Experiment # 0

Printed Circuit Techniques

INTRODUCTION:

More than 40 years after they first appeared, the printed-circuit boards (PCBs) are still the most important means of connecting components into electronic systems. They have evolved from simple systems of etched copper strips connecting 30-50-component terminals on one side of a piece of phenolic board, to multiple layers of copper on polymer substrates connecting upwards of 10000 terminals in a three-dimensional pattern.

On early boards, the copper routes were laid out by hand, and the artwork delineating them photolithographically was hand-drawn; now, the entire process is computer-controlled. Then, conductor paths were 100 mils wide (0.1 inch or 2.5 mm); now they are as narrow as 3 mils. The main objectives of this experiments are:

1. To acquaint the student with printed circuit layout techniques.
2. To acquaint the student with the silk-screen process.
3. To understand and apply the correct procedures for conductor pattern masking.
4. To acquaint the student with etching techniques.
5. To acquaint the student with printed circuit component assembly and soldering.

PRELIMINARY DISCUSSION:

Printed circuit techniques are methods that can be employed in place of using hookup wire for the connection of electronic circuits. These methods allow conductor pattern to be reproduced into copper foil, which is initially bonded to an insulating base material, Fig. 1 a and b.

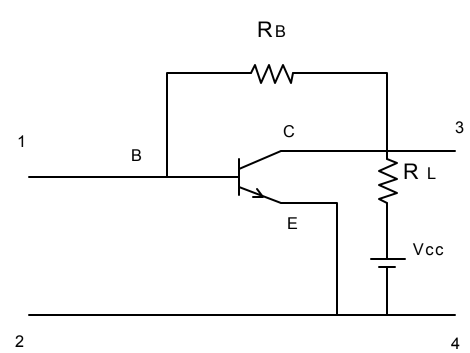

Fig. 1-a: Common Emitter

Copyright © Electrical Engineering Department, KFUPM.

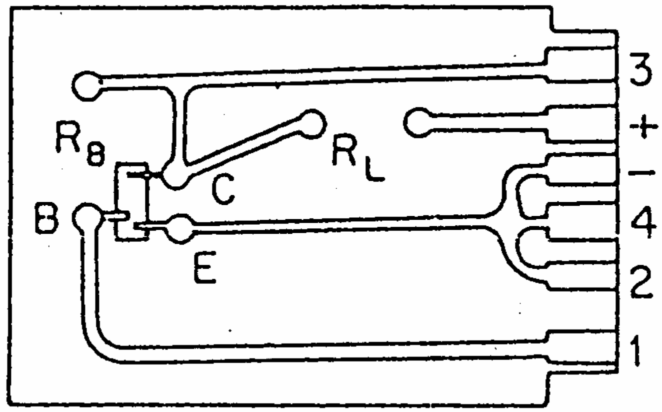

Fig. 1-b: Conductor Pattern

There are a number of procedures for producing conductive patterns. The most common are photoetching, stenciling, spraying, painting, die stamping, vaccum distillation, and chemical deposition. Some of the advantages of printed circuits are that they lend themselves to quantity production, dip soldering can be employed, component assembly size and weight can be reduced, and the probability of wiring errors is minimized.

The common materials that comprise PCBs are electrolytic copper foil bonded to a paperbase phenolic material. The foil may be bonded on one or on both sides of the base material forming single or double copper laminated board. For special purposes, nonstandard materials such as aluminum, steel, silver, and tinfoils are available. However, copper foil is used extensively because of its high conductivity, good soldering characteristics, availability in broad widths, and low cost. Copper foils are available in several thickness and weights. Depending on foil thickness and line width, the current-carrying capacity can range as high as 35 A.

The most important phase in the development of a final printed circuit assembly is the preliminary planning or design stage. There is no basic formula for selecting a PCB assembly. Attention to detail is essential and compromise in design will be experienced. Compromise is due to such considerations as one- or two-sided conductive pattern, pattern configuration, shielding, grounding requirements, component clearance, conductor separation and width, holes size and locations, board size and shape, and method of input and output connections. It is important to note that once the board is etched, the conductive pattern is permanent. Therefore, careful planning is necessary to make all corrections and modifications before the board is to undergo any stamping or chemical phases.

