PROJECT REPORT

E 519

CORPSE FRIDGE TEMPERATURE DETECTOR

NAME:	MATRIKS NO.
AHMAD RIDHUAN ABDULLAH	08 DEU 00F 517
KAIRULISZWAR ABDUL HALIM	08 DEU 00F 506
MUHAMMAD ARIF ALI ANUAL	08 DEU 00F 017

THIS REPORT PRESENTED TO:

ELECTRICAL ENGINEERING DEPARTMENT

POLYTECHNIC OF SULTAN SALAHUDDIN ABDUL AZIZ SHAH

TO FULFIL THE REQUIREMENT OF

DIPLOMA IN ELECTRONIC ENGINEERING (MEDICAL)

COURSE

JUNE 2003

PROJECT REPORT

E 519

CORPSE FRIDGE TEMPERATURE DETECTOR

NAME:	MATRIXS NO.
AHMAD RIDHUAN ABDULLAH	08 DEU 00F 517
KHAIRULIDZWAN ABDUL HALIM	08 DEU 00F 506
MUHAMMAD ARIF ALI ANUAL	08 DEU 00 017

This report presented to:

ELECTRICAL ENGINEERING DEPARMENT

POLYTECHNIC OF SULTAN SALAHUDDIN ABDUL AZIZ SHAH

To fulfil the requirement of

DIPLOMA IN ELECTRONIC ENGINEERING (MEDICAL)

COURSE

JUNE 2003

CONTENTS

DEDICATION.

Specially dedicated to our parents, family and friends. Not forgetting to all the lecturers of Politeknik Sultan Salahuddin Abdul Aziz Shah.

ACKNOWLEDGEMENT.

First of all we would like to express our deepest gratitude towards Allah S.W.T., God Almighty for without His consent and blessing we've couldn't have found and completed our project.

We would also like to express our deepest gratitude to all our friends for their inspiration, motivation and support towards the completion of our project.

We also would like to thank and gratitude towards our parents and family who also supported us all the way through.

Not forgetting, we thank all the lecturers of Politeknik Sultan Salahuddin Abdul Aziz Shah for all the cooperation given.

Last but not least, we want to thank all that contributed towards our success either directly or indirectly.

Thank you very much.

TESTIMONIAL.

Hereby, we would like to declare that this project was developed and produced based on our own effort and works, expect, some references and appendix that are attached together with this report.

Prepared by,

....

(AHMAD RIDHUAN ABDULLAH)
08 DEU 00F 517

....

(KHAIRULIDZWAN ABDUL HALIM)
08 DEU 00F 506

....

(MUHAMMAD ARIF ALI ANUAL)
08 DEU 00 017

29TH AUGUST 2003

Certified by,

....

(MR. ZUNUWANAS B. MOHAMAD)

....

(MR. ABU BAKAR B. HAFIS)

....

(MISS WEE SOO LEE)

INTRODUCTION.

Every students undergoing Diploma in Biomedical Engineering, semester 5 and 6, must take-up the subject E419/519, project. Generally the purpose of this subject is to expose the students the processes and skills to complete a designated project.

The project that we have to complete is "Corpse Fridge Temperature Detector" . This biomedical equipment is a technique used to detect the temperature inside the corpse fridge.

Although the project seems simple, but the process of completing the project is time-consuming, and needs special attention and care in realizing each process.

Anyhow, by completing this project, we have gained many experience along the way. Which we hope can help us in the real working environment soon.

OBJECTIVES.

This project is are of the subjects that combines theory and practical together. The reasons for making the "Corpse Fridge Temperature Detector" is ; -

1. The students can know the electronic component and understands the circuit operation.
2. To give general knowledge to the students on how and the steps driving projects.
3. Student can learn the proper usage of the machine and hand tools and using them to complete the projects.
4. Student show their hand work in completing their projects.
5. To reveal to the students about the responsibility and work the student will face when going out to work soon.
6. to train the students to be confident and has a self-confident and contribute good ideas in delivering dedicated, dynamic, creative, innovative and experience that can help them during maintenance of electronics equipment. After experiencing failure and problems in the process of completing their projects.
7. The students receive the basic expertise like soldering, etching, circuit drawing, circuit connecting, and soon.
8. To bring out the talents in the students in field of electronics.
9. To make the students interested in electronics.
10. To plant cooperation spirit and nature of helping each other amongst students in getting a quality and perfect efficacious work.

CONCLUSSION

With the grace of God Almighty, Alhamdulillah we have completed our report for the Corpse Fridge Temperature Detector. Throughout the process completing each procedures needed for achieving success, we have gained many constructive and meaningful experience. We do admit that we had doubts on ourselves in completing this report in time.

As we have reached to the end of the process, we leave come to a conclusion about the project. The project Corpse Fridge Temperature Detector is a device used for detecting the temperature level inside corpse fried and produce alarm if the unacceptable temperature has been detected. The Corpse Fridge Temperature Detector operates by generating LCD Thermometer circuit, Fridge Alarm Circuit and Alarm Timer Circuit.

We can conclude that the subject of project really helped us out in bringing out our hidden talents in electronics. We managed to understand and implement the circuit with the help of many topped with the readily available knowledge in ourselves.

We also have learnt a lot an electronics principles and the physiological of the corpse or dead body by completing this report. The knowledge that we gained are hoped to be implement in the working environment in the near future.

As a conclusion, we are determined that this subject, not only broadened the aspect of electronics field alone but also made us expose ourselves in the field of pathology and physiology.

DEVICE PURPOSE.

The project, Corpse Fridge Temperature Detector is a device used for detecting the temperature level inside corpse fried and produce alarm if the unacceptable temperature has been detected. The acceptable temperature for the corpse fridge is around -3°C to -5ºC. When the LCD Termometer detect the lower or higher temperature, automatically the alarm circuit produce sound to make an attention which fridge had a problem. The temperature will be displayed on the LCD together.

So, it will help technician to confirm which fridge has a problem, what is the exactly temperature inside the fridge and the result after repairing or maintenance has benn done.

THEORY & SYNOPSIS

SYNOPSIS

Mortuary Temperature Detector is a new project with a new ideas and invention. As we know, the usual and the old stuff for the detecting the temperature is not complete yet. We discover and we find that the temperature detector must have their own alarm to make sure it is still in a good condition. Therefore, we have built one project that call mortuary temperature detector.

This project show a temperature and the alarm will function if the system had failure. For example, the technicians have to set the temperature first. The suitable temperature for the mortuary is about 2 Celsius. When the chiller and boiler that mortuary usually use for the cooling system had failed and because of the temperature is not suitable, the detector will detect the failure and alarm sound will come out.

There are 4 circuits that we used for the combination of this project. There is Thermometer with LCD display circuit. This circuit are the main circuit because the diode which covered with copper wire are working as the detector. We use box of ice to measure the ice temperature for replace the mortuary freezer. ICL 7136 which ordered from United Kingdom are the main component in this circuit. The IC which is combining with the display is link together to perform a temperature display. Actually there are two types of LCD display that can use for this purpose. There are Reflective and Transflective which had different ways to show the display. The ICL 7136 work as a ADC or analog digital converter. Usually the IC convert an analog signal to a digital output after the whole circuit work to measure the inconsistence temperature.

Another circuit that we use are Alarm circuit. When the display show the wrong value, that means the freezer was failure to do so. The alarm will make a busy sound so that the technician will take any responsible action to solve the problem. Actually when the temperature is wrong it must be something with the cooling system. The input of this project is 9 volt through the whole of this project. To make sure this circuit is running , we use a variable resistor to control the resistance and a piezoelectric buzzer for strength and powerful sound.

We also use two more circuit to verify the true measurement of the temperature. The backup temperature detector that use a relay work also as temperature detector but there is still have to combine with the another circuit. The circuit actually for the heat or in the other word the circuit detects a hot temperature. We convert the circuit and change the relay for measure a cold temperature.

Finally we cover all the circuit and put in the small boxes that we built ourselves. We use Perspex to join the box and use a little bit chloroform to make sure it adhesive. Then we put a small Tupperware below this box and put ice in the box so that the copper wire can measure the ice temperature.

Theory Corpse or Dead Body

The beginning process.

As soon as the heart stops pumping, the blood sediments to lower part of the body. This happens due to the gravitational pull of the earth. The bottom of the dead body begins turn to purplish red and upper part will turn to pale white. All of the body part stops functioning. The lung can't change the Oxygen to Carbon Dioxide process or vice – versa anymore. The colon will cease to function. Skin and muscle begins to bend slack and the body temperature is also decrease. All the physiological process ceases to function.

After two to six hours, the muscle hardens beginning with the eye-lid and followed by the rest of the muscle. After two to four hours, the heart returns to slack. The dead body begins to smell between two to three days after. The smell that comes out from the body are so strong, more offensive then that of the animal carcass.

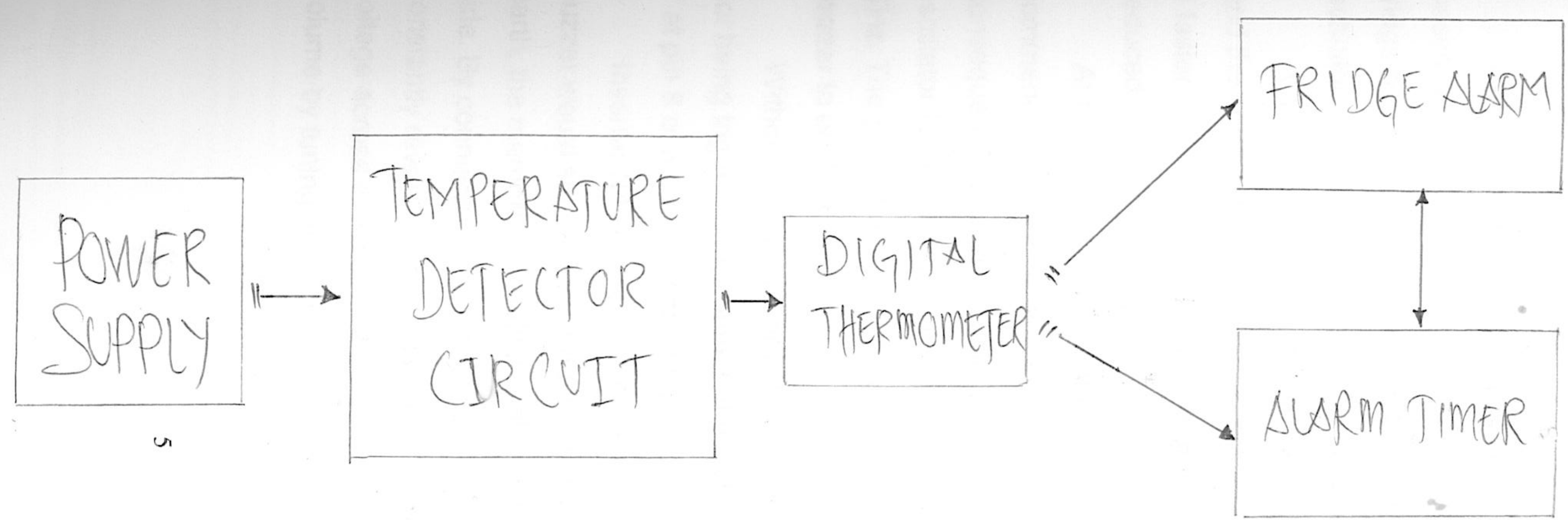
At the same time, decaying process starts from the lower part of the bowel, colon and intestines. The green color begins to show. This is because there are a lot of bacteria in the waste matter inside the colon which also perform the decaying and decomposing on the colon itself.

These bacteria perform a function to help digesting process in a normal living person. This green color will spread to coat throughout the abdomen cavity, chest and thigh. Then it darkens until it is totally black. Stomach swells and ruptures. The eyes will pop out. This is caused by the gas produced by bacteria in the small intestines.

After seven days, the skin will be reddish as though it has caught on fire and it peels easily. After three to four weeks, the nails, hair and teeth will come off one by one. The decaying process happens till the muscle is separated from the bones. Then the decaying continues where the muscle and other organs decomposes in a certain period of time.

This process is also helped by larvae and worms. The soil chemistry factor also influences the decaying rate. The peat soil and the deeper regions of the soil will keep the body conserved. Higher temperature also influences the decaying rate. After two years, only the bones remain but then time factor will eventually erode and crushed the bones within decades.

CIRCUIT THEORY



BLOCK DIAGRAM

Fridge Alarm Circuit

An alarm to tell you that it isn't in a subject of this article. It is based on a light dependent resistor (LDR). As soon as the door of the fridge being guarded is opened, lights fall into LDR, the circuit is then actuated an awarning tone is sounded until the door is closed again.

A delay of about 10 second. Between the opening of the door and the sounding of the alarm is provide by the time constant R3C4. If faster reaction of the circuit is required, the value of R3 may reduced to 220K.

At the moment the threshold of N1 is exceeded, the gate commences to oscillate at a frequency of a few hertz. Each consequent rectangular pulse at the output of a inverter N2 fires oscillator N3 which generated pulse trains whose rates amounts to a KHz. The pulse trains are fed to inverter N4 which causes the piezo buzzer to emit a tone.

Without N2, oscillator N3 would work continuously when N1 is not being triggered, the output of N1 would then be high and the logic 1 at pin 8 of N3 would cause the oscillator to function.

Inverter N4 serves to amplify the output of the buzzer. If the buzzer would simply be connected between the output of N3 and earth, the membrane would merely move from its rest position to 1 side. By connecting the buzzer across an inverter, its polarity is constantly reversed and this caused a doubling of the alternating voltage across it. Preset P2 provides further optimization of the volume by tuning N3 to the resonant frequency of the buzzer.

Preset P1 determines the sensitivity of the alarm, the smaller its value, the less sensitive the circuit is.

The alarm is most conveniently construced on the printed circuit. Current consumption in the quiescent condition is of the order of 0.5mA and when the alarm operates about 4mA.

Part List

Resistor :

R1 = 1K

R2,R3 = 1M (value of R3 may be reduce : see text)

R4 = 10K

R5 = LDR

P1 = preset 10K

P2 = preset 25K

Capasitors :

C1 = 470n

C2 = 10n

C3,C4 = 10u/16V

Semiconductors :

D1 = 1N4148

Ic1 = 4093

Miscellaneous :

S1 = door (micro) switch piezo electric buzzer

PP3 (9V) battery with clips case 100 x 50 x 40 mm

Printed circuit 84437

Alarm Timer Circuit

The circuit begins to operate as soon as its input level become '0'. After about 30 seconds the buzzer sounds four times at 1 seconds intervals. This happen every 30 seconds until the input goes logic high again.

The cicuit is based on 14 stage CMOSbinary counter and oscillator type 4060. The oscillator frequency is determined by f = ¼ R3C1, where f is in Hz, R3 in ohms and C1 in farad.

The oscillator is internally connected to the clock input of the counter. As soon as the reset input is logic low, the counter begins to operate. Becaused at the onset output Q4, Q7 and Q10 are logic '0'. After about 30 seconds, Q10 becomes '1'. The 1 hertz signal on Q4 is then applied to the base of the transistor T1. this transistor therefore conducts in rhythm with the 1 hertz signal and switches the buzzer on and off at the same frequency.

