

University of Pennsylvania

Electrical & Systems Engineering Undergraduate Laboratories

ESE 112: Introduction to Electrical & Systems Engineering

Lab 2b: Electronic Prototyping

Sansern Somboonsong

February 2008

Note: Hand in this lab report as part (continuation) of Lab2 - Light Sensor Lab.

Objective

The objective for this lab is to introduce students to electronic prototyping.

Background

Printed Circuit Boards (PCB)

Printed circuit boards or PCBs are thin plates on which electrical components are attached. A simple PCB consists of top and bottom layer separated by dielectric (non-conductive) layer. The PCB provides the electrical connections between different components through its traces. Figure 1 shows a picture of a PCB and its top copper layer. The lines on the PCB are called traces which are electrical connections equivalent to wires on a protoboard.

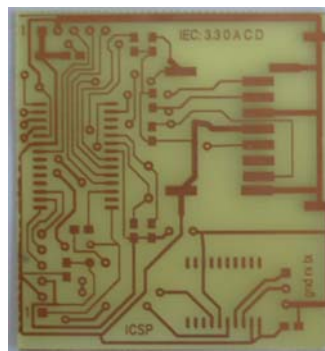


Figure 1 – PCB Top Layer.

The traces on the same layer cannot cross or it will cause the lines to short. To get round this problem PCBs usually have more than one layer. Traces from different layers can be connected by vias which are conductive holes in the PCB. Figure 3 shows a PCB with green solder mask which is present in most production grade PCBs. Solder mask prevents

the solder from burning the dielectric layer but more importantly, insulate exposed traces on the top copper layer. Although several different colors of solder masks are available, (such as red and blue) green is a popular choice. The PCB in figure 3 also has what is known as silk screening. Designers have the option to put text, company logo, and/or graphical indications on the solder mask to help in the assembly and debugging process. It is common practice to use silk screening to number components (ie R1 – for resistor 1) and draw a clearance outline of the components on the PCB. As an example, a PCB design may have resistors of different values. Having a numbering scheme (R1, R2, etc) and a spreadsheet that would match these numbers to a resistor value would reduce human errors during the assembly process. PCBs design can range in complexity from a single layer board with five components to a twenty layers board with thousands of components. The motherboard in the computer is an example of a more complex multiple-layer PCB design.

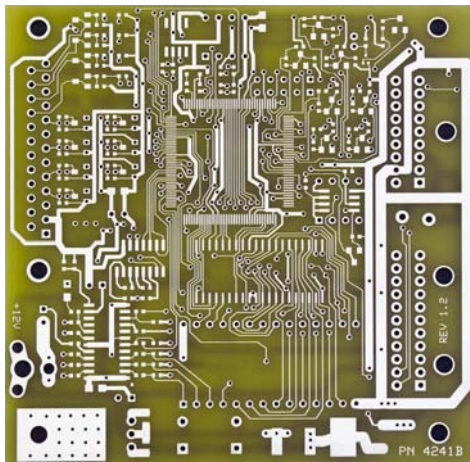


Figure 2 – PCB without Solder mask.



Figure 3 – PCB with solder mask

There are several computer programs that can be used in designing and laying out PCBs. Some powerful software suites can convert a schematic to a PCB layout, route (connect the traces on) the PCB automatically, and do error checking. The design files are usually sent to a company who then manufactures the PCB and ships it back to the designer. The designers have the option of assembling the components him or herself or have it done by a third-party company. The latter option involves shipping out the manufactured PCBs and components along with design files to the company that will assemble the PCBs. A few PCB manufacturers provide complete PCB manufacturing, assembly, and testing services.

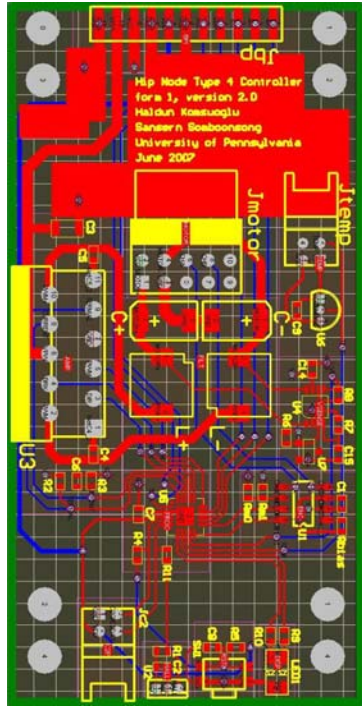


Figure 4 – View of a PCB in Protel DXP.