The two basic procedures that will be discussed in this section for producing PCBs are: 1. the silk-screen process and 2. masking with tape or etchant-resisting paint. The silk-screen process is intended for mass production of a common printed circuit pattern. Although this method gives fast and high-quality results, the tape or paint resist method of masking is quite suitable for producing small quantities of printed circuit having uncomplicated patterns.

The masking process using tape, paint resist, transfers, or ball-point tube resist is less involved than the silk-screen process. It can be done without special facilities or equipment to produce a complete masked conductor pattern. Before masking with either paint or tape resist, the copper should be completely cleaned to remove all grease and containmants. Two basic reasons for cleaning the copper surface before masking are to allow the resist to completely adhere to give optimum protection to the underlying conductor pattern and to allow the etchant to attack and remove the undesired portions of the copper. Avoid grease-base cleaners or abrasives since they leave a film. Fine steelwool may be used when appropriate cleaners are not available. After the cleaning process, the conductor pattern must be

 Copyright © Electrical Engineering Department, KFUPM.

transferred to the copper. This may be done by use of carbon paper. Only thin lines are necessary to show the centers of the conductors making up the conductor pattern. Care must be taken to secure the conductor pattern artwork so that it does not slip during the transfer process. When the paint or tape resist is applied, the correct conductor spacing and location will be obtained by centering the paint or tape width on the transfer lines. There is a wide choice of tape resist widths. The selection primarily depends on required current-carrying and conductor spacing. When applying tape resist and dots over the transfer lines and terminal points on the copper, each piece must be firmly pressed into place to ensure that no etchant can creep under the tape. This is particularly true at the joints between the tape and the dots, Fig. 2-a. When conductor direction changes, the tape does not have to be butted or overlapped. It is preferable to arch the tape to form the desired curves as the resist is being applied. Overlapping is required when the tape joins dots, but this can be avoided when teardrop resist is used at terminal points. The ends of the teardrops are squared off, thereby allowing for butt joints, Fig 2-b. There is less chance for etchant to creep into a butt joint than under a lap joint.

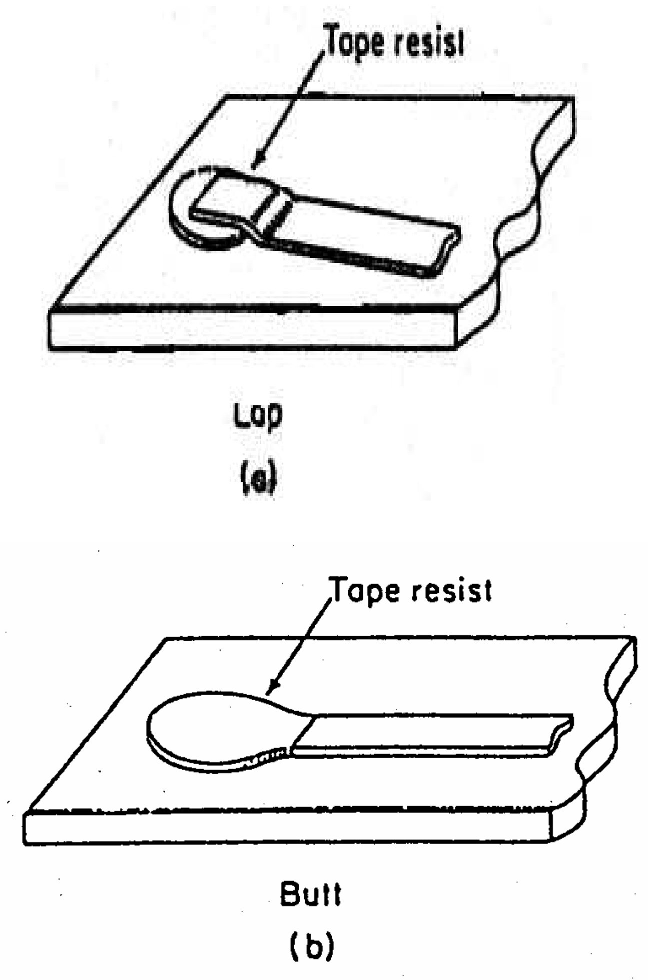

Fig.2:

If paint resist is used, a small, soft brush or an inking pen may be employed. An inking pen will give a more uniform line width. The dots of paint or tape resist placed at terminal points should always

 Copyright © Electrical Engineering Department, KFUPM.

be at least 1/8 in. in diameter. This will provide enough copper to allow a complete ring of solder to be formed around each of the conductor leads passing through the terminal pads. It should be emphasized that the appearance of the tape or paint resist pattern will be duplicated in the etched conductor pattern. If the resist extends beyond intended limits or falls short of them, the copper remaining after the etching process will follow suit. In the case of unintentional gaps in the tape resist, open circuits result in the conductor pattern. If the resist extends beyond set limits, there is an excellent possibility of short circuits being produced. In either case, the conductor pattern would have to be modified. Jumper wires would have to be used to bridge gaps and filing would have to be done to remove short circuits caused by excess copper. Therefore, the masked pattern must be checked before the etching process to avoid unnecessary modifications.

Alternatively, with a special sheet of plastic film-TEC200-a plain paper copier, and an iron, you can produce a professional looking printed-circuit board using what is probably the easiest and cheapest method of producing single or multiple printed-circuit boards. Making PCBs using TEC200 plastic film can be broken down into three easy steps:

1. Photocopy the printed-circuit pattern on to the plastic film.
2. Transfer the copied pattern on to the copper-clad blank (unetched printed-circuit material) by ironing.
3. Let the board cool and then peel off the plastic film.

The toner from the copy machine fuses to the blank, leaving a very tough etch-resist pattern that is also very precise.

It is important to use a "plain paper" type copy machine that uses dry toner. Lower-cost machines are often incapable of recognizing the film sheets (transparent medium) as paper, and as such will not copy to them.

Before handling the fil sheets, make sure that your hands are thoroughly clean, and then handle the sheets only by the edges. Body oils deposited on the plastic film will interface with the toner resist just as it does with other resists.

If you are copying the pattern from a printed page (magazine article, electronics-hobbyist construction book, etc.), the pattern shown is most-likely of the copper side of the board, therefore the image must be reversed. To reverse the pattern, simply copy the pattern to a sheet of TEC200, then copy the image from the first film sheet on to a second sheet of TEC200. The first sheet of film must be placed in the scanner up-side down (i.e. pattern side up), and covered with a clean sheet of paper to prevent a dirty machine-lid background from adding to your pattern.

The image produced on the second sheet of film is then placed toner-side down on a copperclad blank and backed up by a sheet of plain paper. A hot iron is then used to transfer the toner resist pattern on to the copper-clad blank.

The film sheet you used to get a mirror image may now be cleaned and reused, using nail polish remover, acetone, paint thinners, or other solvents that you may already have around.

You can design your own printed-circuit layout on a sheet of white paper using dry-transfer patterns (inserting component labels as you progress). Then when the layout pattern is complete, it can be transferred to TEC200 film. The pattern is then ready to be transferred to the printed-circuit blank, and your original layout (the one on paper) can be used as a parts-placement guide.

 Copyright © Electrical Engineering Department, KFUPM.

If you find minor deficiencies in the transferred pattern, touch them up with a fine-poiint permanent-ink marking pen. The blank is now ready for etching.

The etching solution will not undercut the pattern unless the etching rate is excessively slow, a condition usually caused by weak or cool solution, and/or the lack of agitation. Weak (overly used) solutions should be discarded. The temperature of the etching solution can be raised by subjecting it to a blast of hot air from a hair dryer or similar hot-air blower. Agitating the solution can be handled in several ways, for example by hand agitation (shaking the etching tank).