After four seconds output Q7 also becomes logic '1'. As both inputs of NAND gate N3 are now logic high, its output becomes '0'. This level ensures that the reset input of IC2 briefly goes high, so that the counter resets all outputs. If the input of the circuit is still '0', the process stars a new, otherwise the alarm stays quiet.

Power Supply Circuit

The novelty of this design is that it has a variable output from 0V up without using a transformer with 2 secondary windings. The circuit can either be constructed with the well known 723IC for higher output voltages. The L146 which although less popular is still easily available. The choice is left to the constructor. The output current limitation is also variable, but once set is continuously effective. Table 1 shows all the different components values neede to make three different versions (30, 40 and 60V maximum output).

The circuit diagram illustrates the 40V / 0.8A type. The L146IC was used because this can handle the higher output voltages far better than the 723. Normally speaking,2V is the minimum regulated voltage which either IC can provide. Resistor networks R3,R4 and R5,R6 get over this restriction so that the output can be adjusted right down to practically 0V (with P2).

Another aspect of the design is the unusual way in which T3 is driven. When the required output voltage is below the tolerated minimum of the regulator, the potential at pin 4 is below that of pin 5. This result in the IC trying to compensate for this by attempting to increase the output voltage from pin 9. This however doesn't because pin9 is earthed via R7 and D2 which limits the voltage increase.

When the voltage drop across R1 exceed 0.6V, is shorted by R1 and T3 is cut off. During normal operation, the voltage drop across P1 is a constant 1.2V made up of the forward voltage of D1 amnd the UBE of T2.

Table indicates the component values needed to construct 3 different power supplies dependent on the voltage range required. Bear in mind to limit the output current sufficiently to keep the power dissipation of T3 under 40W. the maximum output current of a40V version is 0.8A. It is possible to connect two 2N3055s in parallel (with emitter resistor) to double the output current, but then a 2A transformer is necessary.

Table 1:

U out	I out	R1	R4,R5	R9	Tr1	C1/C5	IC1	T2
0-30V	1.3A	0.47Ω	33K	2K7	24V 2A	40V	723	BD242
0-40V	0.8A	0.82Ω	47K	5K6	33V 1.5A	63v	L14	BD242A
0-60V	0.6A	1.2Ω	68K	10K	48V 1A	80V	L146	BD242B

LCD Thermometer

In this page, I introduced the LCD Thermometer which is sold by Akizuki Denshi Tsusho. At this thermometer, the IC thermo sensor (S8100) or the diode (1S1588) is used as the thermo sensor. When using the IC thermo sensor, the thermometry to +100ºC from -40ºC is possible. Also, when using the diode, the measurement to + 150ºC from -20ºC is possible. Both sensors are contained in the kit. By using the diode as the thermo sensor, more than 100ºC can be measured.

ICL7136 of Intersil (Harris) is used for the thermometer and is measuring the change of the forward direction minute voltages of the diode by the temperature. The 3-1/2 digits liquid crystal display (SP521PR) is used for the display. The most significant digit can display only "1".

The consumption electric power of ICL 7136 is very small and it is possible to operate about three months continuously with the 9-V cell. The main part are contain in the kit. The plastic case and the cell are contained. But, there is not a connection cable of the sensor.

The ICL7136 is the IC for the analog/ digital converter. This is the IC to display the analog voltage in the digital. The IC can be using this time is the DIP of the 40 pins. The voltage to ±200mV or ±2V can be measured in few errors.

Liquid Crystal Display (SP521PR)

This is the TN (Twisted Nematic) – type liquid crystal display. It is possible to 4 digits display but the display of the maximum figure is only "1". It isn't possible to work continuously in the direct current. However, when applying the about 5-VDC voltage between the backplane (common) and the terminal, the color of the segment which correspondence of the terminal and the display segment can be confirmed.

When using the 5-V power supply, the resistor as much as 10K must be put in series to prevent from big electric current's following in case of the short circuit.

The specification of SP521PR to be using this as follows.

- The letter height : 12.7mm
- The maximum applied voltage :10V
- The operating temperature : -10 to 55°C
- The storage temperature : -20 to 60°

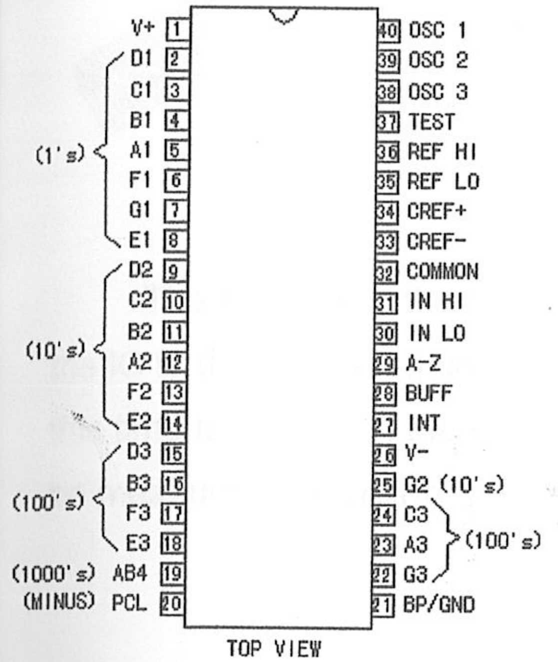
The stick form thermometer is attached on the left side of the agitator tank. The temperature of liquid is monitored with this.

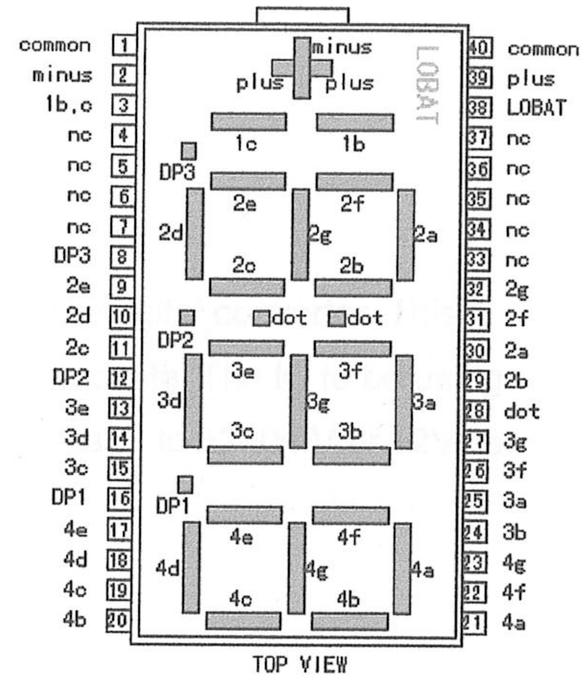
The cover of the agitator tank has small holes for vacating.

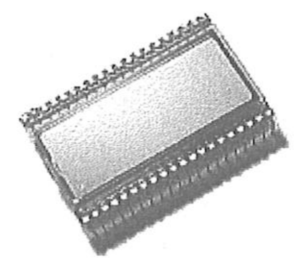
The printed board is suspended from the cover by using the titanium wire.

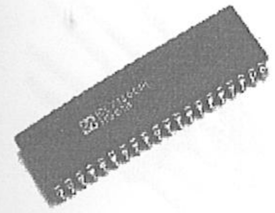
After the temperature of liquid is adjusted, the printed board for etching is suspended from the cover.

Sometimes, you need lift up the board from liquid to confirmed the condition.

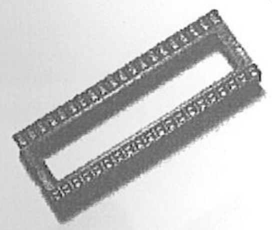

V+ 1
D1 2
C1 3
B1 4
A1 5
F1 6
G1 7
E1 8
(1's)
D2 9
C2 10
B2 11
A2 12
F2 13
E2 14
(10's)
D3 15
B3 16
F3 17
E3 18
(100's)
(1000's) AB4 19
(MINUS) PCL 20
40 OSC 1
39 OSC 2
38 OSC 3
37 TEST
36 REF HI
35 REF LO
34 CREF+
33 CREF-
32 COMMON
31 IN HI
30 IN LO
29 A-Z
28 BUFF
27 INT
26 V-
25 G2 (10's)
24 C3
23 A3
22 G3
(100's)
21 BP/GND
TOP VIEW


common 1
minus 2
1b,c 3
nc 4
nc 5
nc 6
nc 7
DP3 8
2e 9
2d 10
2c 11
DP2 12
3e 13
3d 14
3c 15
DP1 16
4e 17
4d 18
4c 19
4b 20
40 common
39 plus
38 LOBAT
37 nc
36 nc
35 nc
34 nc
33 nc
32 2g
31 2f
30 2a
29 2b
28 dot
27 3g
26 3f
25 3a
24 3b
23 4g
22 4f
21 4a
minus
plus
plus
LOBAT
1c
1b
DP3
2e
2f
2d
2g
2a
2c
2b
dot
dot
DP2
3e
3f
3d
3g
3a
3c
3b
DP1
4e
4f
4d
4g
4a
4c
4b
TOP VIEW



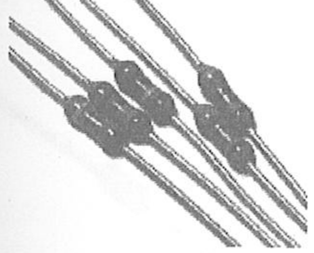


The ICL7136 is the IC for the analog/digital converter. This is the IC to display the analog voltage in the digital.The IC to be using this time is the DIP of the 40 pins.The voltage to ±200mV or ±2V can be measured in few errors.



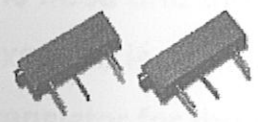
IC socket

This socket is used to install ICL7136 and the liquid crystal display. It is the socket for the DIP of the 40 pins. The socket to use for the liquid crystal display cuts off either side to make the interval of the pin line wide. In case of the socket for ICL7136, the parts are installed inside the socket. So, when there are the supports in the socket, they must be removed.



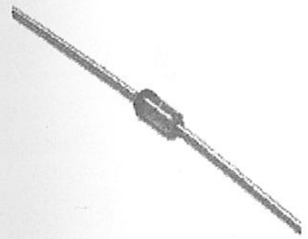
Resistor

As for the resistors which are contained in the kit, the resistors of 1% of the tolerance are used. As the thermometer, the accuracy with the resistance value isn't too necessary. However, because it handles the minute voltage, there is possibility that the resistance value change by the temperature has an influence on the measurement error.



Potentiometer

These potentiometers are used for the zero adjustment and the scale adjustment. Because it changes the resistance value by the pitch of the screw, the resistance value can be changed in detail.



Diode for the temperature sensor (1S1588)

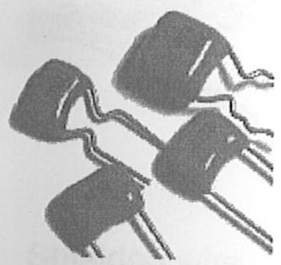
This is the diode to use as the temperature sensor. The original use of this diode is the diode for the switching. The one of the other name can use for the temperature sensor if being the silicon diode. But, the mold type doesn't suit because it is difficult for the temperature change to spread through the joint immediately.

The transistor can be used instead of the diode. It connects the base with the collector and it uses the part of the connection between the base and the emitter for the temperature sensor. The use example is mentioned to the manual of ICL7136. When using the transistor for the temperature sensor, the type which is stored in the metallic case suits.

The lead wires of the diode are insulated with the tubes that the heat can be endured(the glass fiber tube so on). And, the shielded wire is used for the connection with the thermometer.

Multilayer ceramic capacitor

This capacitor is used for the high frequency bypass of the power supply. Because it is for the high frequency bypass, the multilayer ceramic capacitor with the good high frequency characteristic is used.



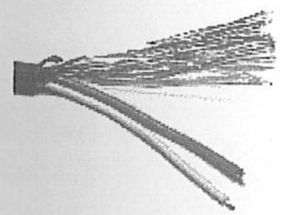
Film capacitor

These capacitors are the capacitors which are used for the input voltage measurement, the reference voltage setting and so on of the ICL7136. The polyester film capacitors are used.

Because it influences the precision of the measurement, it is better to use the capacitor which few changes of the capacity by the temperature. Also, it is better to use the small size one because the place to install is narrow.

Ceramic capacitor

This capacitor is used for the clock generator of ICL7136. To make oscillate at the frequency of about 50 KHz, the ceramic capacitor with the good high frequency characteristic is used.



Shielded wire

This wire is used to connect the diode temperature sensor and the thermometer. The input impedance of ICL7136 is very high and is easily influenced by the noise signal from outside. So, it makes the influence little by the shielded wire
I used the shielded wire of the 2 wires which were at hand, but it is OK in the shielded wire of the single wire.
This part isn't contained in the kit.

ETCHING PROCEDURE

How to Etch a copper PC board

This is a simple protocol for etching a circuit design into a copper coated circuit board. All the necessary chemicals and equipment should be in the Chem room on 5th floor Chamberlin.

Step One: Prepare the design

The first step is to design your circuit and print it out onto transparencies. You should make at least three copies and staple them together so that the design lines up. This is to make sure that during exposure under the UV lamp the light does not expose the design itself and ruin your design. The overall size of the design should be no more than 4"x6" in size, though that limit is approximate. You should check to see that the design will fit in the exposure bracket with the entirety of the design under the glass. It is possible to use larger designs, but one would need an analagous bracket of a larger size. One important point is that it is best to use a PC board that has about 1/4" margins around where your design will go, but no larger. It is better to have a smaller board than a larger one, as it reduces etching time. Another important point is that this is protocol uses a positive photoresist, so the black parts of your design should be where you want the copper to remain and the clear portions are where the copper will be etched away.

