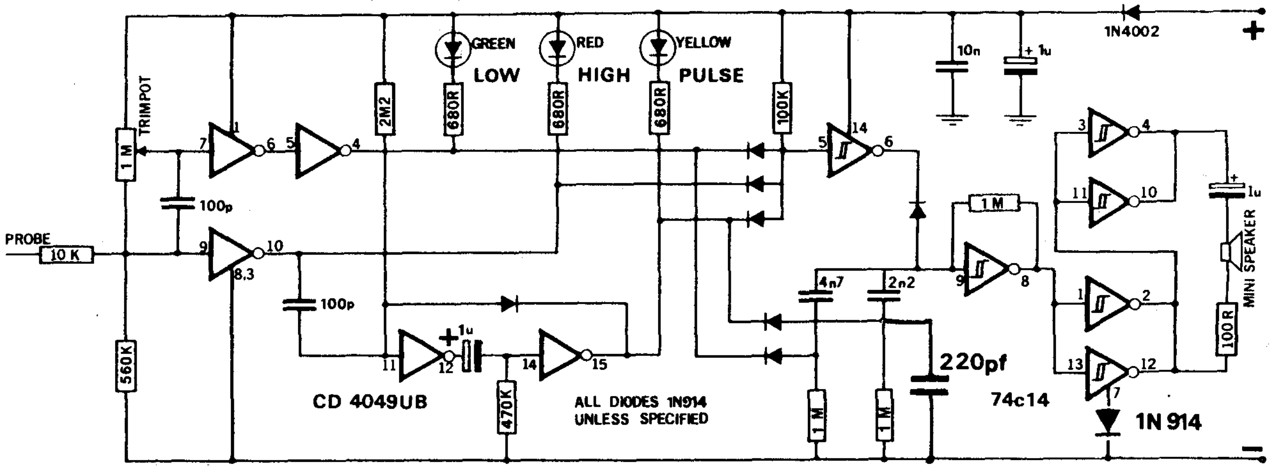
If you attention is diverted to looking at the indicator LEDs, you can allow the probe to slip off the IC pin and cause a short between two pins. This could easily damage the chip.

But with our design, this cannot happen. The audible tone will keep you up to date with the state of each pin and you can concentrate on advancing along each row of outputs.



Cut track and insert 1N 914 diode.





HOW THE CIRCUIT WORKS

The three indicator LEDs are connected to inverters. Each inverter performs a different function and this is how we get the HIGH, LOW PULSE outputs.

The outputs of these three stages must be HIGH when the probe is in the no-signal condition as the LEDs are driven via negative-logic. This simply means the LEDs will illuminate when the output of the driving buffer is low.

The 1M mini trim pot on the input line, sets the voltage level on pin 7 of the top inverter so that the LOW LED is illuminated when the probe detects a LOW.

The section dealing with the LOW signal is the pair of inverters between pins 7 & 6 and 5 & 4. This produces a double inversion so that a LOW on the input will produce a LOW on the output. This LOW illuminates the LED and this is why it is called NEGATIVE LOGIC.

A single inverter between pins 9 & 10 is used to drive the HIGH and once again negative logic causes the LED to be illuminated.

A 100pf capacitor between the inputs of these two circuit blocks acts as a SPEED UP capacitor.

The PULSE-STRETCHER arrangement is a monostable circuit in which pin 11 of the first inverter is normally HIGH, making the output LOW. The 1mfd electrolytic sits in an uncharged condition. The circuit is triggered into operation via the 100pf capacitor on input pin 11. When a LOW is created on pin 10 of the "HIGH" BUFFER, it is passed through the capacitor to pin 11 of the pulse-stretcher circuit. This will cause pin 12 to go HIGH and take pin 14 HIGH with it.

Pin 15 goes LOW and the latching feature is provided by the diode between pins 15 and 11. When pin 15 goes LOW, pin 11 is also pulled LOW and the circuit latches in this state.

The time-delay for the circuit is controlled by the capacitor (and also the 470k resistor). Pin 12 is HIGH and it begins to charge the capacitor at a slow rate due to the 470k resistor in series with the charging voltage.

The voltage on pin 14 gradually decreases until a point is reached when the second inverter will change state. The voltage on pin 14 falls and this will create a HIGH on pin 15. The PULSE LED will be extinguished and the LOW on pin 11 (produced by the diode) will be removed. The first inverter will change state and the charge on the 1mfd electrolytic will be removed fairly rapidly due to the presence of diodes on the input line of the inverter.

The next two blocks on the circuit diagram provide gating for the Schmitt Trigger oscillator.

The actual Schmitt oscillator circuit is made up of the 4n7 capacitor (and/or the 2n2 capacitor), the 1M resistor and the Schmitt Trigger between pins 9 and 8.

But to get this oscillator to produce 3 different frequencies, via a gating arrangement, is not an easy task.

The oscillator requires the earthy end of the capacitors to be connected to ground for the oscillator to keep oscillating. Otherwise it gradually dies away due to the high impedance provided by the gating circuitry.

The three diodes in the centre of the circuit produce a negative logic OR gate and this means it is detecting a LOW on one of the input lines to change the state of the output.

The output of the gate is connected to input pin 5 of a Schmitt Trigger which is acting as an inverter to GATE the Schmitt oscillator.

When pin 5 is HIGH, the output pin 6 is LOW and the oscillator between pins 9 and 8 is prevented from functioning due to the forward-biased diode keeping the voltage below .6v.

When any input of the diode gate goes LOW, the output goes LOW and this causes pin 6 to go HIGH. The oscillator gating diode is now effectively out of circuit and the Schmitt oscillator will produce a frequency which is determined by the value of the capacitors in the earth circuit.

When the LOW LED is illuminated, the 4n7 capacitor will set the frequency. When the PULSE LED is illuminated, the 2n2 will set the frequency. When the HIGH LED is illuminated, the frequency will be set by both capacitors and the 1M resistors in series with them. The isolation diodes on the earthy ends of the capacitors will prevent the HIGH form the inverters affecting the frequency.

With the 4 remaining Schmitt Triggers, we have paralleled them to provide a push-pull arrangement for the mini speaker. Strictly speaking this is not necessary but it is best to utilize all the gates in a package.

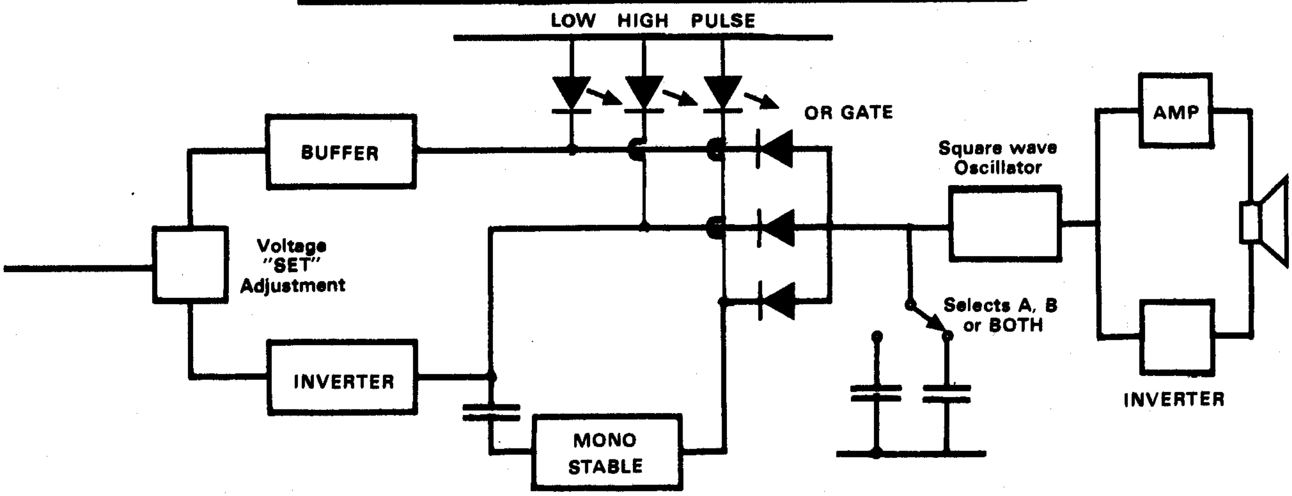
The choice of a 100R resistor and 1mfd electrolytic in line with the speaker prevents overloading of the gates.

A power diode is provided in the positive rail to prevent damaging the probe with reverse voltage if it is connected to the power around the wrong way.

The 1mfd (1u in the schematic) and 10n capacitor across the power rails help remove spikes from the power rails and prevent false triggering.

The 10k input resistor prevents the circuit being tested, from overload or damage.

Finally, one of the inverters of the CD 4049 is unused. Its input line has been tied to earth (pin 3) to prevent the inverter self-oscillating.



LOGIC PROBE BLOCK DIAGRAM

The input block is a 'VOLTAGE SET' control to turn the HIGH/LOW indicator LEDs off when no input voltage is present.

The incoming signal is passed through a non-inverting buffer (two inverting buffers in series) and also one inverting buffer. The output of these gives the HIGH/LOW indication.

A pulse-stretcher circuit provides an indication of a pulse as short as 1 micro-second.

The output of the three buffer circuits is fed to an OR gate and this controls a Schmitt Trigger oscillator.

Three different tones are produced by selecting a combination of the two capacitors. The output of the Schmitt Trigger feeds a push-pull arrangement to drive a mini speaker.