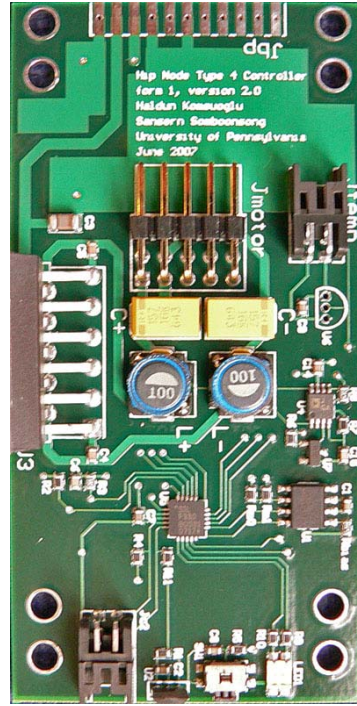
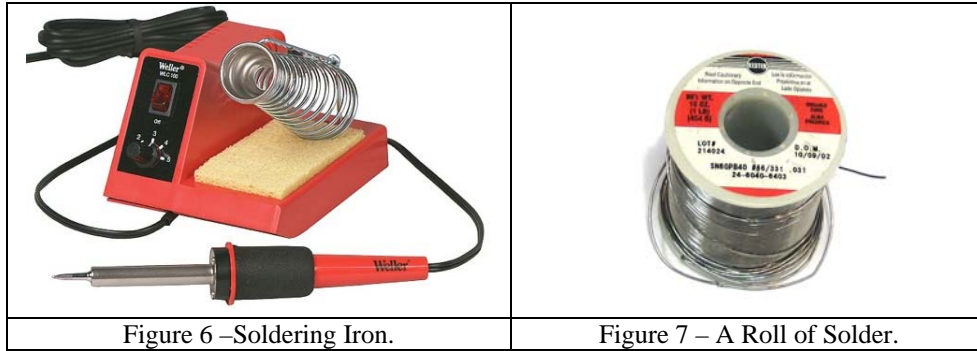


Figure 5 –PCB Assembled

Figure 4 shows a view of a PCB (EduBot motor control circuitry) in a design software (Protel DXP). Figure 5 shows a picture of the same PCB in assembled. The different colors represent different layers of the board. The two copper (conductive) layers are the top (red) and bottom (blue). The silk screen layer (yellow) is non-conductive.

Solder and Soldering Iron

To populate (put components on) a PCB, solder is used. Solder, a silver colored metal alloy (see figure 7) that is conductive and melts at around 350-400°F, holds the components onto the PCB. Solder typically contains lead to make it melt at a lower temperature. Directly inhaling fume created when solder is melted is therefore not recommended. A typical and cheap way to solder is to use a soldering iron. The component is placed on its footprint and the soldering iron's tip is used to melt the solder (see figure 6). The melted solder flows onto the component and the footprint on the PCB. The solder quickly cools when it is not in contact with the soldering iron and solidify holding the component in place. The footprint is the metallic pattern on the PCB on which components are supposed to be placed. Many manufacturers try to keep a standard footprint for their components to minimize the design effort of their customers. Other soldering methods that have higher equipment and maintenance costs include using hot air and infrared waves soldering.



Soldering irons comes in a wide range of prices from \$10-\$500. Better soldering irons come with a temperature control unit like that will be used in this lab is the one shown in figure 6.

Material

Clear Green LED
 CdS Cell
 LM358 Op-amp
 8-pin socket
 Resistors
 Potentiometer
 9V Battery with battery clip
 DMM
 Soldering Iron
 Solder
 PCB

Prelab (week 2)

- 1.) Building a circuit on a protoboard with wires sticking out in all directions may be a quick and cheap way to prototype electronic circuits. How would improve on the design, components, and material to make the light sensor circuit more robust and ready to be mass produced or manufactured and marketed?
- 2.) Search the web and explain in your own words the difference between surface-mounts and through-hole components.
- 3.) In laying out and routing PCBs with several active components, the designer must consider the current going through the trace. Explain how the traces on the PCB should be adjusted for different amount current flow.

Lab Instructions

- 1.) With the given PCB and the picture below, draw a schematic that would represent the PCB. With the knowledge from the previous lab, figure out where and what components should fit onto the PCB. You should build a circuit from the schematic you come up with, and then experiment on the protoboard to make sure the circuit works. A layered view of the PCB will be provided in the lab.
- 2.) When you are certain that the circuit will work, solder the components on the PCB.

Questions

- 1.) Explain the choices you made in part 1. Include your schematics in the report.
- 2.) Look up the information for components used on the PCB. Create a “Bill of Material” for the PCB. This should include the component description, manufacturer, manufacturer part number, vendor, vendor part number, and price per unit. (This is harder than it sounds)