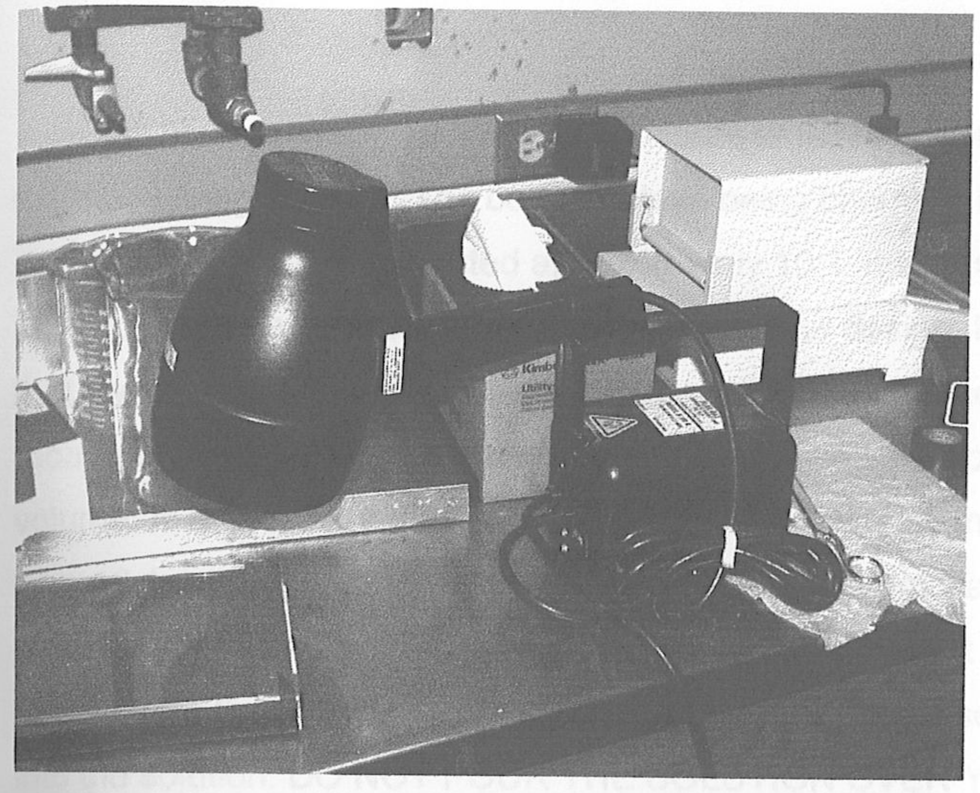
Step Two: Apply the Photoresist

Once down in the chem room, preheat the oven to 150 degrees F and clean the board and the exposure bracket glass thoroughly. I usually scrub off any corrosion and oxidation on the PCB with a brillo pad and water, then wash the board with acetone or ethanol. Be sure that any acetone or ethanol is completely evaporated off of the board before spraying on the photoresist as either of those chemicals will dissolve the photoresist. At this point, you should turn off the lights in the chem room and close the door. Work by the light of incandescent bulbs only, and do not shine theincandescent light directly on the PCB. Leaving the flourescent lights on will expose the photoresist and ruin your pattern. Holding the PCB vertically, spray the PCB with a thin even coating of photoresist by moving across the PCB in horizontal passes and working your way from the bottom of the board to the top. Immediately place the PCB flat on a horizontal surface and let it dry for 10 minutes, and then place it in the oven to force dry it for another 20 minutes (the alternative to this is to leave it dry in a dark place overnight, but you must be sure that it will not get accidently exposed to UV light). It is helpful to place the PCB on some sort of tray to put it in the oven.

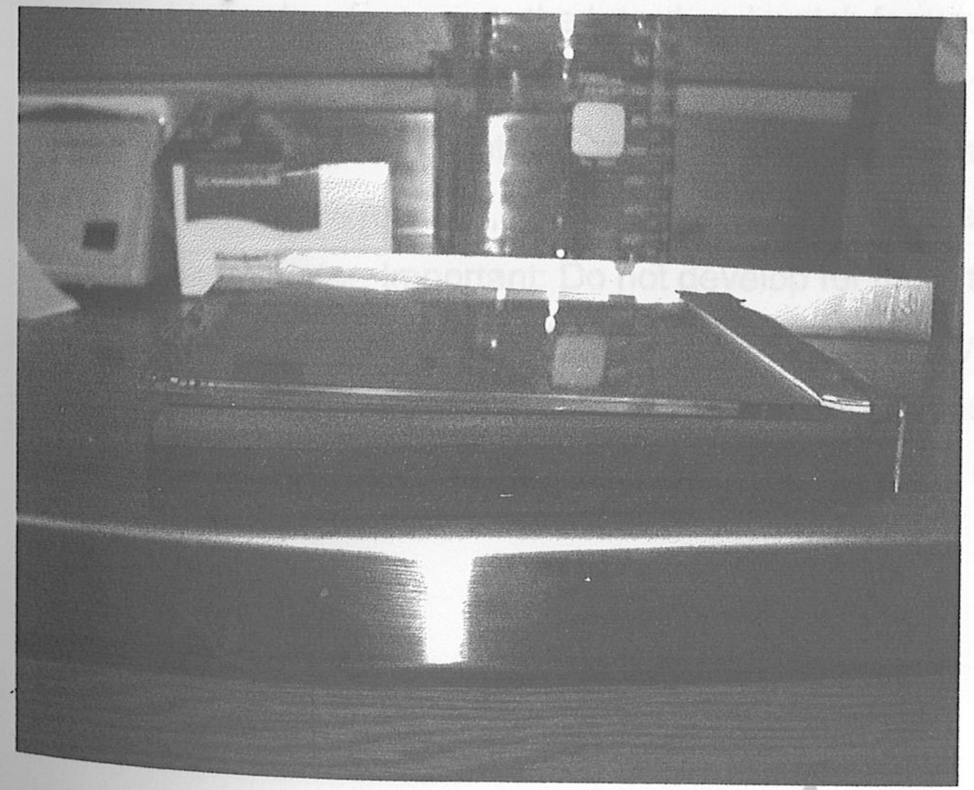
Step Three: Expose Your Design

After the photoresist has been dried, take the PCB out of the oven and set it in the exposure bracket. Set the transparencies with the design on them in the center of the board and close the bracket so that the glass holds the design flat to the board. Place the whole thing under the UV lamp and turn on the lamp. Leave about 5 minutes for the lamp to warm up and then another 10-15 to fully expose the design. If your design is fairly large, you should move the bracket around periodically so that the light falls evenly on all portions of the design. At this stage it is better to spend more time exposing so as to be sure that your whole design will turn out. Both the lamp and the bracket are usually stored on one of the top shelves in the chem room.

The lamp(center) and the bracket(lower right):



Close up on the bracket:



Step Four: Developing

As you reach the end of your exposure time, you should mix up the developing solution. The bottle of developing chemical in the chem room needs to be diluted approximately 10:1 in water. You will need to make enough solution so as to cover your design. I've found that if you use the smaller of the glass casserole trays in the chem room, about 270mL of water and 27mL of developer is a good volume. In any case, it is best make your solution slightly more dilute than 10:1 rather than the other way and using lukewarm water helps the development. Once your solution is made (be sure that it's well mixed) pour the solution into the tray and place your exposed PCB into the solution. DO NOT POUR THE SOLUTION OVER YOUR PC BOARD!!! This will completely erase your design. Now you must slowly swirl the developer over the board and watch for your design to appear. There will not be a color change or anything, but you will be able to see the design faintly on the board. Once you can see the design, or at least its majority, remove the board and immediately rinse it in warm water. Important: Do not develop for more than 1.5-2 minutes. If you cannot see any of the design at this point, you've most likely missed it and the developer erased it. If this is the case, wash the board with acetone or ethanol and start over at step 2. Otherwise, proceed.

Step Five: Etching

At this point you can once again turn on the lights. Place the PC board with a magnetic stirrer into a flask or a tray and put the container on a hot plate. Fill the container with Ferric Chloride so that the board is submerged by approximately 1/4" at least and set the hot plate to 100 deg C. Etching time will vary, though 30-45 minutes is about average.In any case, use latex or vinyl gloves to avoid skin contact with the etchant. Leave the board in the etchant until the clarity of the design is satisfactory, and then remove it from the $FeCl_3$ and rinse it off with water. Both the used $FeCl_3$ and the developer can go down the sink so long as you use lots of water as well.

How to etch a printed circuit board

by Dominic-Luc Webb

Introduction

First, please do not sit around for hours wondering how etching of copper circuit boards is done. It is very, very simple. The basic idea with etching is that you want to 1) transfer your circuit layout to a copper coated fiberglass board and 2) dissolve away all the copper around the transfered layout. Electronics shops will usually have everything you need in stock. I list it all in the materials section. There are a couple ways to accomplish etching. With the diverse background of amateur astronomers in mind, I offer the following cheap and simple protocol that should work well regardless of your technical skills.

Materials

- single or double sided copper circuit board, depending on your circuit
- sodium or ammonium persulphate
- transfer plastic (laser printer overhead transparencies work well)
- black permanent marker
- jar with lid just big enough for the board to lay in.

Printing and transfering the layout

First, you might want to cut a piece of the original PCB to the size you need, since its kind of expensive. This is easily done with a band or jig saw. The board is usually washed to remove fat, or dirt, which can hinder the etching chemical from reaching the copper and reacting with it, although I have never done this and never had a problem. The layout, which was drawn with a graphics program (with nice thick lines, I hope), is printed with a laser printer onto transfer plastic, or alternatively with a copier. It is important to keep track of which way the layout is facing or you could end up with a mirror image of what you wanted. The image on the transfer plastic is then placed onto the copper circuit board. A hot iron is then used to transfer the image to the PCB board. This is possible because the ink from laser printers and copiers comes off the plastic transfer 'paper' and sticks to the copper. The trick here is to not use too high temperature. The 'right' temperature is found by trial and error, but it will always be at or near the 'cotton' setting. If you see melting plastic and smoke, you should deduce that the temperature is too high. If you are not sure the image transfered, try peeling off the the plastic a little bit from a corner, etc. When most or all the image is sticking to the copper, you are at the right temperature.

Once you have the image transfered, it is important to look over the board and make sure you transfered the entire circuit well. Otherwise, you'll end up etching more than you wanted. It is therefore useful to use a permanent marker to go over any problem areas. You're ready for etching once you have an image on the copper that you're content with.

Chemicals

There are a lot of acids and other chemicals that can be used to etch a copper board. Conversely, it may surprise you that some well-known acids like hydrochloric acid, HCl, do not dissolve copper very well, at least, not without addition of an oxidizer like hydrogen peroxide, H_2O_2. I thought I would share some of my experiences about the use of some different chemicals. HCl could be used, as I mention, with presence of an oxidizer. This tends to be rather slow process, and unfortunately, many permanent markers chip off of the board if aggitated in this case. So you will likely end up having to re-draw the whole layout several times (at least) before your PCB will be ready. HCl also produces a lot of toxic outgassing. Acetic acid, CH_3CO_2H, also dissolves copper, but unfortunately is also a very good organic solvent, dissolving ink from virtually any source. Hot sulphuric acid, H_2SO_4, might seem like a good solution, and indeed, it will dissolve copper, but most inks are not resistent to this acid.

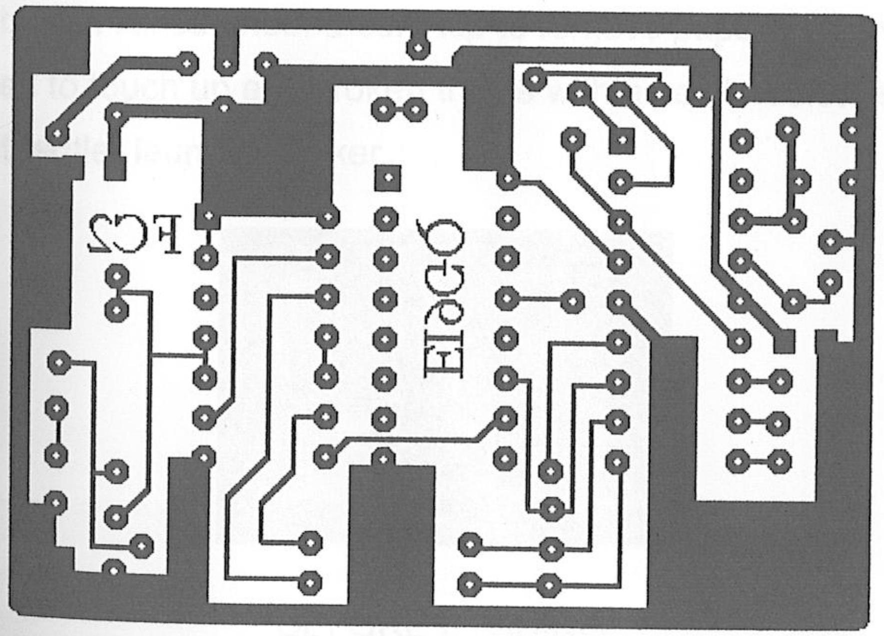
H_2SO_4 reacts with copper to give off some terribly toxic sulphur and copper oxide (S_xO_y and Cu_xO_y) gases that you really do not want to inhale, and won't do much for your social life either. Nitric acid, $HN0_3$, could also be used, but I am told it produces some dangerous outgassing, and it's a really nasty acid that you really would rather avoid. Ferric (FeIII) chloride, $FeCl_3$, is comonly employed in the USA, but in much of Europe its use is illegal for environmental reasons. Thus far, the safest, most effective way I have found to etch a board is with one of the persulphate ($S_2O_8^{2-}$) salts, like sodium or ammonium persulphate. These do not generate outgassing, do not smell bad, and are not so objectionable to environmentalists. Further, persulphate solutions will not dissolve any permanent ink (that I know of). Unlike most of the other chemicals I mentioned, the persulphates are legal and readily available virtually everywhere in the world. I therefore choose sodium persulphate, which is often more available and about half the price of ammonium persulphate.

Etching

Make up the persulphate in water. About 1 teaspoon/25 ml of water is usually adequate. Heat this solution to near boiling in a glass container slightly larger than the board to be etched. Drop in the board, close the lid and submerge the bottle slightly in hot water to keep it warm. The reaction is faster at warmer temperatures. Something like 50-80 degrees C will work. This normally takes me around 30 minutes with occasional mild aggitation. Afterwards, the board can be rinsed, dried and drilled. For drilling, I have had great luck with 1 mm drill bits, which are pretty common. There are smaller, more expensive ones actually made for electronics, but I have never needed them for any of my projects.

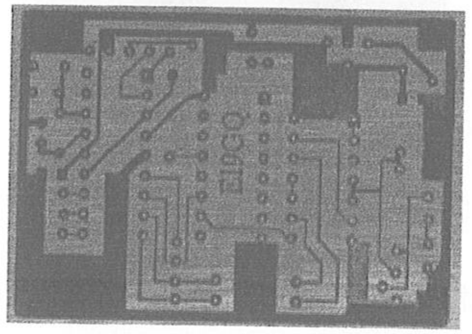
To make smart looking PCB's, all you need is: A computer, a laser printer, copper clad board, etchant, a clothes iron and some Epson glossy photo paper. You can buy special film for making PCB's, but I have found that the Epson paper gives better results. I use Epson photo quality glossy paper for inkjet printers.

Draw the PCB track layout, using a CAD program or a standard drawing program. Remember to reverse the image before printing. Most drawing programs have a 'flip horizontal' function. Print the image on normal A4 paper to make sure that it is the correct size. Check the layout carefully.



EXAMPLE OF PCB LAYOUT

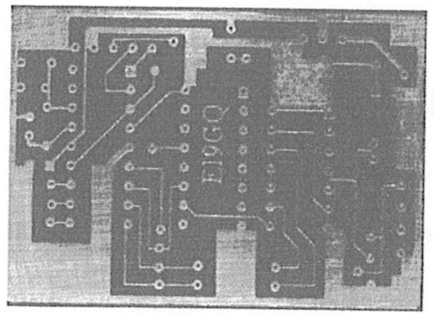
Use a laser printer to print the image on the glossy side of the photo paper. Clean the copper clad board with steel wool or very fine wet sandpaper. Dry the board thoroughly. Make sure that the board is clean and free from fingerprints. Place the photo paper face down on the copper clad board. Use masking tape to hold the paper in position. Don't use vinyl tape. Place the board on a flat surface. You will be using a very hot iron, so don't use the dining room table. I use the back of an old telephone directory. Use a hot clothes iron to transfer the track pattern from the paper to the copper board. Don't be afraid to use lots of heat and pressure. Allow the board to cool. Don't be tempted to lift the paper. Put the board in a container full of warm soapy water. After about twenty minutes the paper will begin to dissolve and disintegrate. Carefully remove the paper from the copper board. Rinse under a cold tap to remove paper residue. You may need to touch up any broken tracks with an etch resist pen. I use a fine Staedtler laundry marker.