ADDITIONAL NOTES AND IMPROVEMENTS:

We have produced many thousands of the LOGIC PROBE kit and they have been a great success. Only one problem has emerged due to different batches of chips having different characteristics. Some of the kits failed to give an audible sound.

This has been traced to the Schmitt trigger failing to go LOW enough to oscillate.

To overcome this we suggest adding a diode between pin 7 of the 74c14 (HCF 40106 or CD 40014) and earth. The cathode of the diode faces earth as shown in the schematic.

A 220pf (100pf to 330pf) capacitor can also be added across either of the 1M resistors which are in series with the 4n7 and 2n2 capacitors, to improve the output tone.

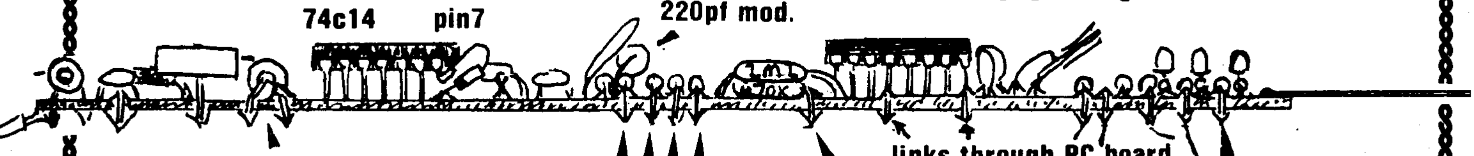
The 1mfd electrolytic in the delay circuit must have its positive end connected to pin 12 of the 4049 for the delay to work properly.

The LOGIC PROBE PC board is a double-sided board and you may be confused when you see it for the first time. All the components are mounted on the top side and this means some lands are drilled and some are not. The board has only about 32 holes and does not need any further drilling.

You can use an IC socket for both chips or solder the pins directly to the board.

Another feature which may be new to some constructors is the need for 6 feed-through links to join some of the bottom tracks to the top tracks. These are short pieces of wire which do not have any components connected to them and require very careful soldering to prevent them from falling off the board when they are being soldered. The PC board layout on P 88 shows the position of the 6 links and they are one of the first items to be soldered.

The probe tip is made from a paper clip.



We have also overcome the second major obstacle - the cost of the case. Apart from using a pen barrel or cigar case, there has been little else on the market to slip the works into. After a lot of high-level searching, I finally came across one of the most obvious products for use as a case. . . It happened this way:

One day I decided to get rid of my shaggy-dog toothbrush and there, neatly displayed on the chemist's counter, was a row of probe cases! Mind you, only a select band of chemists sell our type of probe case. They are called 'AMCAL' Chemists and this is their own 'house' brand of case. Inside the case you get a FREE tooth-brush! I gave every member of the family a free brush and they think I'm the most generous dad who ever lived. Little do they know the real reason.

The cases are good-quality injection-moulded polystyrene halves which slide-fit together to make a sturdy case. One end contains two holes which have been specially included to take the two power leads and the other end can be drilled with a fine drill to take the probe tip.

By spending tens of thousands of dollars, we could not have come up with a more suitable case. In fact it is exactly the right width for the PC board and by mere coincidence, it has the exact right length.

When the probe project is complete, and the two halves of the case taped together, the case is as sturdy and rigid as any $15 case on the electronics market, and you save $14 on the case alone!

PARTS LIST

1 - 100R ¼ watt
3 - 680R
1 - 10k
1 100k
1 - 470k
1 - 560k
3 - 1M
1 - 2M2

2 - 100pf ceramic
1 - 220pf
1 - 2n2 greencap
1 - 4n7
1 - 10n

3 - 1mfd 16v electrolytic

1 - 3mm red LED
1 - 3mm green LED
1 - 3mm orange LED

8 - 1N 4148 signal diodes
1 - 1N 4002 power diode

1 - CD 4049UB Hex Inverter IC
1 - 74c14 Hex Schmitt Trigger IC

1 - 1M mini trim pot

1 - 8R mini speaker

short length of tinned copper wire
1 - 30cm red flex for power lead
1 - 30cm black flex for power lead
1 - red E-Z hook
1 - black E-Z hook

1 - CASE (see text)

1 - LOGIC PROBE PC BOARD

The Probe has been specially designed for trouble-shooting the TEC-1 computer. This computer is described in issues 10, 11 and further issues of TALKING ELECTRONICS.

Unfortunately no Probe boards could be attached to the covers of this book due to the lack of holes for the staples.

The probe PC board is a double-sided board and the majority of the parts are mounted on top of the solder lands. Much to the annoyance of 10 of our readers, we have chosen this arrangement so that the PC wiring remains large and easy to solder to. Jumpers and inter-connecting wires are eliminated. To create the same circuit on a single-sided board would create a little congestion near the front-end of the circuit. And anyhow, we wanted to introduce a double-sided board to our readers.

The PC board manufacturers are hoping for a plate-through-hole board soon, but we assured them that this would not happen for many issues yet.

The only advantage with plate-through-holes is for production runs where jumpers and feed-through pins can be eliminated.

For hobby work, time is not chargeable and plate-through-holes are of no advantage.

BUYING COMPONENTS

Before you start to purchase any of the parts for the probe, remember that all parts must be as small as possible as they have to fit inside the plastic case. To us, all the parts in our prototype are standard as we have been using the smallest and best components for all TE projects, since its inception.

The only new part will be the mini speaker. This is a proper 8ohm voice coil speaker with a metal diaphragm and surrounded by a circular ceramic magnet. It can be used as a miniature speaker for a transistor radio although it does not give the same performance as a normal 57mm speaker. But for our situation, it is ideal. We don't need a loud indication of the logic level and the Schmitt Trigger drives the speaker quite adequately.

The only ODD feature of the layout is the position of pin 1 for each of the chips. They do not correspond with the normal direction of the signal and you will notice the writing on the chips is up-side-down. Take special note of this point and don't go blindly ahead and wire the chips onto the board around the wrong way.

Everything else is straightforward.

Obviously the first item to buy is the case, then you will have an idea of the headroom available for the components.

By now you should have built some of our educational and recreational projects. The Digi-Chaser or Cube Puzzle, for instance, or the Egg Timer, Lotto Selector or Hangman.
If you had difficulty in diagnosing or analysing any of the output stages, you will appreciate the benefit of having a probe. It will inform you of the level of any of the outputs, at a glance.

We have selected CMOS chips for the probe so that it can be used on either TTL circuits (5v operation) or CMOS circuits (5v to 15v).

The input impedance of the probe is about 300k to 400k and this places very little load on the circuit being tested. Thus an oscillator, for instance, will not change frequency appreciably or a delay circuit alter from its specified delay-time.

But the most basic need for a probe is to detect the logic level of any any point in a circuit. This is provided by the HIGH-LOW readout on the two LEDs or via the tone from the speaker.

Along with the HIGH/LOW capability, a very handy feature is to be able to pick up isolated pulses which are of very short duration. Our probe is capable of detecting pulses which are as short as 1 microsecond and this corresponds to a frequency of 1MHz. These pulses are stretched by a PULSE-STRETCHER circuit and appear on the middle LED for an extended period of time so that it is visible. The signal is also passed to the tone oscillator where it is converted to a short beep to let you know a pulse has been detected.

All these features are yet to be found on a commercially available probe at a cost nearing that of our design.

For anyone contemplating building the TEC-1 computer, this probe project is an absolute must. Build it well in advance of the TEC-1 and have it operating correctly for testing the first stage of the computer.

CONSTRUCTION

Lay the components, board and case on the work-bench and get all your tools ready for an hour of quality soldering. You will need snips and long-nosed pliers as well as a fine-tipped soldering iron and a length of fine solder, to achieve a neat result.

A printed circuit board holder will be handy and can be made from two clothes pegs screwed to the bench. The only other item I consider essential is a solder-tray, to accept the dirty solder from the soldering iron,as it must be tapped clean after making each connection.

With all these things around you, an hour will pass very fruitfully as you build ON TOP of the PC board.

The first two components we suggest to be fitted are the two integrated circuits. This is because they are added to the centre of the board. The other components are fitted on either side and access to the solder lands is only available at the commencement of construction.

All the components excepting the mini trim pot will have to be trimmed to fit onto the board as the components should be fairly close to the board to prevent them from bending over and touching other parts.

Because very little heatsinking is available for components with short leads, it is necessary to use long-nosed pliers to act as a heatsink for the LEDs, diodes and resistors, when soldering them to the board.

The electrolytics and capacitors can be held in your fingers when soldering as you will soon know if you are taking too long to tack them in position.

So MUCH has been said about soldering that if you don't know what to do in this case, you should not be attempting assembly.

The only damage to components, I can envisage, will occur when soldering the LEDs. Before soldering them to the board, cut the cathode lead short and try the LED for height before soldering. Do likewise with the other two LEDs. Do not bend the leads apart as this will damage the LED. Tack each one in position so that you can adjust it for height and position.

Basically you can start at one end of the board and add each component as you come to it. The mini trim pot will have to be bent over slightly so that it fits into the case. All the other components will be short enough to enter the case, provided they are kept close to the board.

The electrolytics can be bent over and laid neatly across other components and this will allow their leads to be kept a little longer.

The power leads for the Probe should be as long as practical as the probe derives its power from the project under test. With long leads, the probe can be used to trace the circuit on the top-side of a project as well as under it.

The last two items to be soldered are the alligator clips, as the power leads have to be fitted through two holes at the end of the case.

