Printed circuit board design in a school computer laboratory

This item was submitted to Loughborough University's Institutional Repository by the/an author.

Citation: CHEN, 1999. Printed circuit board design in a school computer laboratory. IDATER 1999 Conference, Loughborough: Loughborough University

Additional Information:

- This is a conference paper.

Metadata Record: https://dspace.lboro.ac.uk/2134/1389

Publisher: © Loughborough University

Please cite the published version.
This item was submitted to Loughborough’s Institutional Repository by the author and is made available under the following Creative Commons Licence conditions.

For the full text of this licence, please go to:
http://creativecommons.org/licenses/by-nc-nd/2.5/
Printed circuit board design in a school computer laboratory

Yu-Charn Chen
San Francisco State University

Abstract
Printed Circuit Boards replaced conventional wiring in most electronic equipment after World War II, reducing the size and weight of equipment while improving reliability and uniformity. PCBs are used in all kinds of electrical and electronic products because they can be mass-produced with greater circuit density and also enable easier trouble-shooting.

Computer Aided Design (CAD) is critical in teaching PCB layout design but it is a challenge for school and college instructors with limited budgets. After discussion of current trends in PCB design and development, as well as basic PCB design criteria, an affordable PCB design using an "educational" microcomputer is presented.

Keywords: PCB, design, layout, via "educational" microcomputers
most of them have not changed significantly over the years, some specific trends continue to exert major influences on the types of PCBs. These trends have led to the larger use of non-organic base substrate, such as aluminum and soft iron. Nakahara (1996:3.1) found:

1. Computers and portable telecommunications equipment require higher-frequency circuits, boards, and materials, and also use more functional components that generate considerable amounts of heat that need to be extracted.
2. Consumer products have incorporated digital products into their design, requiring more functionality at ever-lower total cost.
3. Products for all uses continue to get smaller and more functional, driving the total circuit package itself to become more dense, causing the PWBs (PCBs) to evolve to meet these needs.

Nakahara (1996:3.1-3.14) further classified the PCB according to its base material, graphic or discrete wiring, physical nature, conductor formation, the number of conductor layers, plated-through-holes (PTHs), and process.

Basic PCB Design Criteria

Baer (1973:184-185) suggested the following should be considered when designing PCB layouts: drilling or spotting guides, 90 degrees guidelines, just one critical dimension, land pattern around each hole at least 0.31" in width, the diameter of the copper pad to be at least 0.17" larger than the hole size, sharp corners on conductor paths to be avoided, spacing between conductors or land to be increased at least 0.01" over the required minimum. Figure 1 shows preferred and unacceptable design for PCB assembly.

The following are some minimum design standards:

1. All components will be oriented on the X or Y axis if possible.
2. Components will be mounted on only one side of the PC board.

![Preferred (one axis and well organised)](image1)

![Unacceptable (multi-axis and poor organised)](image2)

![Preferred (two axes and well organised)](image3)

![Unacceptable (multi-axis and poor organised)](image4)

Figure 1 Preferred and unacceptable design for PCB assembly
3. Components should be located so that any component can be removed from the board without removing any other part.
4. All boards must have at least two holes referenced to the layer.
5. Maintain minimum spacing for resistors (0.1"–0.25"+), capacitors (0.1"–0.5"+), ICs.
6. Components which dissipate more than 2W should not be mounted on a PC board and should be heat sunk to the chassis.
7. Allow 0.002"/ounce for etching tolerance.
8. Conductor path shall be at least 0.025" to the edge of the board and components shall be at least 0.05" from the edge of the board.
9. Conductor paths should be oriented to the XY coordinate system, if possible. Avoid sharp angles which can cause foil delamination. Always use the shortest routing.
10. Maintain uniform pattern around holes.

PCB Design Using an "Educational" Microcomputer

Equipment Needed:
Microcomputer 486 or better, Mouse, Keyboard, Laser Printer, AutoCAD, Microsoft Word.

Design Procedure:
1. Prepare a pencil layout with grid paper and pencil. Check carefully the layout with the original schematic diagram as shown in Figures 2 and 3 to make sure the layout is correct.
2. Create the PCB artwork. To create the PCB artwork, you can run the AutoCAD program with an IBM or IBM compatible microcomputer. After successfully getting into the AutoCAD program, you can create some "donuts" to be used. AutoCAD has a distinct advantage over manual drafting in electronic drawing. You can create a "library" of symbols once and then retrieve the symbol in your drawings. You can also use the "COPY" command to duplicate the "donuts" or traces so they will be the same size. The command "ZOOM" allows you to enlarge or reduce your drawings. Generally speaking, a 0.020" wide trace can carry up to 1.5 A of current and a 0.025" wide one can take up to 15 Amperes. Any trace less than 0.020" wide is likely to be eaten up by the etchant, therefore, "It is better to stay with traces at least 0.040" wide, just to be safe" (Reis, 1989:110). Figure 4 and Figure 5 are examples of the PCB artwork for the schematic diagram in Figure 2.

Conclusion
Printed circuits have been used in all kinds of electronic products for a long time. However, a PCB layout design is still new to many high school and college educators. The author found that although AutoCAD and Microsoft Word were not originally designed for PCB purpose, they do provide students with the opportunity to explore the power of using a microcomputer in PCB design and with usable results.
Figure 4 Master layout of the "Touch Switch" PCB artwork (1)

Figure 5 "Negative" layouts of the PCB artwork (2)

References


- Reis, R. A. (1989), Electronic project design and fabrication, Merrill, Columbus.