The silk-screen process is more complicated than the above mentioned masking techniques. A positive of the conductor pattern is first drawn or marked on Myler or drafting vellum. A photosensitive gelatin must be exposed under this positive of the conductor pattern. After the gelatin has been exposed, it is developed, inspected, and rinsed in hot water, producing a negative of the conductor pattern. While the gelatin is soft, it is pressed on to a silk screen and allowed to dry. The result is a stencil of the conductor pattern on the silk screen. When a printed circuit is to be masked, the copper-clad board is placed under the silk screen. Ink resist is forced through the screen with a squeegee on to the copper. The result is a very neat and completely masked printed circuit.

Once the entire blank is etched, clean the resist off with scouring powder, warm water (the warmer the better), and a plastic scouring pad. You can also use solvents to remove resist from the board.

Before components are mounted on to the PCB, the conductor pattern should be dip-soldered when several boards are to be made. When only a few boards are needed and the facilities for dip soldering are not available, the conductor pattern can be hand soldered. It is desirable to tin the conductor pattern for greater reliability and to facilitate soldering wires or component leads to the conductor pads. Excessive heat must be avoided in the tinning process and also when soldering leads to the conductor pattern. Overheating can upset the bond between the copper foil and the board. Once the conductor pattern begins to lift from the board, little can be done to repair it.

Wires and component leads must be carefully soldered to the conductor pattern. Before attempting to mount the parts and components on the printed circuit card, the terminal holes must be drilled. Holes must be at least 0.008-0.012 in. in diameter larger than the lead diameter. This will ensure that the solder will readily fill the gap between the component lead and the conductor pad. It is necessary to secure all components and parts mechanically to the board before soldering. To xsecure components, such as resistors, insert the leads through the opposite side of the board from the copper foil and bend approximately 60 degrees in the direction of the conductor. This will ensure that leads will not move when soldered, Fig. 3.

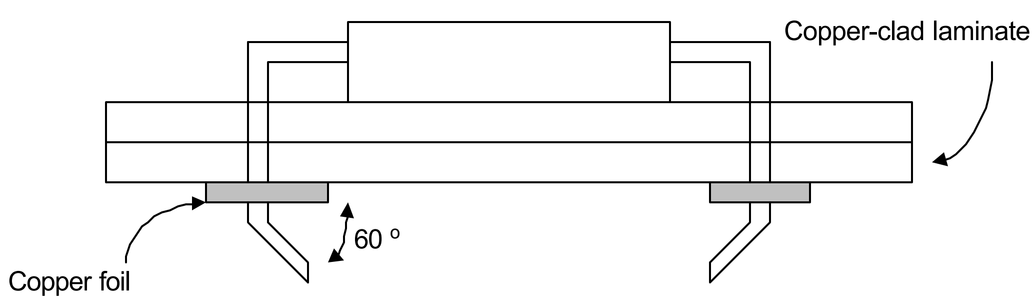

Fig. 3: Components mounting

 Copyright © Electrical Engineering Department, KFUPM.

PROJECT:

The following project is not intended to completely familiarize one in the printed circuit process or to teach all phases of this art. However, the general knowledge gained will allow one to succdessfully mask and etch a conductor pattern to establish electrical connections.

1. Lay out a conductor pattern for the circuit shown in Fig. 1(a) as shown in Fig. 1(b).
2. Cut the copper-clad laminate to the required size.
3. Thoroughly clean the surface of the copper.
4. Print the conductor pattern onto the copper-clad blank.
5. Allow the ink resist to dry.
6. Etch the printed circuit.
7. Place the etched card on to a hard supporting surface. Scribe and drill all terminal pads.
8. Remove the ink resist and tin the conductor pattern.
9. Mount the components on to the PCB.
10. Solder all wires and component leads to specified terminal pads. Cut ecess lead length protruding through the terminal pads per specifications.
11. Mount the components.
12. Clean all soldered terminal connections and conductive strips.

BY THE END OF THIS PROJECT WE HOPE THAT YOU WILL BE FAMILIAR WITH THE MOST IMPORTANT TECHNIQUES AND PROCEDURES USED IN PRODUCING PCBs. HOPEFULLY YOU WILL FIND IT USEFUL IN DESIGNING YOUR FUTURE PROJECTS.

Copyright © Electrical Engineering Department, KFUPM.