But before the works can be put away, a small adjustment is needed via the 1M mini trim pot. With the probe connected to a 9v to 12v supply, connect the tip to earth (negative of the battery) and adjust the pot until the LOW LED is just illuminated.

This completes the probe and it is now ready for encasement.

USING THE PROBE

The power for the probe is obtained from the project you are testing. This way the voltage is automatically adjusted to match the project. It also saves putting a battery in the probe case.

Connect the positive lead to any positive rail on the test-project and the black lead to earth.

The probe tip should be long and THIN so that it will get into the tightest of places. You can put an insulating sleeve on it so that only the end is bare. This will prevent shorts when testing a very compact circuit.

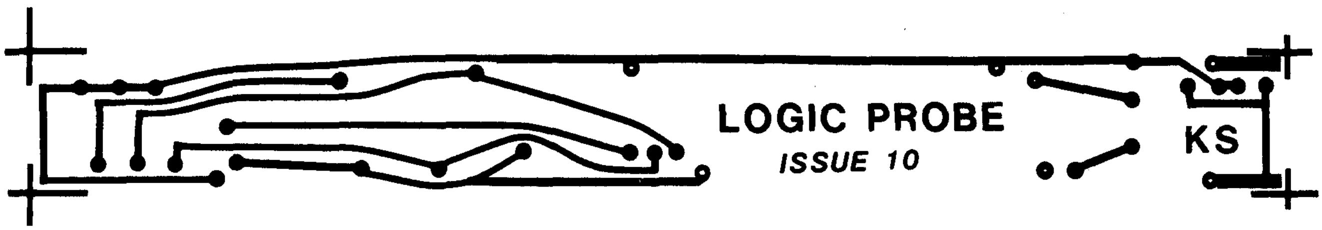
Use the probe project a few times in a KNOWN situation so that you become accustomed to the sound of the high and low signal.

The power leads should be taped together for their full length to prevent them getting tangled.

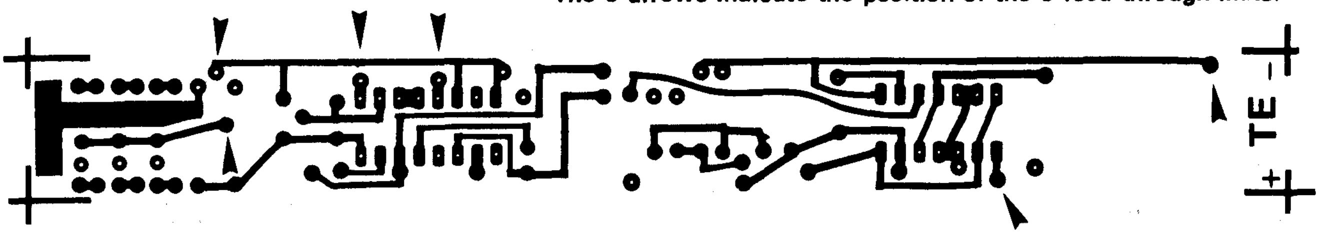
When using the probe to check a project, you will need a circuit diagram. Before touching the probe on any point at all, ask yourself: "Will it be HIGH, LOW or give a PULSE?" The probe will then confirm your decision.

Muster up all the projects you have constructed from Talking Electronics and analyse them with the probe. You will be quite surprised at the various frequencies within within such projects as the LOTTO SELECTOR and MUSIC COLOUR.

Let us know how you like the probe.

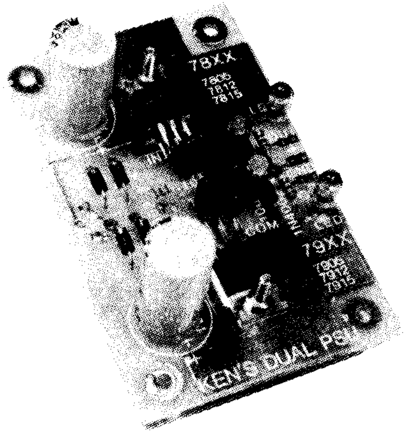


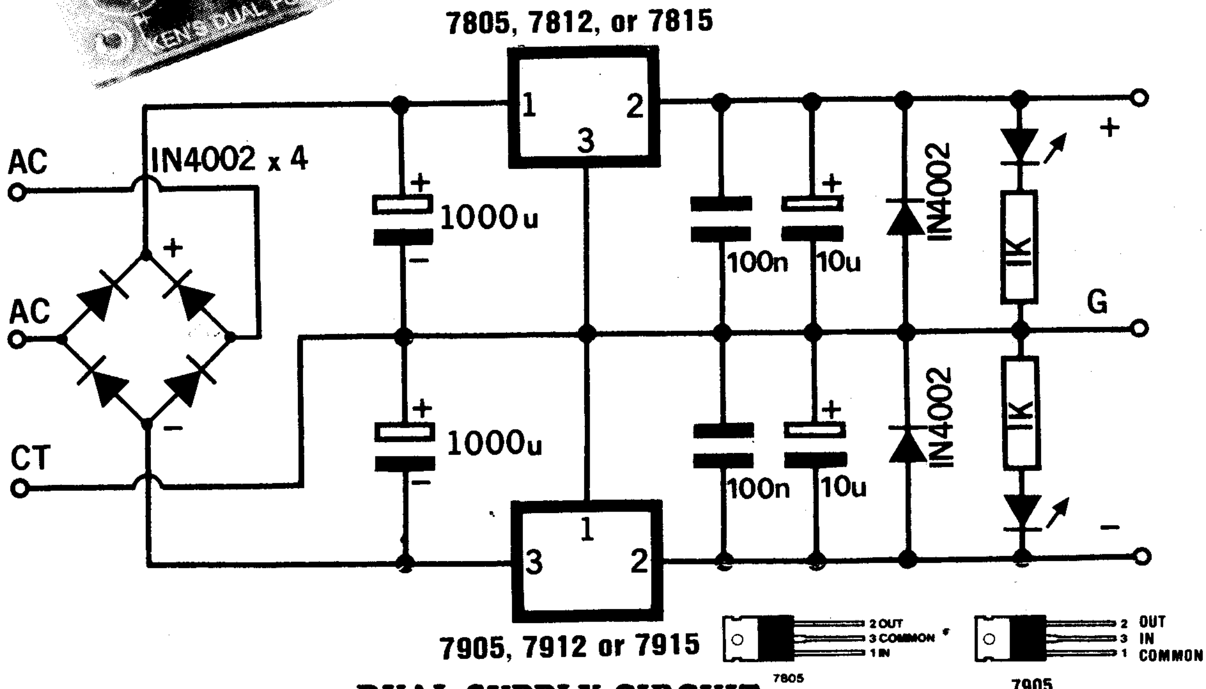
The 6 arrows indicate the position of the 6 feed-through links.



The LOGIC PROBE is built on a double-sided PC board. These are difficult to make in a home-brew situation as the alignment of the two sides must be accurate and the etching must take place at the same time.
You can get around this by making the the lower pattern on a single-sided board and creating the other side with wire jumpers.

KEN'S DUAL POWER SUPPLY





DUAL SUPPLY CIRCUIT

'Necessity is the mother of invention.

That's how this project came to be.

It's a simple 500mA power supply providing a positive and negative rail with an output voltage which is governed by the voltage of the regulator you use.

The PC layout is somewhat simplified by the fact that the board acts as the heatsink. It will dissipate a total of 5 watts for a rise of 75°C. This equates to 2.5 watts per regulator and will give a 600mA output when the voltage-difference between input and output is 4 volts.

If this voltage difference increases, the output current must be reduced to prevent overheating. If you don't understand this, it is discussed later in the article.

The main need for a simple dual power supply was born when Ken began designing synthesiser projects for future issues of TE. These were basically audio projects and they required both positive and negative rails.

As no dual supply is readily available on the market at a low price, he had to assemble one on Matrix board each time he needed a supply. After producing more than 7 of these, he decided it was time to present a PC design for TE. He could then take a PC board from stock and run up a supply in double-quick time.

This project is the result of specific needs and also fills a gap for a simple dual supply using the minimum of components.

PARTS LIST

2 - 1k ¼watt
2 - 100n ceramic or greencap

6 - 1N 4002 diodes

2 - 10mfd 16v electrolytics
2 - 1000mfd 25v electrolytics

1 - 7815 regulator (positive)
1 - 7915 regulator (negative)

2 - 3mm red LEDs

2 sets nuts and bolts
small amount of thermal grease
15cm hook-up wire, 6 colours

1 - DUAL PSU PC BOARD

1 or 2 - 2155 transformers
or 6672 transformer (see text).

It is mainly designed as a +15v and −15v circuit but can be used as a single 5v or 15v supply. It can even be used with different value regulators however this is not recommended as the power lost in the lower-voltage regulator will determine the maximum current.

Because there are a number of options for output voltages, it is necessary to know what you are doing and what parts will be required, before starting.

We will cover 4 possibilities and show which transformer is required. If you use the wrong transformer or the wrong tappings, you will fail to get the full voltage and current.

The two transformers we have selected are the 2155 and 6672.

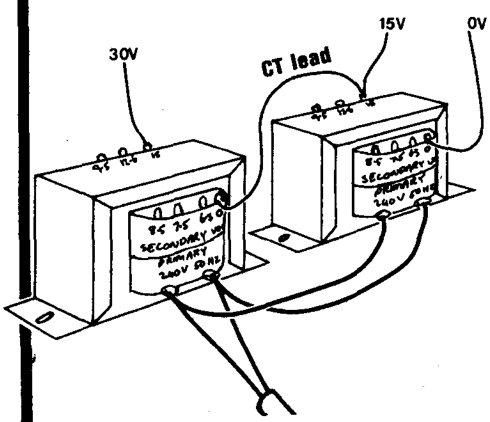
Although they are not ideal designs, the are the cheapest and most readily available.

