X190994

UNIVERSITY OF NORTH CAROLINA AT CHARLOTTE

Department of Electrical Engineering

Experiment No._____Printed Circuit Boards

INTRODUCTION

The printed circuit board (PCB) is a very useful way in which to mount and connect electronic components. PCB technology has developed rapidly since the advent of the transistor and integrated circuits (IC). In addition to being small and light, they can easily be mass produced.

The PCB is made of an insulating material such as phenolic, fiber-glass, or teflon that is copper-cladded on one or both sides. They are available with various thicknesses of copper-clad (generally specified in ounces). Basically, the electronic components are mounted on one side of the board with their leads soldered to copper conductors on the opposite side. This is also true for the double-sided board, except it will have copper conductors on both sides. The requirements for very high packing density (such as in space) has led to even a higher level of sophistication such as the multi-layer PCB. In recent years, computer software and technology have become available that makes “one of a kind” or small production quantities possible at a reasonable cost.

One of the disadvantages of the PCB compare with other methods such as the wire-wrapped board is the time it takes to prepare the PCB. Another disadvantage of the PCB is that it is difficult to modify once produced.

A considerable amount of paperwork and artwork is required in getting from the schematic diagram of a circuit to the finished PCB. The basic steps are listed below:

A. PARTS LIST: The parts list is generated from the schematic diagram. Oftentimes, the component specifications are also noted on the schematic. If not, specifications must be determined for each component. A schematic diagram of a flasher circuit is shown in Figure 1a.

B. COMPONENT DIMENSIONS: There are templates available for a large number of electronic components of various magnifications, i.e., 1X, 2X, etc. If template is not available for a particular component, then it's the dimensions must be determined physically.

C. WORKING SKETCH: Using the component dimensions, a rough sketch of the desired magnification is made of the circuit. This is basically laying out the circuit components on paper where they would be physically on the PCB as if LOOKING DOWN ON THE COMPONENT SIDE OF THE BOARD. An effort should be made to make the circuit as small and compact as possible. A working sketch of the example flasher circuit in Figure 1a is shown in Figure 1b.

IT IS MOST DESIRABLE THAT THERE BE NO JUMPER WIRES!

D. ARTWORK: Using computer software, the circuit is drawn using the working sketch as a guide. The widths of the conductors (lines) will be a function of the current requirements and the thickness of the copper-clad and should be chosen carefully. The “doughnut” pads are chosen depending on the component leads or pin dimensions. VERY CAREFUL CONSIDERATION MUST BE GIVEN TO THE SPACING OF THE LINES, THE PADS, AND THE COMPONENTS ON THE BOARD IN ORDER TO ASSURE THAT THERE IS ROOM FOR THE MOUNTING AND SOLDERING OF THE COMPONENTS ON THE BOARD. Any lettering that is to be read from the “bottom” of the board (opposite side from the components) should be drawn in reverse. The artwork made from the working sketch of Figure 1b is shown in Figures 1c and 1d.

E. PRINTED CIRCUIT BOARD: The printed circuit board should be large enough to accommodate the desired circuitry and any mounting requirements.

F. MOUNTING OF COMPONENTS: The printed circuit board should be drilled from the copper side of the board (usually with a No. 60 bit). CAUTION: To keep from breaking the bit, it should be COMPLETELY WITHDRAWN from the board BEFORE MOVING the board! The components should be mounted on the board before clipping their leads. Slightly bend the leads after the components are mounted to hold the components on the board while they are being soldered. Use a small wattage (approximately 50 watts) soldering

iron with a small, pencil-like tip. Place the hot tip of the soldering iron against the copper and the component lead allowing a few seconds for the copper and the lead to heat up. Apply a small amount of 60/40 rosin-core solder at the soldering iron tip and the copper-lead junction. The solder should flow freely around the lead and onto the copper forming a nice smooth fillet. The soldering process should take NO MORE THAN TEN SECONDS! After all the soldering is completed, clip off the excess leads just at the top of the solder. BE CAREFUL TO POINT THE BOARD AWAY FROM ANYONE OR ANY AREA THAT COULD BE DAMAGED BY FLYING METAL! There will be rosin flux residue left on the board which should be cleaned off using ethyl alcohol and a brush.

The purpose of this experiment is to introduce the student to the techniques of making a printed circuit board. For more complete details on the subject, the student is referred to the book, Intermediate Electronic Fabrication, by J. A. Markum and M. P. Silva, The Technical Education Press, 1976.

PRELIMINARY

P-1. You will given one of the circuits of Figure 2 by your instructor. Make a parts list for the circuit and include the component specifications.

P-2. Obtain a blank PCB and the parts for your circuit from the Electronics shop. Determine any necessary component dimensions.

P-3. Make a twice-size (2X) working sketch of the circuit given you by the instructor. The pinouts for the different ICs are shown in Figure 3. The PCB should be designed for a edge-connector that has an input-output configuration as shown below looking down on the component side of the board. There should be NO JUMPER WIRES!

P-3. (continued)

INSTRUCTOR'S SIGNATURE_____DATE_____

PROCEDURE

F-1. Make sure that the laboratory instructor has check the working sketch in Preliminary P-3 above for correctness before proceeding with the artwork.

F-2. Using the software, TANGO, produce the artwork for the given circuit from the working sketch done in Preliminary P-3 above. Include your initials, laboratory section, semester, and year (i.e., WAS1F87).

F-3. When the artwork is completed, turn in a printout and the computer disk to the laboratory instructor to have the PCB made. The laboratory instructor will return the PCB with the computer disk if the artwork is correct or the computer disk if the artwork needs to be corrected.

F-4. Clean the copper on the PCB with a “CU3 SOLUTION” and thoroughly rinse and dry the board. Wipe a small amount of “TINNING SOLUTION” onto the copper on the PCB until it looks silver.

F-5. Using the drill press provided, drill the printed circuit board BEING CAREFUL NOT TO BREAK THE SMALL BIT!

F-6. Using the proper tools, shape the PCB as needed to fit the edge connector.

F-7. Using the soldering iron and solder provided, mount the circuit components on the printed circuit board.

REPORT

R-1. The finished printed circuit board with components mounted will be presented for inspection and test the last week in the semester. It will be graded on layout (including lettering), soldering, and circuit operation.

R-2. The lab report will contain in place of the tabulated results the

- original circuit diagram
- 2x working sketch
- a printout of the artwork
- truth table
- bill of materials with manufacturer's specifications.

R-3. The analysis should include design considerations, fabrication problems, circuit function, and a discussion of the quality of the resulting printed circuit board (does it work?).