BEFORE ETCHING

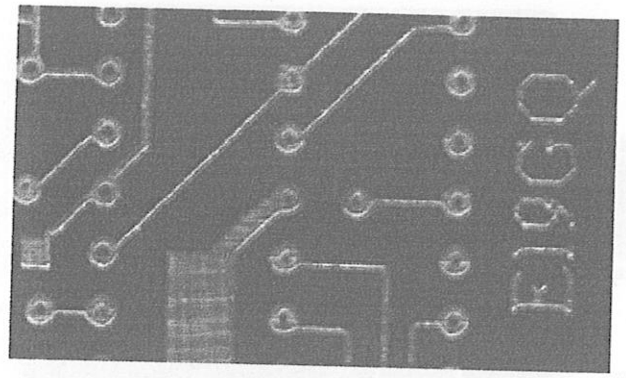
Etch the board in a Ferric Chloride etching solution. You can buy the etchant in liquid form or as anhydrous Ferric Chloride powder. Follow the instructions. NEVER add water to dry Ferric Chloride. Don't get any on your clothes.

AFTER ETCHING



PCB

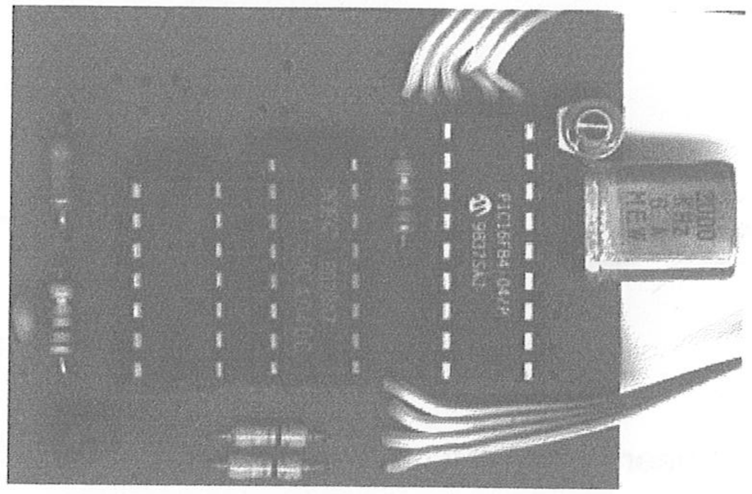
After etching, rinse the board under a cold tap. Remove the etch resist with some steel wool. Dry the board. Use a 0.8 or 1.0 mm drill to make the holes for component leads.



CLOSE UP

The close up picture shows that the tracks are not as clear and well defined as they would be if the board was produced by photographic methods.

The procedure for making double sided boards is a bit tedious. Coat one side of the board with aerosol paint or clear lacquer. Etch the other side of the board as for a single sided board. Remove the paint or lacquer. Drill the component lead holes. Paint the etched side of the board. Then etch the unetched side of the board as for a single sided board. It is difficult to line up the two sides correctly. Use the component holes as a guide.



END RESULT

I have used this method to make PCB's for both DIL and surface mount IC's with 0.05 Inch pin spacing.

Making of a Printed Circuit Board (PCB)

Exposure → Developing → Etching → Stripping → Drilling → Tin-Plating → Soldering

PRINTED CIRCUIT BOARDS - Background

Printed circuit boards (PCB's) are laminates. This means that they are made from two or more sheets of material stuck together; often copper and fibreglass. Unwanted areas of the copper are etched away to form conductive lands or tracks which replace the wires carrying the electric currents in other forms of construction. Some parts of the side with copper tracks is coated with solder resist (usually green in colour) to prevent solder sticking to those areas where it is not required. This avoids unwanted solder bridges between tracks. Sometimes the boards are double-sided with copper tracks on both sides. Tracks on one side can be joined to tracks on the other by means of wire links. Plated through holes are available which do the same thing but these make the PCB more expensive. Components are stuffed into the board by hand or by pick and place machines. Soldering is done by hand or by flow wave soldering where the PCB passes over a wave of molten solder. Most recent PCB's use surface mount techniques where components are on the same side of the board as the tracks. Components are stuck to the board with adhesive and the solder caused to flow by heating the board in a hot gas or by some other technique.

When fitting components ensure that they are orientated correctly and lay flat on the board unless otherwise stated. When the board is assembled avoid flexing it which may crack tracks. Avoid touching the board which may cause contamination due to dirty fingers or damage due to static electricity carried on your body. It is best to handle PCB's by holding them by two edges only, between thumb and forefinger.

Making a PCB

The design layout of the PCB is done on the computer using CAD; the program is EasyPC or EPCPROX.

The layout is printed out on a transparent A4 size sheet called acetate, which is especially used for the purpose. This is done in the same way as printing out a word document. Care must be taken to ensure the circuit layout will be to scale and won't be too big to fit on the sheet. The layer to be printed out must be defined and pad holes must be set to 'Avoid' so as black dots and not rings are printed to indicate holes to be drilled.

The base material is FR4 epoxy all woven glass laminate, thickness 1.6mm with copper foil cladding 1 oz per sq. ft.

The surface resistance is 100,000 Megohms

Photo-resist is positive working sensitive to ultra violet light with a developed image of blue/green tint.

Supplier : Farnell Electronic Components LTd , Cat Ref : 149-063

The copper-clad laminate board consists of a layer of copper, covered over by a layer of green resin called photo-resist. The protective black plastic tape, that protects the copper laminate from scratches, is removed to reveal green positive photo-resist covering the copper.

The printout mask of the image (on acetate) is put over the photo-resist face down, so a mirror image of the circuit layout can be seen over the photo-resist side of the laminated board. On single sided boards this is important because the PCB is designed from looking down from the component side, but the tracks are on the opposite side of the laminated board on the copper side, therefore a mirror image of the PCB layout must be seen. With double sided PCB manufacture, the board is put between two sheets of acetate.

One sheet having the design for the top layer and the other sheet having the design for the bottom layer.

Both sheets are taped together or stapled at the edges ensuring alignment of the printouts. Double-sided copper coated laminate is put between the sheets of acetate so that the two images are lined up on the laminate. With single a sided board, the acetate is placed over the photo-resist side. In each case the laminate and acetate are enclosed under ultraviolet light and agitated for 2 to 8 minutes.

Developing

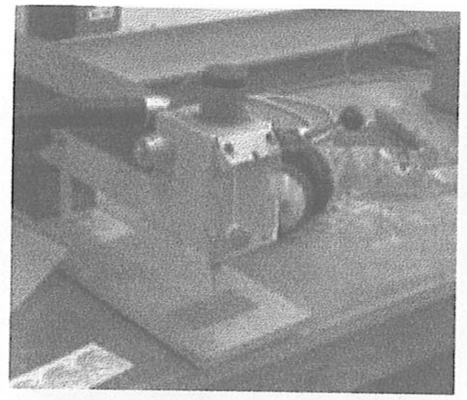
A solution of **Liquid photo-resist Developer concentrate** is mixed in a beaker with 1 part developer to 9 parts water, total 500mls and poured into a basin. A beeper will sound when the 2 minutes are up, the board is taken out of the UV enclosure, and (the acetate is not required any more). The green photo-resist that was exposed will appear a lighter colour and the darker imprint of the PCB can be seen when examined closely. The board is put into the solution and the liquid is flowed over and back on the board. The lighter photo-resist will flow away showing copper and the PCB layout will be revealed. It will be necessary to wash the board under tap water and clean with tissue paper to ensure no traces of photo-resist remain on the copper, otherwise etching would be difficult.

A PCB marker pen can be used to correct any errors such as breaks in the track at this stage

Stripping (optional)

After etching the positive resist 9 (photo-resist) maybe left on the copper to act as protection. Solder is readily achieved through the resist. This green photo-resist can be removed using a tube of photo resist stripper, (like shoe polish) and the PCB washed clean under tap water and dried using tissue paper.

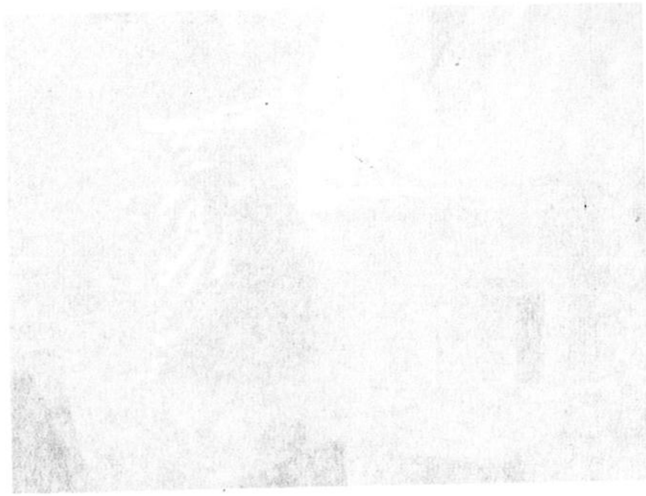
Drilling

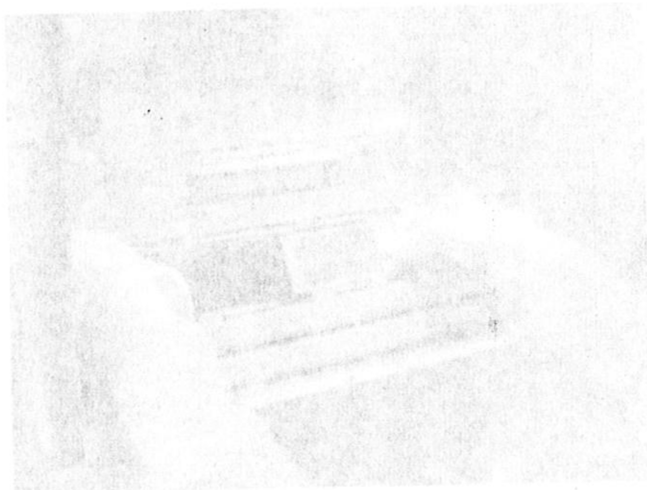


After cutting the PCB to size around the perimeter using the guillotine, drilling using a 0.9-mm drill can now be performed in the workshop.

The board is now ready to stuff with components

Etching





Following inspection to satisfaction the board is etched in etchant as follows:

The etching tank is about half the size of the household water tank in the attic. The tank consists of two compartments. One compartment is 2/3 the size of the other compartment. The larger compartment consisted of a thermostatically controlled heater element. Covering the heater is a protective grill mesh. This grill mesh filters the waste copper away from the heater element .The tank would be filled to the level of the filter with Ferric Chloride Hexahydrate solution, about 5 litres.

The solution is made up of etchent granules dissolved in water. An electric motor spins a shaft enclosed in a tubular barrel 18 inches long containing holes perforated on the circumference from top to bottom. The motor and tube are vertical from the top of the tank so as when switched on, the motor spins the tube and stirs the solution. The effect is to suction of the solution into the barrel and to spray the board being etched.

The solution is heated to 50 degree Celsius before use; a light on the control panel will extinguish when the temperature is reached.

All of our paramedical examinations require the submission of a blood sample and a urine specimen. This information, in addition to all other underwriting requirements, is necessary for the determination of your premium rate class. In addition to the laboratory tests the examiner will also collect your physical measurements. Specific instructions will be provided to you before the exam. In general, these guidelines should be followed:

- For best results, fast at least eight hours before your scheduled examination appointment. Consult with your physician if you have a medical condition (diabetes, hypoglycemia, etc.) that may be adversely affected by fasting. If medically necessary, a non-fasting sample can be obtained. A fasting sample may be easiest for you to provide if the fasting period coincides with your regular sleep time. For your convenience, try to schedule the examination soon after you normally awake. Be sure to take all necessary prescriptions at your regularly scheduled time. Please inform the examiner of all such medications, including over the counter preparations, vitamins and herbs.

Drink only water prior to your appointment (if medically acceptable). Avoid coffee, tea and other caffeinated drinks because of caffeine's effect on blood pressure and heart rate. Avoid fruit juices because they contain natural sugars that can raise your blood sugar level.

Zurich Life Insurance Company will have your blood and urine analyzed by an approved laboratory. The results will be provided to us in a confidential manner for underwriting review. You will be given a copy of our Privacy Promise, Notice of Information Practices, Medical Information Bureau Notice, Federal Fair Credit Reporting Act Notice and HIV Consent Form (where applicable) in addition to any other required disclosure notification. These documents detail your rights and our privacy policies and practices related to the handling of your health and financial information.

When assembling an electronic circuit, a board is needed on which the components can be mounted and wired together.

Mainly, I use the universal prined wiring board (PWB) for assembling the circuit. But I will explain about instruments that makes the printed board.

When you make the high frequency circuit, you need to consider the wiring length and the route of wiring etc.

Therefore when you make the high frequency circuit (radio, high speed CPU etc.), you need to make the printed wiring board.

In other countries, they are refered to as "Printed Circuit Boards," or PCBs.

The printed board is doing the structure which stuck copper leaf on insulation board such as the epoxy material of glass or the epoxy material of paper or the material of bakelite. The copper leaf becomes a wiring part. There are 2 kinds of types in the printed board. They are single-sided printed board and double sided printed board.
To make the printed wiring board, leaving the necessary(wiring part) place of the copper leaf, the unnecessary(insulate part) place is lost. This work is called Etching.

There are several method to make the printed board.
Fundamentally the copper leaf is melted with the solution of the Ferric Chloride. The mask pattern is used to leave the wiring part.
You can write the mask pattern directly on negative printed board with oily ink. Some ink is melted by solvent, then you had better to check beforehand.

Even this method is good in the case that you make only 1 sheet. This time, I will introduce the method using positive exposure printed board as negative printed board. The sensitizer has been applied on negative printed board. The nature of the sensitizer when hit the light changes. Lighted part of sensitizer can melt to the developer and the part not lighted does not melt.

The mask is made with the black color ink where the part that you want to leave the copper leaf. (positive mask)Dissolve the copper leaf is called Etching.

Positive exposure printed board

The photograph left is the positive exposure printed board. The board is put in the bag for the shading. When it hits to the sunshine, the nature of sensitizer changes. It is different from the sensitive paper of the photograph and in a little light, it doesn't change.

You do not need to handle to a nervous temperament.

There are several kinds of size of the board. Then you choose board close to the device that you make.

It can even to cut and use that board and decide large. It needs to cut it so as not to hit to light much. It is serious a little bit.

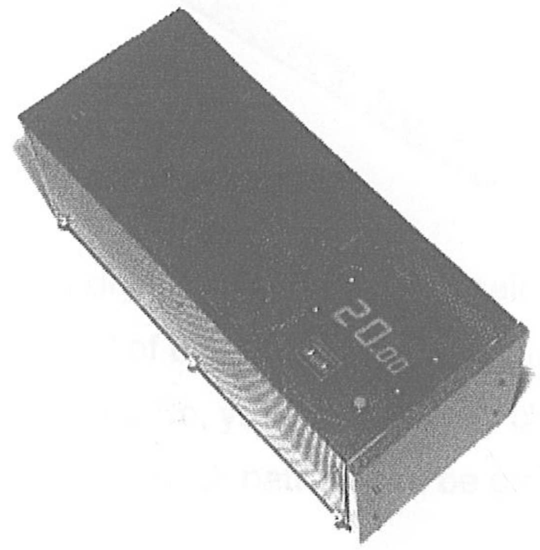
The size is about 150 mm x 100 mm to 300 mm x 200 mm.