The 2155 is a 15v types, rated at 1 amp. These are AC values and later in the text you will see how this current rating must be reduced for DC conditions.

The 6672 is a 30v type @1 amp. Again, these are AC values.

In effect, two 2155's are equal to one 6672 transformer.

When using two 2155's, they must be connected as shown in the diagram. This is called PHASING and will produce a transformer equal to a single 30v winding. When phasing is correct, the voltage between the 0v tap on one transformer and the 15v tap on the other will be 30v. If the phase is incorrect, the output voltage will be zero across these terminals.



PHASING TWO 2155 TRANSFORMERS

To produce the correct phasing, the 240v inputs on each transformer are connected in parallel and the secondary terminals are connected in series. We are assuming that the primary and secondary of both transformers are wound in the same direction.

HOW THE CIRCUIT WORKS

The 4 diodes form a full-wave bridge rectifier but the operation of this bridge and the transformer is a little different to normal.

If we take the case of plus and minus 5v using a single 2155 transformer, we can explain the operation as follows:

Take the positive half of the supply. The 7805 sees the transformer as two separate windings and receives a pulse of energy from one of the windings and one of the diodes near the 7805 during the first half of the AC cycle. During the second half of the cycle, the other winding supplies energy to the 7805 via one of the lower diodes.

At the same time, the alternate winding of the transformer is supplying power to the negative regulator via the other two diodes in the bridge.

This means both windings of the transformer are delivering throughout the cycle and two of the diodes are in use at any one time.

+15v −15v MODE

This is the most useful mode for this project. It provides and ideal positive and negative rail voltage for op-amps and other dual-rail chips.

To supply this voltage you will need two 2155's or a 6672 transformer. See the notes on phasing two 2155's to obtain the 0v - 15v - 30v outputs.

SINGLE +15v SUPPLY

In this mode a single 2155 transformer will be needed and one link must be made on the board as follows:
The two lower diodes are connected at their anodes and this point is connected to the CT hole on the board with a jumper wire.

The 0v and 15v tappings of the 2155 are connected to the other 2 input lines. The output is taken from the +ve and ground terminals.

SINGLE 5v MODE

This mode requires a single 2155 transformer plus the link as described in the previous mode. The 0v and 7.5v tapping on the 2155 are used for this mode to prevent the positive regulator overheating.

+5v −12v MODE

Whenever two different voltage regulators are used in this circuit, the lower voltage regulator will have a considerably reduced output current. This is due to the input voltage to the board being high to suit the other regulator. It is not possible to adjust the taps so say 7.5v for the lower voltage regulator and 12.5v for the high regulator. This is because the regulators take it in turns to receive a pulse of energy from each winding.

In this case the 12.5v winding on two 2155's must be used. This gives a very high voltage differential across the 5v regulator and will limit the output current to:

$$= \frac{2.5}{12.5\sqrt{2} - 5}$$

$$= \frac{2.5}{12}\text{Amp}$$

$$= 210\,\text{mA}$$

AN OVERALL LOOK AT THE PC BOARD

This dual power supply project uses a double-sided PC board. This enables the regulators to be bolted directly onto the PC board so that no additional heat-fins are required for currents up to 500mA.

This provides a slight saving in cost and leaves your supply of heatsinks for high current appplications.

The input to the board is designed for a centre-tapped transformer. You may not think a 2155 has a centre tap but this can be created by connecting the 7.5v tap to the CT connection on the PC board. When using a 6672, the centre voltage terminal is the 15v tap. The zero-volts and highest voltage tap are each taken to holes near the 'AC' identification. This gives us 3 input lines as found on a centre-tapped transformer.

The positive regulator goes in the 78xx position and the negative regulator in the 79xx position. They cannot be reversed or swapped over!

Nor will the board work with 2 positive or 2 negative regulators. You will notice the 'IN, COMMON and OUT' terminals are in different positions for the positive and negative regulators. This makes them non-operational in the wrong location.

The 100n capacitor prevents high frequency oscillations from occurring. The 10mfd electrolytic on the output provides a small amount of filtering.

The resistor and LED provide an indication of the size of the output voltage and when the power is applied.

The two diodes across the output are extremely important. They provide short-circuit protection in case the positive lead is connected to the negative terminal. If this were to occur, the current limiting inside the regulators would not detect the short and both would be damaged if the diodes were not present.

The way they operate is very clever.

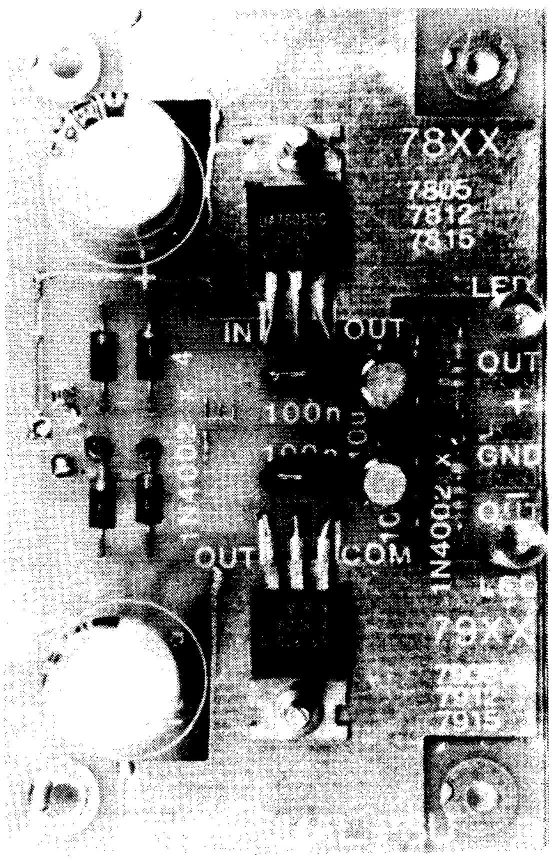
The lower diode becomes forward biased when the positive is connected to the negative and thus it forms a short-circuit to turn on the 7815. The same occurs with the upper diode and the 7915. Thus we save two regulators from being destroyed for the cost of a couple of diodes.

CURRENT RATING

We consider the maximum current is 600mA for each of the regulators. This maximum coincides with two other design factors. They are:
1. The maximum current rating of the transformer and the maximum wattage which can be safely dissipated by the regulator and its heat fin.

We obtained the max. dissipation by loading our prototype until the temp of the regulator reached max. We then reduced the current and re-tested the temp of the regulator. You can generally judge the correct temp of the regulator by feeling it with your fingers. You should be able to touch it for at least 5 seconds. At this temp, the output current was measured as well as the voltage across the input and output terminals. These are the results:
Input voltage: 9v
Output voltage: 5v
Current: 625mA.

This gives (9 - 5) x .625 = 2.5 watts. This is the power lost in the regulator and it must be transferred to the heat fin to prevent the regulator from over heating.



The complete layout with 7808 and 7905 fitted.

The other factor which determines the maximum current available at the output of a power supply is the rating of the transformer. We hear so much about the rating of a 2155 as being 15v at 1 amp. But little do we realize this rating is an AC value.

Working out the DC current rating is quite complex and we will approach it in a simple way.

The power rating of a 2155 is obtained by multiplying the secondary volts by the current. This gives 15v x 1 amp = 15VA. This is 15 volt-amps but for a simple discussion we can call it 15 watts. This is the power rating of the transformer.

Now let's see what happens when we connect the transformer to a full-wave bridge rectifier circuit. The DC voltage which appears across the bridge circuit is $15\sqrt{2}$ volts DC. We readily accept and appreciate the higher voltage which appears across these circuits but fail to take into account the fact that the current rating must be reduced by the same ratio to maintain the value of 15 watts.

This means the output current must be reduced by the ratio:

$$\frac{15}{15\sqrt{2}}$$

This gives us 700mA max for the DC condition. This is a far cry from the 1 amp we so readily accept and expect.

The transformer will deliver higher currents if the load is increased however the output voltage will fall with the result that the regulator may drop out of regulation.

This will means you will have to use a higher voltage tapping and the energy lost in the heatsink will increase dramatically.

When using the 7.5v tapping and drawing 600mA, this project will operate very reliably for long periods of time.

HEATSINKING

We have determined that the maximum power dissipation for each rail of the power supply will be 2.5 watts. When this amount of energy is being dissipated by the regulator and PC heatsink, the regulator is still cool enough to be touched on its plastic case with your finger.

Because the copper fin on the PC board is very thin, it is essential that the regulator be adequately heat-sinked. This involves filing the surface of the board to remove any of the bumps and the use of heat-sinking compound. This compound is absolutely necessary if you wish to run the regulator for longer than 10 minutes at greater than 100mA.

The nut and bolt used for the regulator must be brass or steel to conduct the heat to the other side of the PC board.

If you wish to increase the dissipation, an additional heatfin will be required. The critical part of the fin is the thickness of the material between the regulator and the PC board. This part has the greatest effect on conducting the heat from the regulator to the free air. (Free air is normal air. You can also have fan forced air). A piece of aluminium bent into an L shape will be suitable provided it offers a flat surface to the regulator and PC board. The maximum dissipation with this type of arrangement is 5 - 6 watts.

CONSTRUCTION

The first components to be mounted are the two regulators. File the surface of the PC board to remove any high spots. Place a very small amount of heat conducting grease on the back of the positive regulator and fit its three leads into the holes next to the 78xx identification on the PC board. Use a 1/8" or 6BA nut and bolt to tighten the regulator onto the board. Watch the grease ooze out from around the sides of the regulator. Repeat with the negative regulator.

The board is now ready for mounting the rest of the components. All the parts should be pushed firmly onto the board so that they almost touch it. We still see projects sent in by readers with parts high above the board. And it always looks very messy. In this project there is an exception. The two 1000mfd electrolytics can be kept slightly above the board so that the heat from the fin is not conducted into the electrolytic itself. If they were to get too hot, they would dry out and lose their capacitance.

Solder the lands on the top of the PC board. The last items to connect are three leads to the input (say yellow, white and blue) and three leads to the output (red, black and green).

Mount the board on 4 standoffs and it is now ready to be tested.

TESTING THE UNIT

We will assume you have used the most common voltage regulators giving a +15v and +15v supply, however the same procedure will apply to any arrangement.

Firstly the power supply is tested under NO LOAD conditions. Place a 47ohm or 100ohm ¼ watt resistor in each of the two outside input lines. Make sure the 3 output leads are not shorting together.

Switch on the AC and note the two output indicator LEDs will illuminate. The two protection resistors should also remain cold throughout this test. If they get hot or burn out, a fault will exist. Fix the fault immediately.

Remove the two protection resistors and wire the input directly to the transformer.

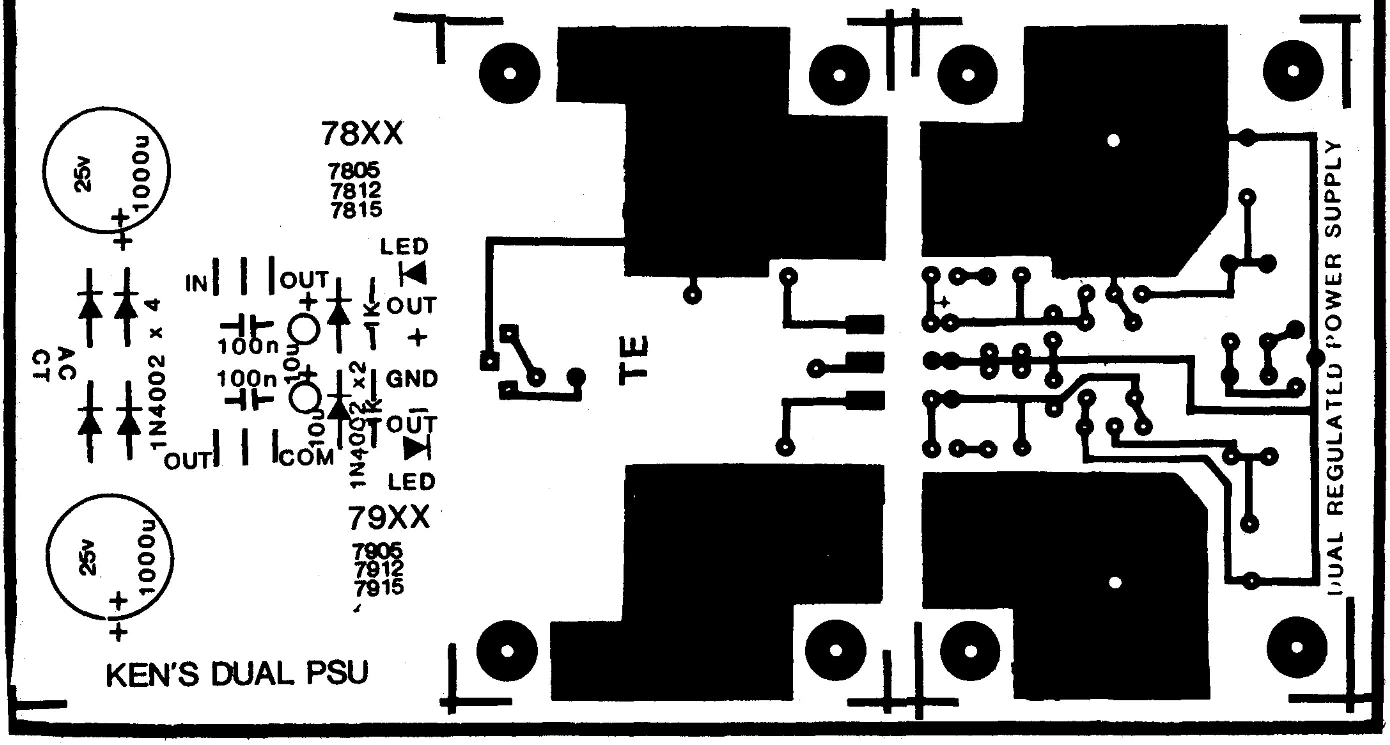
The best way of determining the current capability of the power supply is to connect a specific load to the output. The most accurate arrangement is to use one or more wire wound resistors to dissipate the heat. If you have 8R2 10R or 15R in either 5watt or 10watt sizes, they will be ideal. Globes or motors do not give a very reliable indication of the current flow as they draw a varying current according to the load or brightness. And the current will vary enormously with only a slight change in voltage.

For a 15v regulator, you can use three 8R2 resistors in series for an accurate determination of the 7815 current and a globe on the other regulator for an approximate load.

At 600mA, the regulators can still be touched with a finger while the nut and bolt is a better conductor and will be too hot to touch.

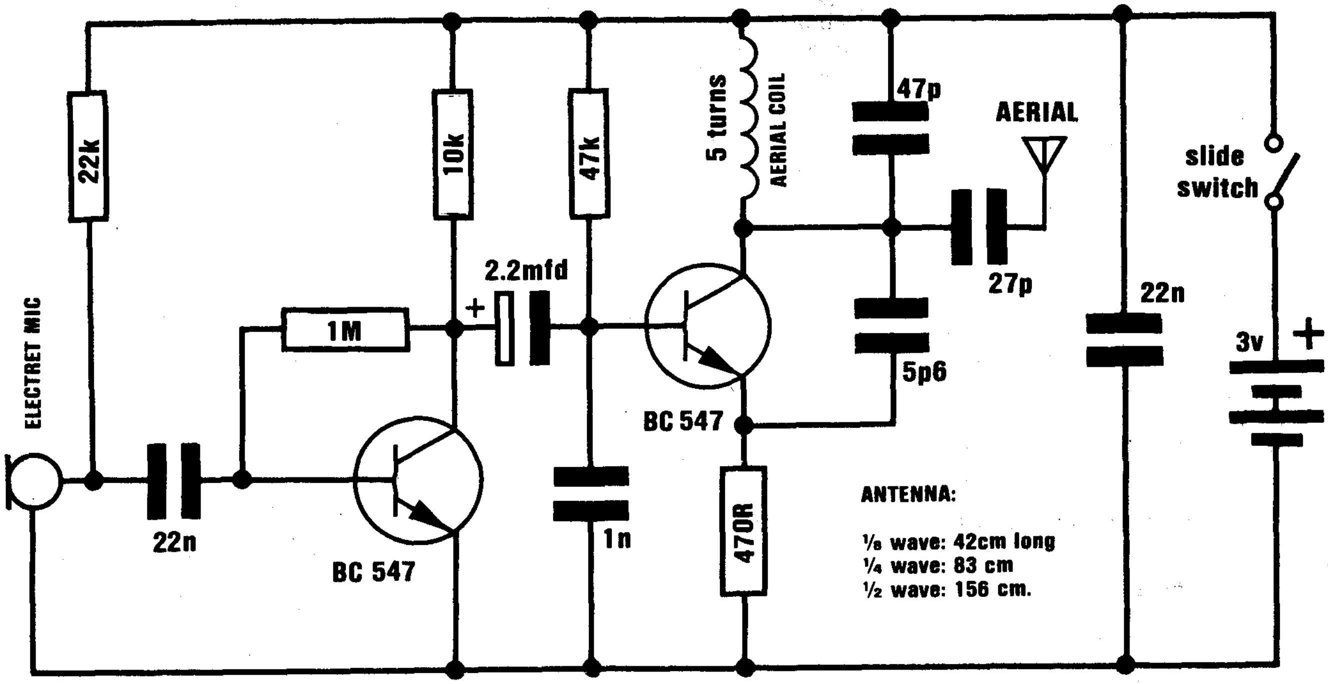
If everything is working correctly, the test load can be removed and the PSU connected to a project.

I hope you find it as handy as we have.



FM BUG

No Bugs with this BUG. A guaranteed performer using readily available parts.



COMPLETE FM BUG CIRCUIT

Corporate espionage is reaching new heights in sophistication. The latest information to be released shows the depths firms will go to pry into a rival firm's operations.

By using the latest in electronic bugging, they have stolen information, secrets and even formulas known only to the inventors themselves.

Take the example of one firm:

Leaks from Top Management level remained a mystery until, one day, a bug was discovered inside the Managing Director's office.
Sitting prominently on his desk was a gift box of imported cigars!

Cleverly concealed in the lower part of the box was a miniature FM transmitter . . all a gift from a phoney sales rep.

This is just one of the many bugging devices available on the eaves-dropping market. The range includes pen and pencil holders, trophies, framed pictures and office furniture with false bottom drawers.