Even the kind of the insulation board there is the epoxy material of glass or the epoxy material of paper.

The epoxy material of paper is cheap, but durability of it is bad a little. There is not a problem where I used.Photograph central under is the development medicine of a sensitizer. The development material is used with dissolving in hot water.

The pink colored bag of the photograph right is transparent film. The mask pattern is written to this film with the lettering tape or ink. I depict the pattern to the preparation of the block copy with the PC and print it with the printer.

Ultraviolet ray exposure equipment

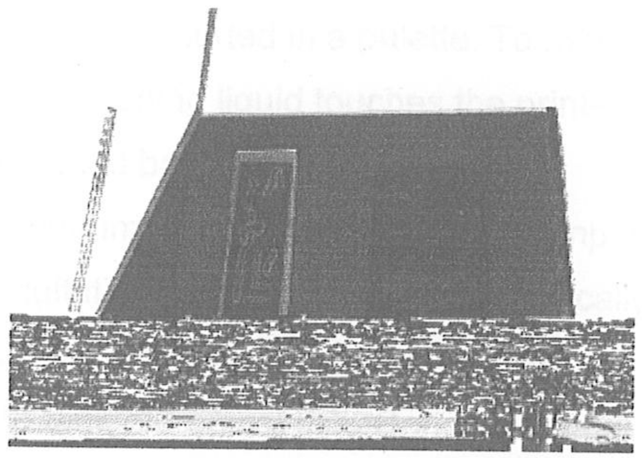


The ultraviolet ray is used when baking the print pattern on the positive exposure printed board. This equipment uses the ultraviolet rays which go out of the fluorescence light. It takes about 20 minutes for baking, so, this equipment has the timer circuit.

Resist pen

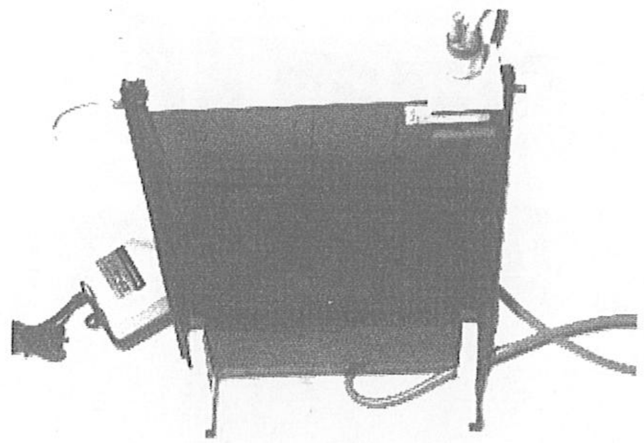
This is the oily dry ink to use for the repair of the mask pattern. Depending on the kind of the dry ink, there is one which has been dissolved in the etchant. So, you had better to check beforehand. With this pen, the direct mask pattern can be pictured, too.

Clamp



This is an instrument that sticks and fixes the mask pattern and the positive exposure printed board. The part that is seen black is made of the sponge. Piling up the mask pattern and the positive exposure printed board to the sponge and holds down and fixes with glass.

Etching apparatus



This is an instrument that melts the unnecessary copper leaf of printed board. Without using this instrument, you can make with the etching liquid that inserted in a palette. To melt well it is the hang to do as the new etching liquid touches the printed board and agitate a few of the printed board.

This instrument does and gives the temperature adjustment of liquid and agitation of etching liquid automatically.

You can see an air pump on the left. It is the same as the one that sends air to tank of tropical fish. Connecting this to the agitator tank with the vinyl pipe. Air blows out from under of the agitator tank and agitate liquid. The heater is attached on the right side of the agitator tank. As the temperature of liquid becomes regular with the thermostat. As for the temperature of liquid, 40°C to 43°C is appropriate and don't exceed absolutely over 45°C.

SOLDERING PROCEDURE

Soldering Process

Cleanliness is essential for efficient, effective soldering. Solder will not adhere to dirty, greasy, or oxidized surfaces. Heated metals tend to oxidize rapidly. This is the reason the oxides, scale, and dirt must be removed by chemical or mechanical means. Grease or oil films can be removed with a suitable solvent. Connections to be soldered should be cleaned just prior to the actual soldering operation.

Items to be soldered should normally be "tinned" before making a mechanical connection. Tinning is the coating of the material to be soldered with a light coat of solder. When the surface has been properly cleaned, a thin, even coating of flux should be placed over the surface to be tinned. This will prevent oxidation while the part is being heated to soldering temperature. Rosin-core solder is usually preferred in electrical work. However, a separate rosin flux may be used instead. Separate rosin flux is frequently used when wires in cable fabrication are tinned.

Tinning copper wire and cable

Wires to be soldered to connectors should be stripped so that when the wire is placed in the barrel, there will be a gap of approximately 1/32 inch between the end of the barrel and the end of the insulation. This is done to prevent burning the insulation during the soldering process and to allow the wire to flex easier at a stress point. Before copper wires are soldered to connectors, the ends exposed by stripping are tinned to hold the strands solidly together. The tinning operation is satisfactory when the ends and sides of the wire strands are fused together with a coat of solder. Do not tin wires that are to be crimped to solder less terminals or splices.

Copper wires are usually tinned by dipping them into flux (view A of figure 2-25) and then into a solder bath (pot) (view B of the figure). In the field, copper wires can be tinned with a soldering iron and rosin-core solder. Tin the conductor for about half its exposed length. Tinning or solder on the wire above the barrel causes the wire to be stiff at the point where flexing takes place. This will result in the wire breaking.

Basic soldering

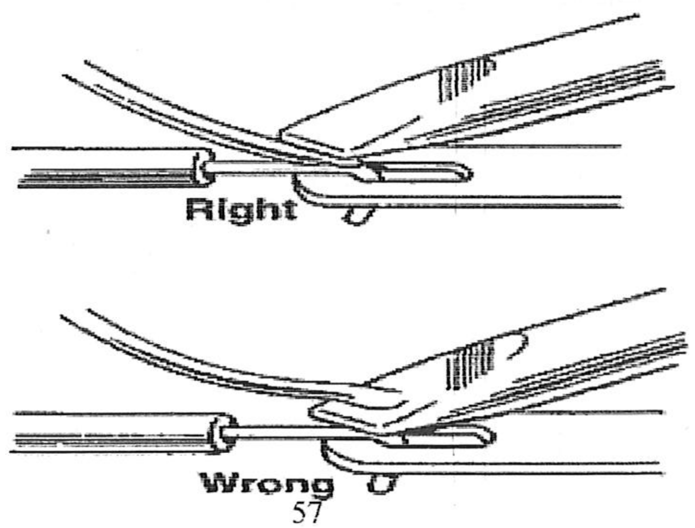
Overview

Soldering is accomplished by quickly heating the metal parts to be joined, and then applying a flux and a solder to the mating surfaces. The finished solder joint metallurgically bonds the parts - forming an excellent electrical connection between wires and a strong mechanical joint between the metal parts. Heat is supplied with a soldering iron or other means. The flux is a chemical cleaner which prepares the hot surfaces for the molten solder. The solder is a low melting point alloy of non ferrous metals.

Solder and Flux

Solder is a metal or metallic alloy used, when melted, to join metallic surfaces together. The most common alloy is some combination of tin and lead. Certain tin-lead alloys have a lower melting point than the parent metals by themselves. The most common alloys used for electronics work are 60/40 and 63/37. The chart below shows the differences in melting points of some common solder alloys.

Tin/Lead	**Melting Point**
40/60	460 degrees F (230 degrees C)
50/50 418 degrees F	(214 degrees C)
60/40 374 degrees F	(190 degrees C)
63/37 364 degrees F	(183 degrees C)
95/5 434 degrees F	(224 degrees C)



Most soldering jobs can be done with fluxcored solder (solder wire with the flux in a "core") when the surfaces to be joined are already clean or can be cleaned of rust, dirt and grease. Flux can also be applied by other means. Flux only cleans oxides off the surfaces to be soldered. It does not remove dirt, soot, oils, silicone, etc.

Base Material

The base material in a solder connection consists of the component lead and the plated circuit traces on the printed circuit board. The mass, composition, and cleanliness of the base material all determine the ability of the solder to flow and adhere properly (wet) and provide a reliable connection.

If the base material has surface contamination, this action prevents the solder from wetting along the surface of the lead or board material. Component leads are usually protected by a surface finish. The surface finishes can vary from plated tin to a solder - dipped coating. Plating does not provide the same protection that solder coating does because of the porosity of the plated finish.

The Correct Way to Solder

Some Reasons for Unwettability

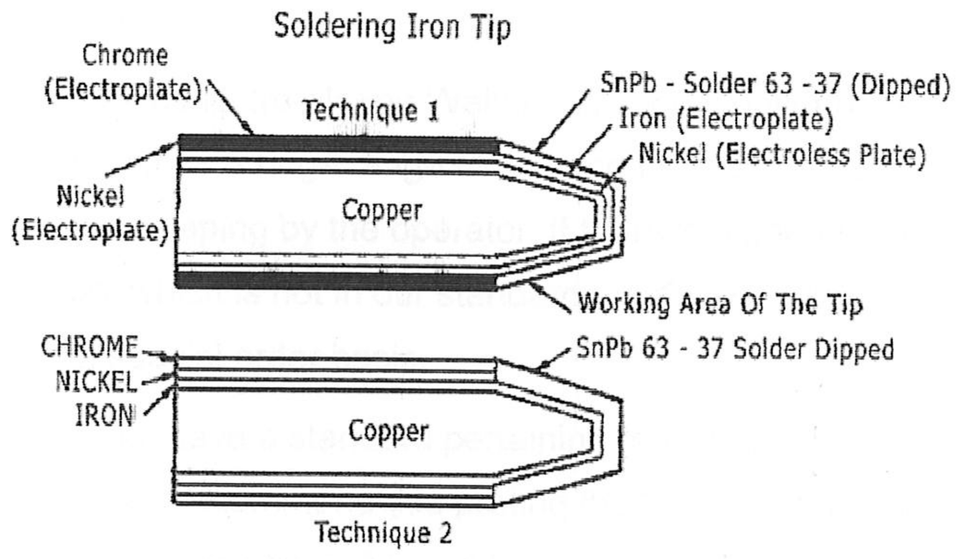
1. The selected temperature is too high. The tin coating is burnt off rapidly and oxidation occurs.
2. Oxidation may occur because of wrong or imperfect cleaning of the tip. E.G.: when other material is used for tip cleaning instead of the original damp Weller sponge.
3. Use of impure solder or solder with flux interruptions in the flux core.
4. Insufficient tinning when working with high temperatures over 665 degrees F (350 degrees C) and after work interruptions of more than one hour.
5. A "dry" tip, i.e. If the tip is allowed to sit without a thin coating of solder oxidation occurs rapidly.
6. Use of fluxed that are highly corrosive and cause rapid oxidation of the tip (e.g. water soluble flux).
7. Use of mild flux that does not remove normal oxides off the tip (e.g. no-clean flux).

The Soldering Iron Tip

The soldering iron tip transfers thermal energy from the heater to the solder connection. In most soldering iron tips, the base metal is copper or some copper alloy because of its excellent thermal conductivity. A tip's conductivity determines how fast thermal energy can be sent from the heater to the connection.

Both geometric shape and size (mass) of the soldering iron tip affect the tip's performance. The tip's characteristics and the heating capability of the heater determines the efficiency of the soldering system. The length and size of the tip determines heat flow capability while the actual shape establishes how well heat is transferred from the tip to the connection.

There are various plating processes used in making soldering iron tips. These plating operations increase the life of the tip. The figure below illustrates the two types of plating techniques used for soldering iron tips. One technique uses a nickel plate over the copper. Then an iron electroplate goes over the nickel. The iron and the nickel create a barrier between the copper base material and tin used in the solder alloy. The barrier material prevents the copper and tin from mixing together. Nickel-chrome plating on the rear of the tip prevents solder from adhering to the back portion of the tip (which could cause difficulty in tip removal) and provides a controlled wetted area on the iron tip. Another plating technique is similar but omits the nickel electro less plating, leaving the iron to act as the barrier metal.



What is a Weller*(r)* Tip - How Does It Work?

A Weller tip is made of a copper core, which is electro-plated with iron to extend the life of the tip. The non-working end of the tip is plated with nickel for protection against corrosion and then chrome plated to prevent the solder from adhering except where desired. The wet table part is tin covered.

The task of the tip is to store the heat, which is produced by the heating element, and to conduct a maximum amount of this heat to the working surface of the tip.

For fast and optimal heat transfer to the solder joint the tip mass should be as large as possible. When choosing a soldering tip always select the largest possible diameter and shortest reach. Use fine-point long reach tips only where access to the work piece is difficult.

How to Care For Your Tip

Because of the electro-plating Weller tips should never be filed or ground. Weller offers a large range of tips and there should be no need for individual shaping by the operator. If there is a need for a specific tip shape, which is not in our standard range, we can usually provide this on a special order basis.

Although Weller tips have a standard pertaining (solder coating) and are ready for use, we recommend you prating the tip with fresh solder when heating it up the first time. Any oxide covering will then disappear. Tip life is prolonged when mildly activated rosin fluxes are selected rather than water soluble or no-clean chemistries.

When soldering with temperatures over 665 degrees F (350 degrees C) and after long work pauses (more than 1 hour) the tip should be cleaned and tinned often, otherwise the solder on the tip could oxidize causing Unwettability of the tip. To clean the tip use the original synthetic wet sponges from Weller (no rags or cloths).

When doing rework, special care should be taken for good pertaining. Usually there are only small amounts of solder used and the tip has to be cleaned often. The tin coating on the tip could disappear rapidly and the tip may become uneatable. To avoid this the tip should be retained frequently.

Additional Tip and Triplet Care Techniques

Listed below are suggestions and preventative maintenance techniques to extend life and wet ability of tips and disordering tiplets.

1. Keep working surfaces tinned, wipe only before using, and retin immediately. Care should be taken when using small diameter solder to assure that there is enough tin coverage on the tip working surface.
2. If using highly activated rosin fluxes or acid type fluxes, tip life will be reduced. Using iron plated tips will increase service life.
3. If tips become unwettable, alternate applying flux and wiping to clean the surface. Smaller diameter solders may not contain enough flux to adequately clean the tips. In this case, larger diameter solder or liquid fluxes may be needed for cleaning. Periodically remove the tip from your tool and clean with a suitable cleaner for the flux being used. The frequency of cleaning will depend on the frequency and type of usage.
4. Filing tips will remove the protective plating and reduce tip life. If heavy cleaning is required, use a Weller WPB1 Polishing Bar available from your distributor.
5. Do not remove excess solder from a heated tip before turning off the iron. The excess solder will prevent oxidation of the wettable surface when the tip is reheated.
6. Anti-seize compounds should be avoided (except when using threaded tips) since they may affect the function of the iron. If seizing occurs, try removing the tip while the tool is heated. If this fails, it may be necessary to return the tool to Weller for

service. Removing the tip from the tool on a regular basis will also help in preventing the tip from seizing.