These products are readily sold to fledgling companies, eager to nestle into big brother's market.

And for a while these bugging devices worked. Few firms knew of their existence, and even less on how to sniff them out.

But that has all changed now. If a corporation suspects a leak at any level, the first thing they order is an investigation into security. Not only personnel, but information and electronic security.

Debugging has grown into big business. Most large security organisations have

Everyone has been absolutely amazed at the performance of this bug.

a section concentrating on electronic surveillance including bugging and debugging.

They use scanners to detect hidden devices and can locate absolutely anything, anywhere, and on any frequency.

It was only after the firm above had commissioned a scan of the entire floor, that the cigar box was discovered. Its innocence had deceived everyone. And cost them a small fortune!

Bugging of this kind is completely illegal and we don't subscribe to this type of application at all.

PARTS

1 - 470R ¼watt
1 - 10k
1 - 22k
1 - 47k
1 - 1M

1 - 5.6pf ceramic = 5p6
1 - 22pf ceramic or 27pf or 33pf
1 - 47pf ceramic
1 - 1n ceramic = *1000pf or 102*
2 - 22n ceramic = .022 or 223

1 - 2.2mfd 16v or 25v PC electro

2 - BC 547 transistors (Not SGS type)

1 - mini slide switch spdt.

1 - electret microphone (insert)

2 - AAA cells

10cm tinned copper wire
2 - metres aerial wire

1 - FM BUG PC board

1 - Toothbrush case.

But the uses for our SUPER-SNOOP FM WIRELESS MICROPHONE can be harmless, helpful and a lot of fun.

Our unit is both compact and very sensitive and can be used to pick up even the faintest of conversations or noises and transmit them 20 or so metres to any FM receiver.

When you build the FM BUG you will see why we consider the design to be very clever. We have used only low priced components and they are all easy to obtain.

No air trimmer capacitor is required as the coil is squeezed slightly to obtain the desired frequency. This has allowed us to fit the bug into a tooth-brush case so that it can be carried around or placed on a shelf.

If it is set between two books it will be hidden from view or as a supervision accessory it can be placed on a small child, etc. The transmitted signal will over-ride the background noise and the the output will be clean. If the child wanders beyond the range of the transmitter, the background noise will come up and signal that the tot is out of range.

As an added bonus, you can listen to the chatterings and squabbles as the children amuse themselves in the back yard.

It is also great for picking up the first signs of a child awakening from his afternoon sleep or it can be used as a HELP-WANTED indicator from a bed-ridden patient.

The great advantage of the bug is the absence of wires. And since it draws only about 5 - 10 milliamps, the pair of AAA cells will last for many months.

The success of this FM BUG is the use of TWO transistors in the circuit. To create a good design, like this, each transistor should be required to perform only one task. In any type of transmitter, there is a minimum of two tasks.

One is to amplify the signal from the microphone and the other is to provide a high frequency oscillator.

The amplified microphone signal is injected into the oscillator to modify its frequency and thus produce a FREQUENCY MODULATED oscillator. If an aerial is connected to the output of the oscillator, some of the energy will be radiated into the atmosphere.

To increase the output of our design, an RF amplifier would be needed but this gets into legal technicalities with maximum transmitting power.

It may be of interest to know that a record distance of 310 miles was achieved with a 350 micro-watt transmitter in the USA, some 15 years ago. This equates to an astounding ONE MILLION miles per watt!

In simple terms, an RF amplifier becomes a LINEAR amplifier. This can be seen as per the second transistor in the Polykit design as presented in issue 4.

We have opted for sensitivity and the first transistor is employed as a pre-amplifier. This will enable you to pick up very low-level sounds and transmit them about 20 to 50 metres.

MAKING THE OSCILLATOR COIL

The only critical component in the FM BUG is the oscillator coil. When I say critical, I am referring to its effect on the frequency. Its critical nature only means it must not be touched when the transmitter is in operation as this will detune the circuit completely.

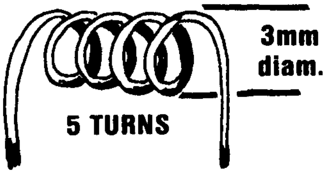
It is the only component which needs to be adjusted or aligned and we will cover its winding and formation in detail.

The oscillator coil is made out of tinned copper wire and does not need any insulation. This is not normal practice but since the coil is small and rigid, the turns are unable to touch each other and short-out.

The coil is made by winding the tinned copper wire over a medium-size phillips screw-driver. The gauge of wire, the diameter of the coil and the spacing between turns is not extremely important and it will be adjusted in the alignment stage. However when the project is fully aligned, it must not be touched at all.

Don't be over-worried at this stage. Just follow the size and shape as shown in the diagram and everything will come out right in the end.

THE DETAILS:



The coil has 5 turns. To be more specific, it has 5 loops of wire at the top and each end terminates at the PC board. The coil must be wound in a clock-wise direction to fit onto the board and if you make a mistake, rewind the coil in the opposite direction.

A pre-wound coil comes with the kits supplied by the magazine however you can make you own very easily in a few minutes.

Collect all the necessary components and lay them on the work bench ready for the next stage:

CONSTRUCTION

Construction is quite straightforward as everything is mounted on the printed circuit board. The only point to watch is the height of some of the components. The capacitors and electrolytic must be folded over so that the board will fit into the case.

Positioning of the parts is not as critical as you think as the final frequency is adjusted by squeezing the coil together or stretching it apart.

However it is important to keep the component leads as short as possible and the soldering neat due to the high frequencies involved. The components must be soldered firmly to the board so that they do not move when the transmitter is being carried.

Even the poorest of soldering will work but who wants to see poor soldering on a project? Especially when it is housed in a clear perspex case.

The soldering may not affect the resulting frequency but poor layout of the components certainly will.

All the resistors must be pressed firmly against the PC board before soldering and the two transistors must be pushed so that they are shorter than the opening in the case.

Some BC 547 transistors will not work in the circuit. Maybe the frequency is too high. SGS BC 547 transitors did not work at all. The other two types: fBC 547 and Philips BC 547 worked perfectly.

All the small-value capacitors are ceramic as they are not critical in value and do not need to be high stability. But you must be careful when identifying them. It would be a very simple mistake to buy a 56pf instead of 5p6 because there is no difference in the size. 22n may be identified with 223 or 22n or .022. A capacitor marked 22k will be a 22pF cap and will not be suitable. The 1n capacitor may be marked 1n or .001 or 102. These are all the same value. The value 101 or 103 is NOT 1n so be careful, the caps may be about the same size. The rule is: don't use a capacitor unless its markings are clear and you are sure of the value.

The switch is mounted on the PC board with its three terminals fitted into the large holes.

Later, a square cut-out will be made in one half of the plastic case so that the slide of the switch protrudes through.

The final items to add to the board are the two AAA cells. These are available at Tandy stores and we have chosen them for slenderness so that they can be fitted side-by-side in the case. A small piece of tape will keep them together ready for connecting into circuit.

It is very difficult to solder to the zinc case but if you roughen the surface with a file and use a large, HOT, soldering iron, the job can be done very quickly. Use a piece of tinned copper wire to join the positive of one to the negative of the other. At the other end, solder longer lengths of wire so that they can be connected directly to the PC board. Make sure the positive terminal connects to the plus on the PC board.

AAA cells are also obtainable at photographic shops. The only alternative is an 'N' cell which is nearly as thin as an AAA cell but only half the length. If all this fails, you can use 2 AA cells in a long tooth-brush tube by connecting them end-to-end.

The terminal marked A on the board is the antenna output. For a frequency of 90MHz, the antenna should be 165cm long. This is classified as a half-wave antenna and provides one of the most effective radiators. If you find the antenna gets in the way you can opt for a quarter-wave antenna and this will be 83cm long. If you only require to transmit 10 to 20 metres the antenna can be as short as 42cm or even as low as 5 or 10 cm.

The most suitable length will depend on the sensitivity of the FM radio used to pick up the signal and the obstructions between the transmitter and receiver. It will be a good experiment for you to 'cut' your own antenna and determine which is the most suitable for your application.

HOW THE CIRCUIT WORKS

The circuit consists of two separate stages. The first is an audio pre-amplifier and the second is a 90MHz oscillator.

The first stage is very simple to explain. It is a self-biasing common-emitter amplifier capable of amplifying minute signals picked up by the electret microphone. It delivers these to the oscillator stage. The amplification of the first stage is about 100 and it only operates at audio frequencies. The 22n capacitor isolates the microphone from the base voltage of the transistor and allows only AC signals to pass through. The transistor is automatically biased via the 1M resistor which is fed from the voltage appearing at the collector. This is a simple yet very effective circuit. The output from the transistor passes through a 2.2mfd electrolytic. This value is not critical as its sole purpose is to couple the two stages.

The 47k, 1n, 470R and 22n components are not critical either. So, what are the critical components in this circuit?

The critical components are the coil and 47pF capacitor. These determine the frequency at which the bug will transmit. In addition, the effective capacitance of the transistor plays a deciding factor in the resulting frequency.

This stage is basically a free-running 90MHz oscillator in which the feedback path is the 5p6 capacitor.

When the circuit is turned on, a pulse of electricity passes through the collector-emitter circuit and this also includes the parallel tuned circuit made up of the oscillator coil and the 47pf capacitor. This pulse of electricity is due to the transistor being turned on via the 47k resistor in the base circuit.

When ever energy is injected into a tuned circuit, the energy is firstly absorbed by the capacitor. The electricity will then flow out to the coil where it is converted to magnetic flux. The magnetic flux will cut the turns of wire in the coil and produce current and voltage which will be passed to the capacitor.

In theory, this current will flow back and forth indefinitely, however in practice, there are a number of losses which will cause the oscillations to die down fairly quickly.

If a feedback circuit is provided for the stage, the natural RESONANT frequency of the coil/capacitor combination will be maintained. The 5p6 provides this feedback path and keeps the transistor oscillating.

The 5p6 feeds a small sample of the voltage appearing at the collector, to the emitter and modifies the emitter voltage. The transistor sees its base-to-emitter voltage altering in harmony with the resonant frequency of the tuned circuit and turns the collector on and off at the same frequency.

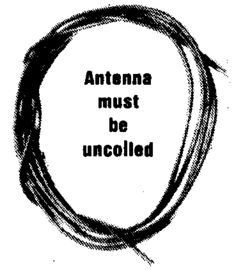
Thus there is a degree of stability in the oscillator frequency.

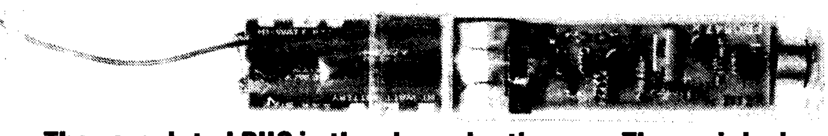
The actual frequency of the stage is dependent upon the total capacitance of the circuit and this includes all the other components to a minor extent.

Once the basic frequency of 90MHz is set, the variations in frequency are produced by the changes in effective capacitance of the transistor. This occurs when its base voltage is increased and reduced. The electret microphone picks up the sound waves which are amplified by the first transistor and the resulting frequency is passed to the base of Q2 via the 2.2mfd electrolytic.

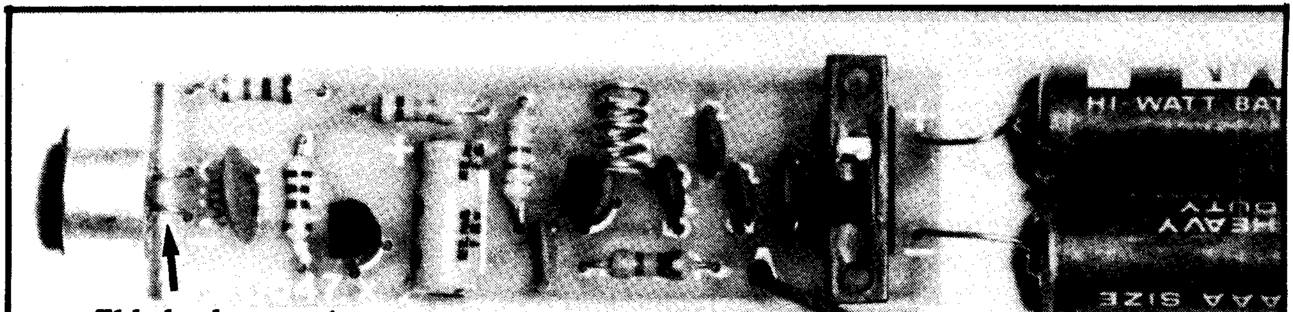
This alters the gain of the transistor and changes its internal capacitance. This junction capacitance modifies the oscillator with a frequency equal to the sound entering the microphone thus FREQUENCY MODULATING the circuit. A short length of antenna wire is connected to the collector of the oscillator via a coupling capacitor and some of the energy of the circuit will be radiated to the surroundings.

Any FM receiver will pick up this energy and decode the audio portion of the signal.





The completed BUG in the clear plastic case. The aerial wire supplied is sufficient for a 165cm half-wave antenna and can be cut to 83cm or shorter, depending on the range you require.



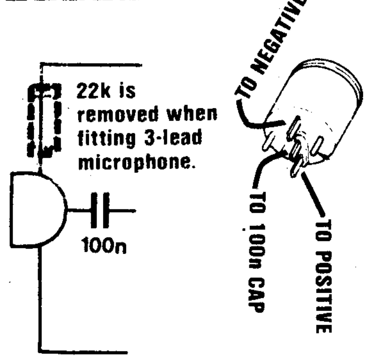
This lead connects to the CASE of the microphone.

3-LEADED ELECTRET MIC:

If you have a 3-leaded electret microphone, it can be used in the circuit provided a simple modification is made. Three-leaded mics have internal dropping resistors and thus the 22k resistor in our circuit is not needed.

All microphones are polarity sensitive and must be inserted into the circuit around the correct way. See the accompanying diagram.

The 2 holes in the PC take the negative pin and the output pin. A jumper wire is taken from the positive pin on the mic to the positive line on the PC, at a point where the 22k meets the positive rail.



SETTING UP THE TRANSMITTER

When the FM BUG is complete, checked and ready for insertion into its case, there is one slight adjustment which must be made to align it to the correct frequency.

As we have said, the only critical component is the oscillator coil. It is the only item which is adjustable.

Since we are working with a very high freqency, the proximity of your hand or even a metal screw-driver will tend to de-tune the oscillator appreciably.

For this reason you must use a plastic aligning stick to make the adjustment. Any piece of plastic will do. A knitting needle, pen barrel or plastic stirring stick can be used.

Place the bug about a metre from the FM radio and switch both units on. Tune the radio to an unused portion of the band and use the alignment stick to push the turns of the coil together. Make sure none of the turns touch each other as this will short out the operation of the oscillator.

All of a sudden you will hear the background noise diminish and you may even get feed back. This amount of adjustment is sufficient. Place the BUG in its case and tape up the two halves.

The fine tuning between radio and transmitter is done on the radio. Peak the reception and move the BUG further away. Peak the fine tune again and move the BUG into another part of the house and see how far it will transmit.

IF THE BUG FAILS

If the bug fails to operate, you have a problem. Simple digital tests will not fix it nor will ordinary audio procedures. The frequency at which the BUG operates is too high.

You have to use a new method called comparison.

This involves the comparing of a unit which works, with the faulty unit.

This means it is ideal for a group of constructors to build a number of units and compare one against the other.

This will not be possible with individual constructors and they will have to adapt this fault-finding section.

The first fact you have to establish is the correct operation of the FM receiver.

If you have another BUG and it is capable of transmitting through the radio you know the radio is tuned to the correct frequency. Otherwise you will have to double-check the tuning of the dial and make sure the radio is switched to the correct setting.

The next stage is to determine if the BUG is functioning AT ALL. The only voltage measurements you can make are across the collector-emitter terminals of the first transistor (1v to 1.5v) and across the collector-emitter terminals of the second transistor (1.3v to 1.5v) These values won't tell you much, except that the battery voltage is reaching the component.

Tune the radio to about 90MHz and lay the radio antenna very close to the antenna of the BUG. Switch the BUG on and off via the slide switch. You should hear a click in the radio if the BUG is on a frequency NEAR 90MHz. Move the turns of the aerial coil together or apart with a plastic stick as you switch the unit ON and OFF.

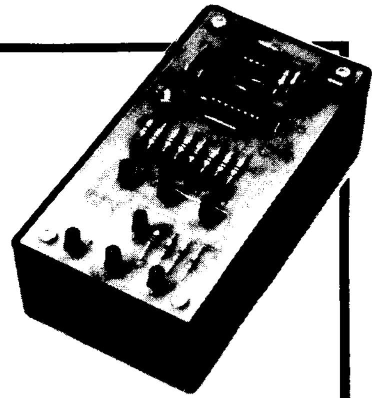
If a click is heard but no feed-back, the oscillator will be operating but not the pre-amp stage. This could be due to the electret microphone being around the wrong way, the transistor around the wrong way, a missing component or an open 2.2mfd electro.

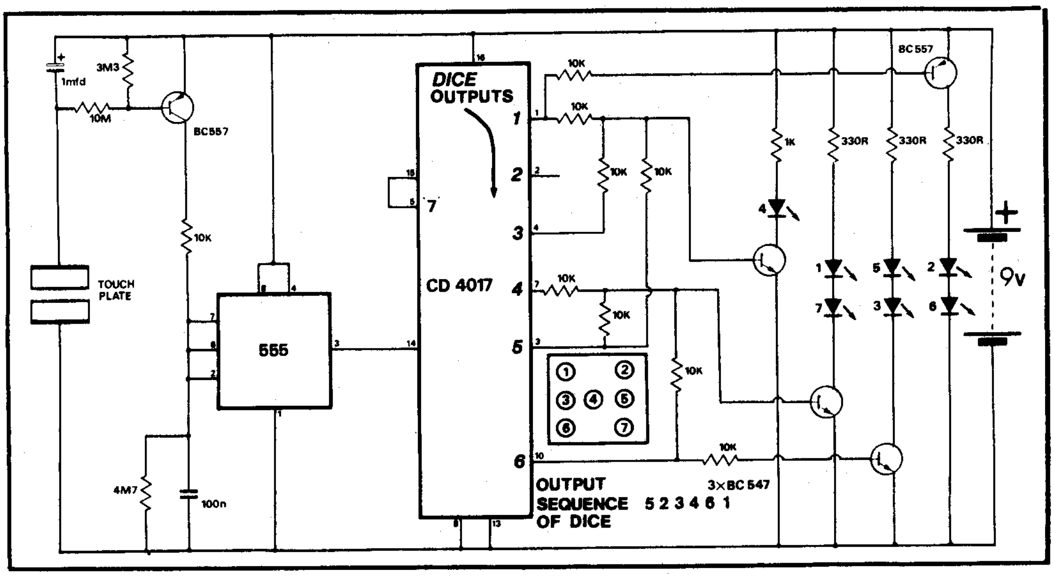
If the fault cannot be located, compare your unit with a friend's. You may have made a solder bridge, connected the batteries around the wrong way, made the coil too big or used the wrong value capacitor for one of the values.