7. We recommend using distilled water when wetting the cleaning sponge. The mineral content in most tap water may contaminate your soldering tips.
8. Storing tips after production use:
   -- Clean hot tip thoroughly with damp sponge.
   -- Apply coating of solder to tip.
   -- Turn unit off to allow tip to cool.
   -- Put tip away in proper storage or in iron holder

How to "Renew" Your Tip

Emery cloth may be carefully used to wipe away oxidation when the tip is hot. The tip should then be immediately retinned to prevent further oxidation. In extreme cases of tip oxidation or "tip burnout" they may be cleaned using a soft steel brush along with an active flux. Once again, retinning the tip immediately is important.

Soldering Iron Temperature Settings

In order to raise the temperature of solder above it's melting point, soldering tip temperatures are usually set between 700 degrees F and 800 degrees F. Why such a high temperature when the most commonly used solders have a melting point under 400 degrees F? Using a higher temperature stores heat in the tip which speeds up the melting process. The operator can then complete the solder connection without applying too much pressure on the joint.

This practice also allows a proper formation of an intermetallic layer of the parts and solder. This is critical for reliable electrical and mechanical solder joints.

How Precise is the Indicated Tip Temperature?

Very fine long soldering tips have less heat conductivity than large short tips and therefore will run slightly cooler. Electronic control soldering stations have a tip temperature control accuracy of at least plus or minus 10 degrees F (6 degrees C) which is the current Mil Spec. Weller tips for electronic soldering tools are carefully designed to give accurate temperatures measured at the center of the solder wetted area. The specifications of the individual soldering stations are assured only if Weller tips are used. The sensor hole in these tips is very critical to their proper operation. Use of other than Weller tips may cause damage by overheating or tip freezing on the sensor or in the tool barrel.

Tip Temperature Measuring

Weller offers two methods for measuring tip temperature. One is a contact method which may yield low readings but is useful in verifying tip temperature stability and showing that the tip is within the desired range for soldering. The second method employs a welded thermocouple tip. This approach is based on using a standard calibration tip and results in much more accurate tip temperature measurements. Both methods require the use of the WA2000

Soldering Iron Analyzer. Please consult with your Cooper Tools representative or your local distributor for more information.

The Operator's Effect on The Process

The operator has a definite effect on the manual soldering process. The operator controls the factors during soldering that determine how much of the soldering iron's heat finally goes to the connection.

Besides the soldering iron configuration and the shape of the iron's tip, the operator also affects the flow of heat from the tip to the connection. The operator can vary the iron's position and the time on the connection, and pressure of the tool against the pad and lead of the connection.

When the tip of the iron contacts the solder connection, the tip temperature decreases as thermal energy transfers from the tip to the connection. The ability of the soldering iron to maintain a consistent soldering temperature from connection to connection depends on the iron's overall ability to transfer heat as well as the operator's ability to repeat proper technique.

The Reliable Solder Connection

Two connection elements must properly function for a solder joint to be reliable. The solder within the connection must mechanically bond the component to the PCB. The connection must also provide electrical continuity between the device and board. The proper intermetallic layer assures both.

Mechanical

In surface mount and nonclinched through-hole technology, the solder provides the mechanical strength within the connection. Important factors for mechanical strength include the wetting action of the solder with the component and board materials, physical shape and composition of the connection, and the materials' temperature within the connection during the process. The connection temperature should not be too high, causing embrittlement, or too low, resulting in poor wetting action.

Electrical

If a solder connection is mechanically intact, it is considered to be electrically continuous. Electrical continuity is easily measured and quantified.

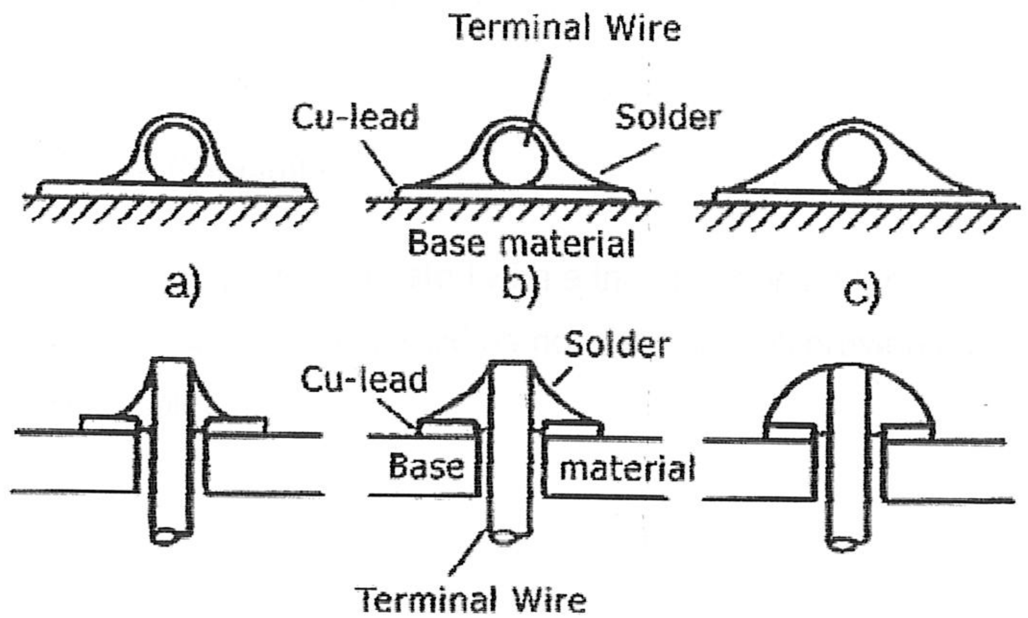
Recognizing the Reliable Solder Connection

Two easily measured indicators in the soldering process that can determine the reliability of the solder connection are the soldering iron's tip temperature and the solder's wetting characteristics. The tip's temperature during the soldering process is an indicator of the amount of heat being transferred from the tip to the connection. The optimum rate of heat transfer occurs if the soldering iron tip temperature remains constant during the soldering process.

Another indicator for determining reliability is the solder's wetting action with the lead and board materials. As operators transfer heat to the connection, this wetting characteristic can be seen visually. If the molten solder quickly wicks up the sides of the component on contact, the wetting characteristic is considered good. If the operator sees the solder is flowing or spreading quickly through or along the surface of the printed circuit assembly, the wetting is also characterized as good.

Right Amount of Solder

a) Minimum amount of solder
b) Optimal
c) Excessive solder

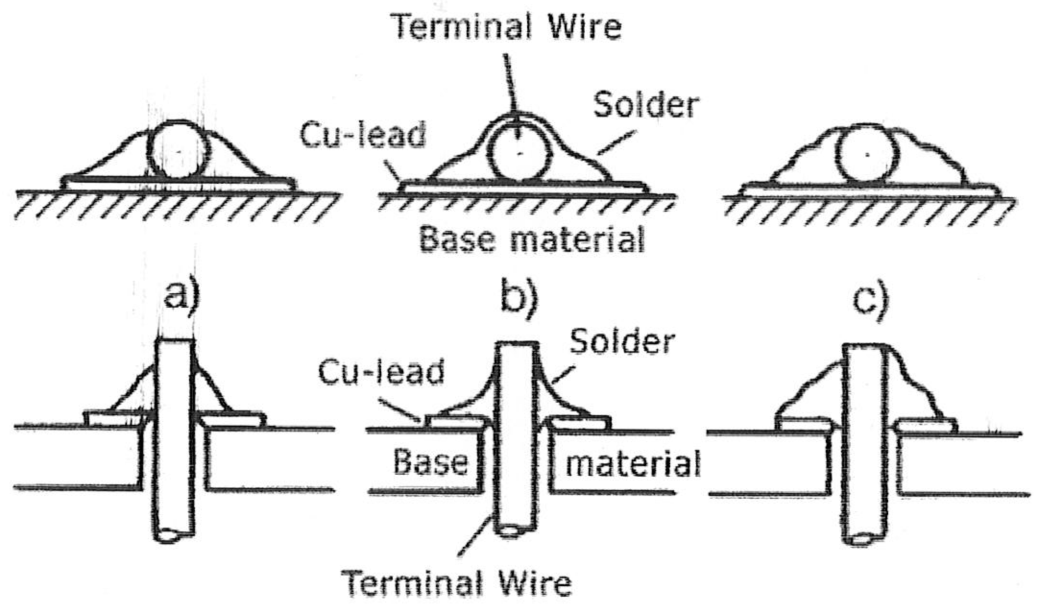


Solderability

a) Bad solderability of terminal wire

b) Bad soldering of PCB

c) Bad soldering of terminal wire and PCB



Key Points to Remember

1. Always keep the tip coated with a thin layer of solder.
2. Use fluxes that are as mild as possible but still provide a strong solder joint.

3. Keep temperature as low as possible while maintaining enough temperature to quickly solder a joint (2 to 3 seconds maximum for electronic soldering).
4. Match the tips size to the work.
5. Use a tip with the shortest reach possible for maximum efficiency.

WIRING

The wiring under IC1

This time, tin coated wire of 0.32 mm is used.The wiring between 16F and 18M does not need to pass under of IC1 (the parts side). This wiring is just to practice. To avoid the intersection of the connection, the twisted wire covered with soft insulation material can be used instead of tin coated wire. Choose in your taste. In my case, I make not to use twisted wire as much as possible for easiness of checking.

Don't mistake the position where passes the tin coated wire.(18F and 18K) This position becomes the starting position of wiring after this. It's better to mark at 18F with red colored pen.

First, pass the wire from wiring side to parts side at the hole of 18F. Pass that wire from parts side to the wiring side at the hole of 18K.
Put the wire of the 18K to the 18M. Hold down the wire of 18K to the printed board, and solder the 18K only. Pull the wire of the parts side as it becomes straight from 18F. Bend the wire of the wiring side toward 16F(right angle to the wire of the wiring side), hold down the wire of 18F to the printed board, and solder the 18F only.
Cut the wire at 16F.

Mount the IC socket

The hollow where shows the mount direction of IC is attached at IC socket. It has no problem functionally even if it is put in opposite direction. But it's better to put on right direction to avoid careless mistake.

Mount the IC socket to the parts side. 1st pin is put in accordance with 19E. Hold down the socket to the printed board, and solder the 19E(1st pin). And solder the 16I(10th pin). With this, the socket is fixed to the printed board.

Mount the resistor R1

Mount the resistor R1 to 14J and 14M with the same direction of color code. Bend the lead of 14M toward 13M a little. Bend the lead of 14J to 16J and cut it at 16J. Hold down R1 from the parts side, and adjust the angle of the lead of R1 of 14M side, and put the lead of 14J side lead of R1 to 16J. Solder the lead of 16J of R1 and the lead of IC socket together. Bend the lead of 14M to 14L, and bend more to 15K, and cut it at 15K. Solder the lead 14M.

Mount the diode D1

Mount D1 to the 15K and 15N from parts side. The anode side is 15K and the cathode side is 15N. Bend the lead of 15N to 15O a little. Cut and leave about 0.5 mm the lead of 15K of D1 that is bended toward 16J so as not to contact with surrounding land. Solder the lead of 15K of R1 and the lead of D1 together. Bend the lead 15N to 16N, and bend more to 17M, and cut it at 17M. Sloder the lead 14L and 16N.

Mount the capacitor C5

Mount the capacitor C5 to 17M(positive lead side) and to 18M(negative lead side). At the body of the capacitor the mark that shows negative side lead is printed. Do not mistake absolutely even this. Mount the capacitor, providing about 1-mm distance from printed board. Bend the lead of 17M to 17L, and bend more to 16K. At 16K there is the 8th pin of the IC socket so cut the lead of C5 at 16K so as not to overlap with the lead of IC socket. Solder the lead of C5 and IC socket at 16K together. Adjusted C5 vertically with printed board. Cut and leave about 0.5 mm the lead of 18M of C5 that is bended toward 18N so as not to contact with surrounding land. Solder the wire that passes under IC1 and the lead of C5 at 18M. Solder at 17M, 17L and 18L to the turn.

Mount the capacitor C6

Mount the capacitor C6 to 12H(positive lead side) and to 12I(negative lead side). Mount the capacitor, providing about 1-mm distance from printed board. Bend the lead of 12H to 12G a little. Bend the lead of 12I to 13J, and bend more to 14J, and cut it at 14J. Solder the lead of 12I of C6. This soldering is a little difficult because C6 is tall Adjusted C6 vertically with printed board. Bend the lead of 12H to 13I, and bend more to 14I and 15I, and cut it at 15I. Solder the lead of C6 and the lead of D2 at 15I together. Solder at 14J, 13J, 15J, 15H, 14I and 13I to the turn.

Mount the transistor TR1

Mount the transistor TR1 that flat face(name is printed) toward 9G, and the center lead(collector) to 9F, lead of emitter to 10E and the lead of base to 8E. If you don't understand, see 'Component lead of the transistor'. Mount the transistor, providing about 5-mm distance from printed board. Bend the lead of 9F toward 9G, 8E toward to 7E and 10E toward to 11E each to prevent it fall. Solder the lead of TR1 and R3 at 8E. (The hight of TR1 is fixed) Adjusted the TR1 vertically with printed board. Cut the lead of TR1 at 8E. Cut and leave about 0.5 mm the lead of 9F of TR1 that is bended toward 9G so as not to contact with surrounding land. Cut and leave about 0.5 mm the lead of 10E of TR1 that is bended toward 10D so as not to contact with surrounding land.

Mount the resistor R4

Mount the resistor R4 to 3D and 6D. Bend the lead of 3D of R4 to 2D a little. Bend the lead of 6D of R4 to 5E and bend more to 5F. Hold down R4 from the parts side, and adjust the angle of the lead of R4 of 3D side, and put the lead of 6D side lead of R4 to the 5F. Solder the lead of 6D of R4. Cut the lead of 6D of R4 at 5F. Cut and leave about 0.5 mm the lead of 3D of R4 so as not to contact with surrounding land

Mount the Light Emitting Diode D3(red)

The light emitting diode(LED) has polarity. The long lead is a anode side(connect to plus voltage) in the case of a new article. If you don't understand the polarity, see the page 'Light Emitting Diode'. Mount D3 to 5F(anode side) and 6F(cathode side) from parts side. Mount the LED, providing about 5-mm distance from printed board. Solder the lead of D3 and R4 at 5F together. Cut the lead of D3 at 5F. Bend the lead of D3 of 6F with confirmation of it's hight to 7F, and bend more to 8F and 9F, and cut it at 9F. Solder the lead of D3 and TR1 at 9F together. Solder at 5E, 6F, 7F and 8F to the turn.

Mount the transistor TR2

Mount the transistor TR2 that flat face(name is printed) toward 9L, and the center lead(collector) to 9J, lead of emitter to 10K and the lead of base to 8K. If you don't understand, see 'Component lead of the transistor'. Mount the transistor, providing about 5-mm distance from printed board. Bend the lead of 9J toward 9I, 8K toward to 7K and 10K toward to 11K each to prevent it fall. Solder the lead of TR2 and R5 at 8K. (The hight of TR2 is fixed) Adjusted TR2 vertically with printed board. Cut the lead of TR2 at 8K. Cut and leave about 0.5 mm the lead of 9J of TR2 that is bended toward 9I so as not to contact with surrounding land. Cut and leave about 0.5 mm the lead of 10K of TR2 that is bended toward 11K so as not to contact with surrounding land.