If all this fails, put the unit aside and start again. This time buy a complete kit and see how much more success you have.

LED DICE *WITH SLOW DOWN*

A REALISTIC DICE WITH "TUMBLING" ACTION





This project has two advantages over previous dice circuits. It produces a realistic readout as well as a 'slow-down' feature.

These combine to make it one of our most popular games projects.

As can been seen on the photo above, the 7 LEDs are positioned on the board in the same positions as the spots on a dice (die). When these are activated, they simulate the rolling of a real dice as it rolls across the table.

Consequently we have called the project LED DICE WITH SLOW DOWN.

The whole project fits neatly on top of a medium sized Zippy box with the printed circuit board mounted in place of the aluminium lid. It's another one of our projects which you will be pleased to show around.

A small pre-printed panel can be placed under the LEDs before soldering and this will add to the realism of the project.

Since the LEDs have a low level of illumination, they may be lost in bright sunlight but in a normally lit room, they will be very effective.

All the parts for this project are readily available and the best way to get everything is to buy a kit. The only parts not included in a kit are the Zippy box and 9v battery.

HOW THE CIRCUIT WORKS

The way in which the circuit works is very ingenious. When you touch the TOUCH PLATE wires, the LEDs start flashing in a similar manner to a dice rolling over and over. This gradually slows down to rest and a number is displayed exactly like the spots on a dice.

This "illuminated dice" effect has fascinated me ever since I saw it on an illuminated display near a busy city junction.

It showed a pair of brightly lit dice tumbling over and over and finally coming to rest on a randomly selected number. An apt caption below the sign read: "DON'T GAMBLE: USE SHELL.

Unfortunately the sign was pulled down to make way for road widening and I don't think it has appeared elsewhere.

In those days, whenever I passed the sign, I had a personal bet as to which number would come up next. I don't think I ever won, even though the chances were just 36:1.

PARTS LIST

3 - 330R
1 - 1k
9 - 10k
1 - 3M3
1 - 4M7
1 - 10M

1 - 100n greencap
1 - 1mfd 16v electrolytic

3 - BC 547
2 - BC 557

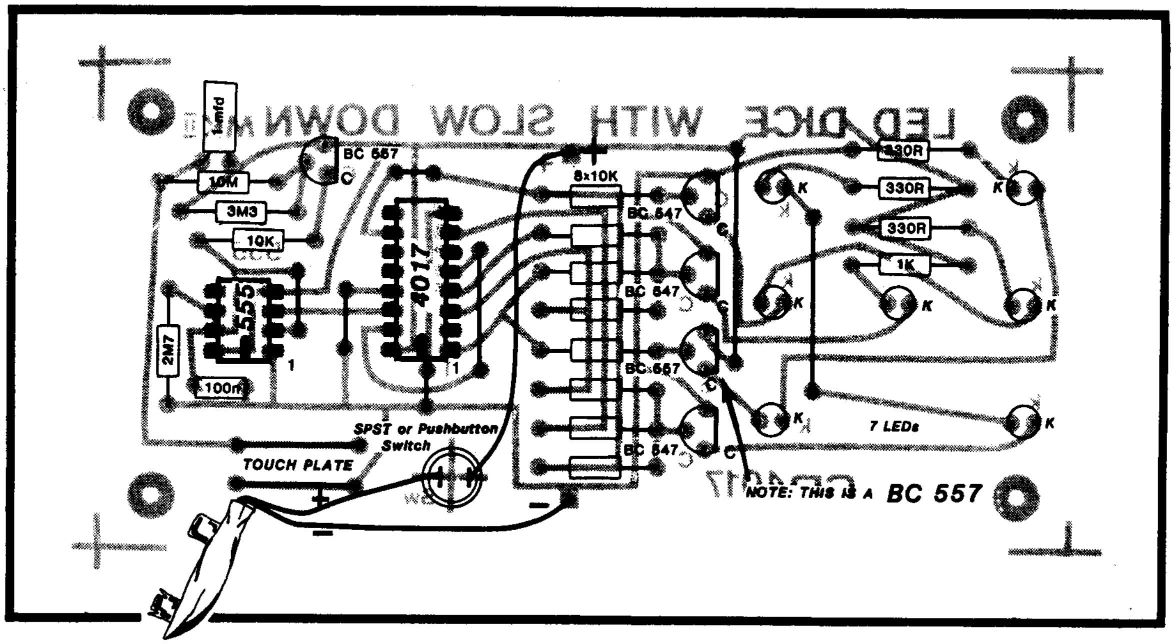
1 - 555 IC
1 - CD 4017 IC

7 - 5mm Red LEDs

1 - switch

1 - battery snap

1 - PC board LED DICE WITH SLOW DOWN



With the passage of 20 years, the mechanics and electronics of the display can be reduced to a couple of IC's and a handful of parts. I hesitate to think how many switches and relays were used in the original sign.

As I mentioned before, the operation of the circuit is quite ingenious. We have programmed the first six outputs of a DECADE COUNTER to light various combinations of LEDs. These LEDs are arranged at one end of the PC board to form the dot layout of a normal dice. One extra LED is placed in the centre to create the pattern for 'one' and 'five'.

Instead of sequencing each output to correspond to one, two, three, four, five atc on a dice, we have jumbled up the sequence to simulate the rolling of a real dice.

The first unusual feature you will notice with the circuit is output pin 2. It seems to go nowhere. Only after examining the make-up of the numbers 1, 2, 3, 4, 5, and 6 will you notice that the LEDs representing the number 2 are also used for 3, 4, 5, and 6. The only time when they are extinguished is for the number 1. We have used this fact to reduce the number of components.

The next important feature involves the buffer transistors. They are necessary to drive the LEDs adequately to obtain maximum brightness. We found it impractical to drive the LEDs directly from the output of the CD 4017, especially when a number of LEDs were to be illuminated. The chip has a maximum output current and this is only sufficient to drive one LED.

The 10k resistors have a dual function. They are used for gating and also as current limiting.

Take pin 1 of the CD 4017 for instance. When it is HIGH, it will turn off the upper BC 557 transistor and also turn on the transistor driving LED 4. Also connected to pin 1 is a network of 10k resistors which will produce a voltage dividing effect on the voltage to the base of Q2. Since a transistor requires only about .6v to turn it on, we have plently of reserve in the arrangement shown.

When output pin 2 goes HIGH, the only LEDs to be illuminated will be 2 and 6. This represents the number 2 on the dice.

The rest of the cicuit is fairly straight-forward. It comprises two building blocks: a 555 oscillator and a transistor slow-down circuit.

MOUNTING THE PARTS

The printed circuit board has been designed to fit exactly on top of a UB 3 Zippy or Jiffy box. The corner holes take self-tapping screws which fit into the moulded pillars.

All that will be housed inside the box is a 9v battery.

Before soldering any components, make sure the board fits neatly over the opening. Trim the sides of the board with a fine file or sand-paper to give it a neat fit. Open out the four corner holes to take self-tappers.

Begin construction by mounting the resistors, capacitors and transistors. Note that a BC 557 fits between the set of BC 547 transistors. You will also need to take care when inserting the LEDs. All the cathodes face one direction except ONE. LED 2 is positioned around the other way. Everything else is as shown on the overlay and on the layout drawing.

All the components on the board are neatly laid out and not cramped. The TOUCH PLATE consists of two parallel wires fitted over the top of the board like two staples. The on/off switch can be either a press switch or a single pole toggle switch. If a push-button is used, it can be used to clock the 4017 to show a random number without having to wait for the slow-down.

The IC's are the last items to be fitted. You can use sockets or mount them directly on the board. Finish assembly with a battery snap and clip it onto a 9v battery.

Drag out your old MONOPOLY game. This LED DICE will add new enthusiasm to playing the game. It may even bring you good luck!

TESTING THE UNIT

Connect the battery and you should see a number of LEDs light up. Place your finger on the TOUCH PLATE and all the LEDs will appear to be lit. They will gradually slow down to a flicker and finally come to rest. This is the correct action for the project but sometimes things don't work out quite so simply.

There are three possible areas for faults to develop.
1. The slow-down section.
2. The oscillator section.
3. The counter and readout section.

Without any test equipment it is only possible to test these three sections by starting at the display end and working back to the slow-down stage.

If the LEDs do not begin to flash when you touch the TOUCH PLATE, you will need to isolate the fault by removing pin 3 of the oscillator from the input of the 4017. This can be done by removing the 555 or cutting the track. Manually clock the IC by tapping a 10k resistor from the positive rail to the input line of the chip. This should make the LEDs change from one state to another.

Next re-connect the 555 and bridge Q1, the slow-down transistor, with a 10k resistor between collector and emitter. This will make the 555 clock the counter fairly quickly. If this does not happen, check the 555 and the 100n capacitor.

To test the slow-down circuit, short the two TOUCH PLATES together with a wire link to produce a very fast-changing display. If this has no effect, check the BC 557 and its surrounding components.

If one or more LEDs do not light up, check the way they are inserted or swap them with another LED which is lighting up.

With this method you will "home-in" on the fault.