Mount the resistor R6

Mount the resistor R6 to 3L and 6L. Bend the lead of 3L of R6 to 2L a little. Bend the lead of 6L of R6 to 5K and bend more to 5J, and cut it at 5J. Hold down R6 from the parts side, and adjust the angle of the lead of R6 of 3L side, and put the lead of 6L side lead of R6 to 5J. Solder the lead of 6L of R6. Cut and leave about 0.5 mm the lead of 3L of R6 so as not to contact with surrounding land.

Mount the Light Emitting Diode D4(green)

The light emitting diode(LED) has polarity. The long lead is a anode side(connect to plus voltage) in the case of a new article. If you don't understand the polarity, see the page 'Light Emitting Diode'. Mount D4 to 5J(anode side) and 6J(cathode side) from parts side. Mount the LED, providing about 5-mm distance from printed board. Solder the lead of D4 and R6 at 5J together. Cut the lead of D4 at 5J. Bend the lead of D4 of 6J with confirmation of it's hight to 7J, and bend more to 8J and 9J, and cut it at 9J. Solder the lead of D4 and TR2 at 9J together. Solder at 5K, 6J, 7J and 8L to the turn.

Mount the capacitor C1

Mount the capacitor C1 to 21M(minus lead) and 22M(plus lead). Attention the polarity. Bend the lead of C1 of 21M to 20M, and 22M to 23M. Mount the capacitor, providing about 1-mm distance from printed board. Solder the lead 22M of C1 and cut it. Cut and leave about 0.5 mm the lead of 21M of C1 so as not to contact with surrounding land.

Mount the capasitor C2

Mount the capasitor C2 to 21K and 22K. Bend the lead of 21K to 20K, and 22k to 23K. Solder the lead of C2 at 22K and cut it. Cut and leave about 0.5 mm the lead of 21K of C2 so as not to contact with surrounding

Mount the three terminals voltage regulator IC2

Mount IC2 that flat face(name is printed) toward 23H, and the center lead(ground) to 21H, lead of input to 22I and the lead of output to 22G. If you don't understand, see 'Component lead of three terminal voltage regulator'. Mount the three terminal voltage regulator, providing about 5-mm distance from printed board. Bend the lead of IC2 of 21H to 20H, 22G to 23G and 22I to 23J. Solder the lead of IC2 at 22G and cut it. Cut and leave about 0.5 mm the lead of 21H and 21I of IC2 so as not to contact with surrounding land.

Mount the capasitor C3

Mount the capasitor C3 to 21E and 22E. Bend the lead of 21E to 20E, and 22E to 23E. Solder the lead of C3 at 22E and cut it. Cut and leave about 0.5 mm the lead of 21E of C3 so as not to contact with surrounding land.

Mount the capacitor C4

Mount the capacitor C4 to 21C(minus lead) and 22C(plus lead). Attention the polarity. Bend the lead of C4 of 21C to 20C, and 22C to 23C. Mount the capacitor, providing about 1-mm distance from printed board. Solder the lead 22C of C4 and cut it. Cut and leave about 0.5 mm the lead of 21C of C4 so as not to contact with surrounding land.

The ground wiring of C1, C2, C3, C4 and IC2

Solder the lead of C4 at 21C. Solder the tip of 0.32 mm tin coated wire(In the following, it is called only wire). (This is called presoldering.) Presoldering wire is taken to 21C as it goes along to the print board as much as possible from 21D direction and solder to 21C. (The first point holds the wire with the hand of an other side. and the wire can be fixed only melting the solder.)

At this time you do not worry about the finish of soldering. Because this soldering is only done tentatively, presoldering is not necessary. Cut the wire at 21O. Solder at 21H, 21E and 21K to the turn. At this time, desolder at 21C and 21M by useing solder sucker.

(For cleaning of the tentative solder)

Solder at 21C and 21M again.

Solder at 21D, 21F, 21G, 21I, 21J, 21L and 21N to the turn.

The ground wiring of IC1

Solder the lead of IC1 at 19K. Presoldering wire is taken to 19K as it goes along to the print board as much as possible from 19L direction and solder to 19K. Cut the wire at 19O, and solder it at 19N.
Solder at 19L and 19M to the turn.

The ground wiring of TR1 and TR2

Solder the lead of TR1 at 10E. Presoldering wire is taken to 10E as it goes along to the print board as much as possible from 10F direction and solder to 10E. Pull the wire and fix it to 10M by temporary soldering.
Solder at 10K first and solder at 10H, 10F, 10G, 10I and 10J to the turn.
Don't solder of 10F previously, solder of 10E is to have melted with the heat at the time of soldering of 10F. Bend the wire from 10N to 11O and

toawrd 12O direction. To solder of long distance, soldering is easy when it holds down with the clip shown with the photograph below.

Check the shortage of the power line

The mode of the tester is made low resistance value measurement. Connect the red side stick to 25K(ground) and connect the black side stick to 25I(+12V). It is not misconnection. Because, in case of analogue type tester the plus voltage of inside-battery appears to the black side stick, and minus voltage appears to the red side stick. Check that resistance is not almost zero-ohm. In my case, there is about 600-ohm. If I connect conversely, there is 40-ohm. In this case the voltage is opposite then there is no problem.

Connection of the power supply

Connect the AC adapter to the circuit via the connector. Connect the AC adapter to the power supply outlet. Check the output voltage polarity. (Which line is plus and minus or which confirms.) Absolutely do not short-circuit.

After confirmation, remove the AC adapter from the power supply outlet. The capacitor is including in the AC adapter. So the output voltage of AC adapter dose not drop soon. The electric discharge time depends on the value of the capacitor including in the AC adaper. Output it may be dangerous when it short-circuits carelessly. However, as for capacity is small, there is not case it gets broken even if it short-circuits. A little spark appears. Solder the power supply wire to the circuit. The red colored twised wire is soldered to 25I, and the black colored twised wire is soldered to 25K.

Check the power supply voltage

Clean the work bench. It is dangerous when there be metal etc., because the work from this supplies and carries out the power supply. It is for the sense. It's better to work on the insulation sheet. When you carry out it on the desk of the metal the paper or vinyl of a rather thick are spreaded. The sheet of the rubber is best. The tester is made the DC voltage measurement mode. The measurement range makes the range close to it with 15V over. Connect the AC adapter to the circuit. Connect the AC adapter to the power supply outlet. Touch the black side stick(minus) of the tester to 25K. (Other piont connected with 25K is available. e.g. some point from 11O to 24O

Check the +12V voltage.

It is OK that the voltage is over +12V at the following point. Even if the rating output voltage of AC adapter is +12V, actual output volage is not constant +12V. Usualy, the output voltage is high without loading. Under +15V is OK. In my case it was 13.7V. Touch the red side stick of the tester to 25I. Touch to 3L. Touch to 22M. The check point above are enough, because the wiring check is completed.

Check the +5V voltage.

It is OK that the voltage is +5V at the following point. The output voltage of three terminals voltage regulator is constant +5V. Touch the red side stick of the tester to 22G. Touch to 22C. Touch to 16E. Remove the AC adaper from the power supply outlet. Disconnect the power supply wire of the circuit from the AC adapter.

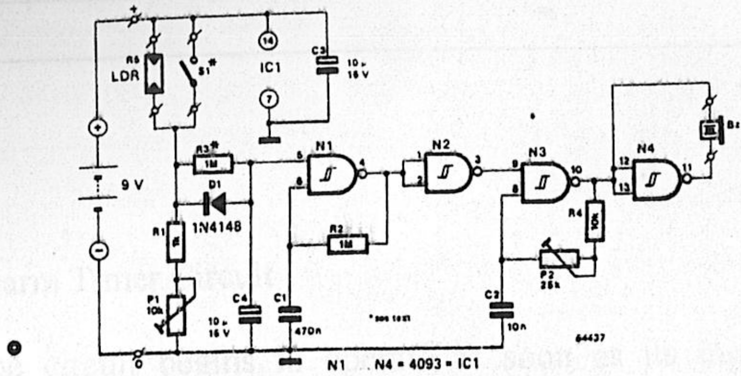
Mount the IC1

Check the voltage of +5V line and +12V line it is becoming zero-volt before mounting the IC1. Mount the IC1 to the IC socket. You pay attention

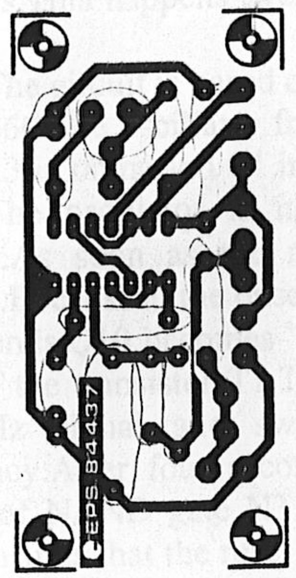
REFERENCES

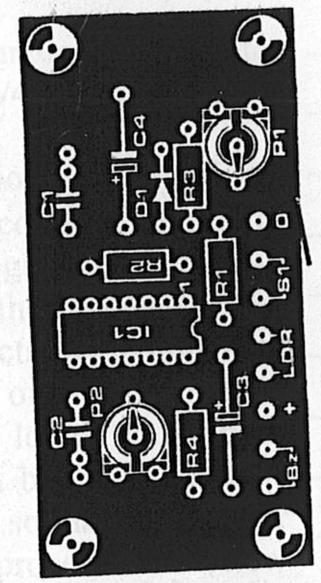
1. Campbell Reece Mitchell. ' Biology book'. Fifth edition.
2. J.V.Basmajian. ' Primary Anatomy'. Sixth edition. International Student Edition.
3. Farnell Catalogue 2002/03.
4. 302 circuit / Mcgraw & Hill
5. http://www.farnellnewark.com
6. http://www.btinternet.com
7. http://www.datasheet.com/project/ ICL 7136.
8. http://www.electronicshobiyist.com

ATTACHMENT



seconds until
age CMOS bi
is determine
connected t
it is logic l
Q4,Q7 and Q1
Hz signal on
stor therefore
e buzzer on
t Q7 also be
logic high,its
C2 briefly goe
ircuit is still '0





ELECTRONIC SYMBOLS

Buzzer 4–9 V = 2.

Parts list

Resistors:
R1 = 1 k
R2,R3 = 1 M (value of R3 may be reduced: see text)
R4 = 10 k
R5 = LDR
P1 = preset 10 k
P2 = preset 25 k

Capacitors:
C1 = 470 n
C2 = 10 n
C3,C4 = 10 µ/16 V

Semiconductors:
D1 = 1N4148
IC1 = 4093

Miscellaneous:
S1 = door (micro) switch
piezo electric buzzer
PP3 (9 V) battery with clips
case 100 × 50 × 40 mm
printed circuit 84437

A delay of about 10 s between the opening of the door and the sounding of the alarm is provided by the time constant R3C4. If faster reaction of the circuit is required, the value of R3 may be reduced to 220 k.
At the moment the threshold of N1 is exceeded, the gate commences to oscillate at a frequency of a few hertz. Each consequent rectangular pulse at the output (pin 3) of inverter N2 fires oscillator N3 which generates pulse trains whose rate amounts to a few kilohertz. The pulse trains are fed to inverter N4 which causes the piezo buzzer to emit a tone.
Without N2, oscillator N3 would work continuously when N1 is not being triggered: the output of N1 would then be high, and the logic 1 at pin 8 of N3 would cause the oscillator to function.
Inverter N4 serves to amplify the output of the buzzer. If the buzzer would simply be connected between the output of N3 and earth, the membrane would merely move from its rest position to one side. By connecting the buzzer across an inverter, its polarity is constantly reversed and this causes a doubling of the alternating voltage across it. Preset P2 provides further optimization of the volume by tuning N3 to the resonant frequency of the buzzer.
Preset P1 determines the sensitivity of the alarm: the smaller its value, the less sensitive the circuit is.
The alarm is most conveniently constructed on the printed-circuit board shown in figure 2.
Current consumption in the quiescent condition is of the order of 0.5 mA and when the alarm operates about 4 mA.

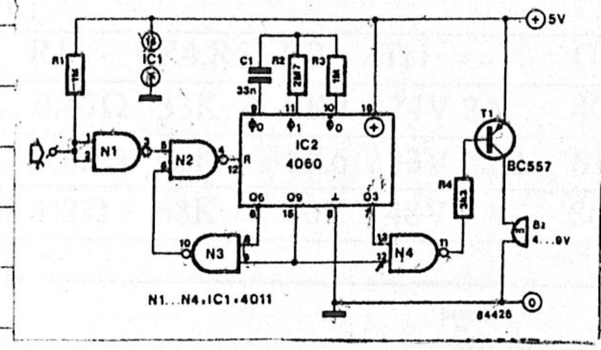
from an idea by W. Groot Nuland

Alarm Timer Circuit

The circuit begins to operate as soon as its input level becomes '0';after about 30 seconds the buzzer sounds four times at one seconds intervals.This happens every thirty seconds until the input goes logic high again.

The circuit is based on a 14 stage CMOS binary counter and oscillator type 4060.The oscillator frequency is determined by f=1/4R3C1,where f is in Hz,R3 in ohms and C1 in farad.

The oscillator is internally connected to the clock input of the counter.As soon as the reset input is logic low,the counter begins to operate.Because at the onset output Q4,Q7 and Q10 are logic '0'.After about 30 seconds,Q10 becomes '1' .The 1 Hz signal on Q4 is then applied to the base of the transistor T1.This transistor therefore conducts in rhythm with the 1Hz signal and switches the buzzer on and off at the same frequency.After four seconds output Q7 also becomes logic '1'.As both inputs of NAND gate N3 are now logic high,its output becomes '0'.This level ensures that the reset input of IC2 briefly goes high,so that the counter resets all outputs.If the input of the circuit is still '0',the process stars a new ;otherwise the alarm stays quiet.



Power Supply Circuit

The novelty of this design is that it has a variable output from 0V up without using a transformer with two secondary windings.The circuit can either be constructed with the well known 723IC,or for higher output voltages,the L146,which although less popular is still easily available.The choice is left to the constructor.The output current limitation is also variable,but once set it is continuously effective.Table 1 shows all the different component values needed to make three different versions(30,40 and 60V maximum output).

The circuit diagram illustrates the 40V/0.8A type.The L146IC was used because this can handle the higher output voltages far better thah the 723.Normally speaking,2V is the minimum regulated voltage which either IC can provide.Resistor networks R3,R4 and R5,R6 get over this restriction so that the output can be adjusted right down to practically 0V(with P2).

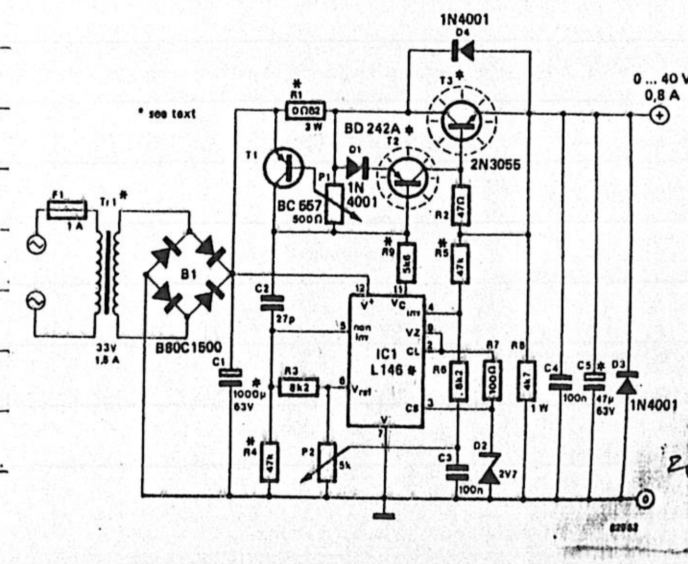
Another aspect of the design is the unusual way in which T3 is driven.When the required output voltage is below the tolerated minimum of the regulator ,the potential at pin 4 is below that of pin 5.This results in the IC trying to compensate for this by attempting to increase the output voltage from pin 9.This however does not work because pin 9 is earthed via R7 and D2,which limits the voltage increase.

When the voltage drop across R1 exceeds 0.6V,is shorted by R1,and T3 is cut off.During normal operation,the voltage drop across P1 is a constant 1.2V,made up of the forward voltage of D1 and the UBE of T2.

Table 1 indicates the component values needed to construct three different power supplies dependent on the voltage range required.Bear in mind to limit the output current sufficiently to keep the power dissipation of T3 under 40W.The maximum output current of a 40V version is 0.8A.It is possible to connect two 2N3055s in parallel(with emitter resistor) to double the output current,but then a 2A transformer is necessary.

Table 1 :

U out	I out	R1	R4,R5	R9	Tr1	C1/C5	IC1	T2
0-30V	1.3A	0.47Ω	33K	2K7	24V 2A	40V	723	BD242
0-40V	0.8A	0.82Ω	47K	5K6	33V 1.5A	63V	L14	BD242A
0-60V	0.6A	1.2Ω	68K	10K	48V 1A	80V	L146	BD242B

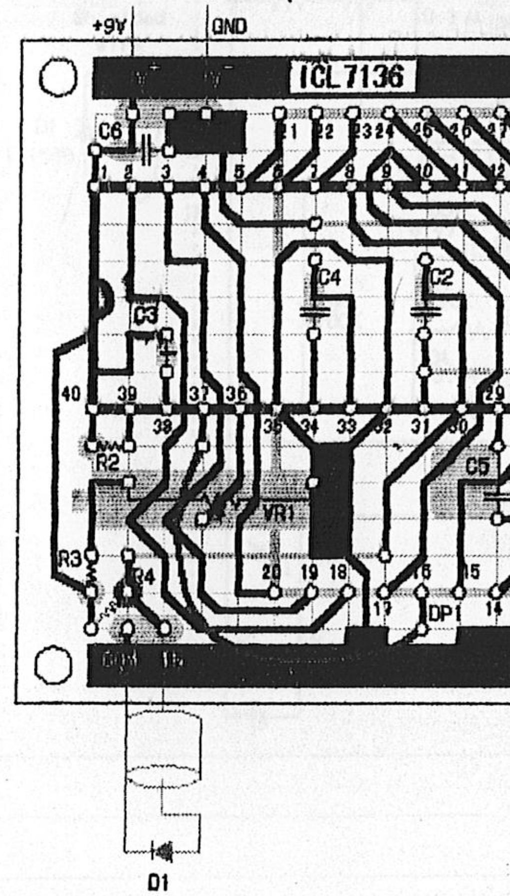


LCD Thermometer

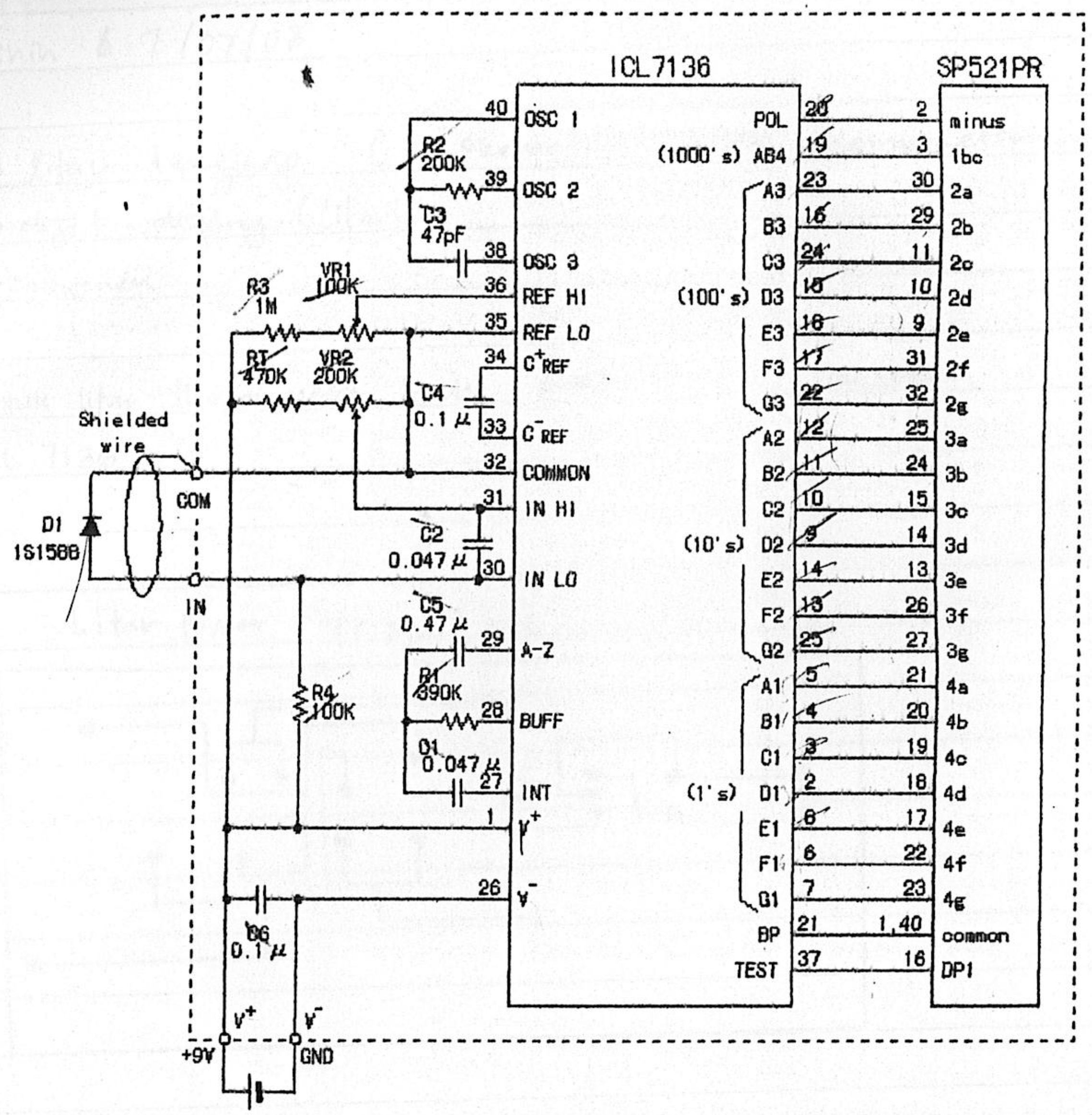
In this page, I introduce the LCD thermometer which is sold by "Akizuki Denshi Tsusho".
At this thermometer, the IC thermo sensor (S8100) or the diode (1S1588) is used as the thermo sensor.
When using the IC thermo sensor, the thermometry to +100°C from -40°C is possible. Also, when using the diode, the measurement to +150°C from -20°C is possible. Both sensors are contained in the kit.
This time, I used the diode as the thermo sensor to measure more than +100°C.
ICL7136 of Intersil (Harris) is used for the thermometer and is measuring the change of the forward direction minute voltages of the diode by the temperature. The 3-1/2 digits liquid crystal display (SP521PR) is used for the display. The most significant digit can display only "1".
The consumption electric power of ICL7136 is very small and it is possible to operate about 3 months continuously with the 9-V cell.

The main parts are contained in the kit. The plastic case and the cell are contained. But, there is not a connection cable of the sensor.

Pattern Drawing



Circuit Drawing



Isnin 6 7/07/03

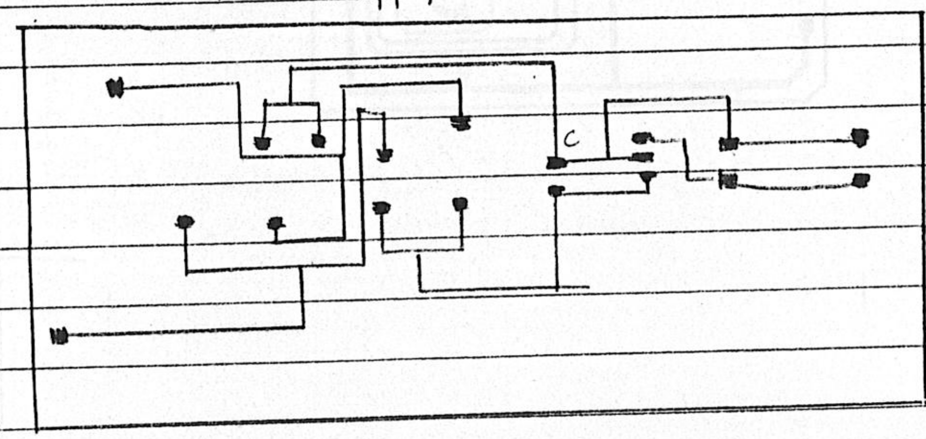
- 4 litar lengkap (2 skematik) untuk thermometer
- 3 digit display. (litar)
- komponen

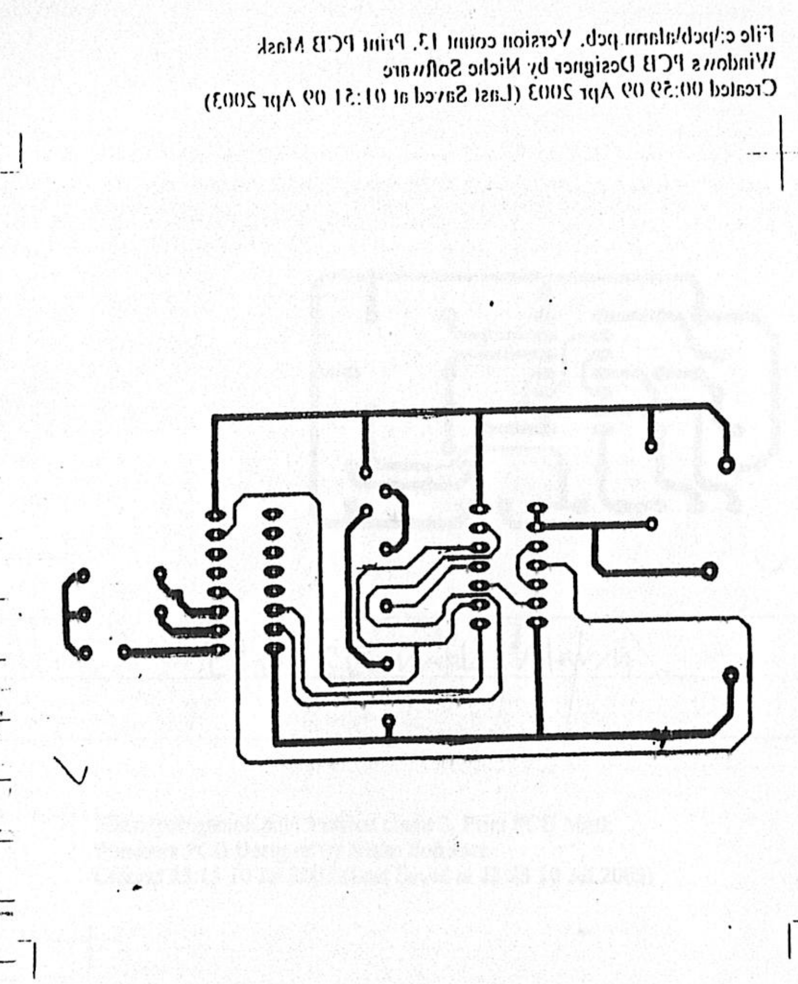
- Untuk litar thermometer, kalli perlu menempah

ICL 7136.

-

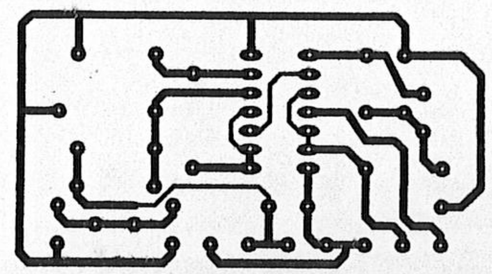
Litar power supply 9 volt





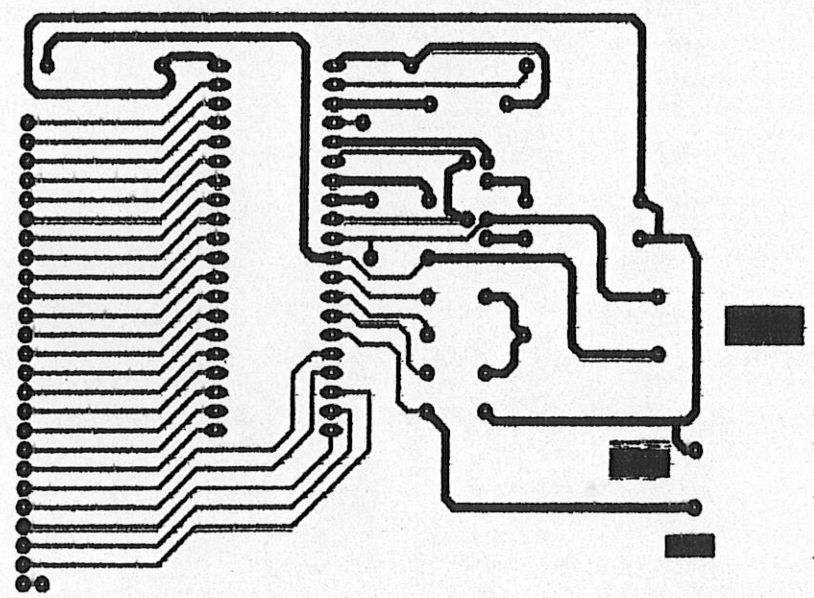
Litar Skematik Alarm timer

File c:\pcb\fridge.pcb, Version count 4, Print PCB Mask
Windows PCB Designer by Niche Software
Created 01:51 09 Apr 2003 (Last Saved at 02:23 09 Apr 2003)



Litar Skematik Alarm

File c:\pcb\gemok.pcb, Version count 3, Print PCB Mask
Windows PCB Designer by Niche Software
Created 23:15 10 Jul 2003 (Last Saved at 23:23 10 Jul 2003)



Litar Digital Thermometer