Introduction to Photolithography Concepts
via printed circuit board (PCB) manufacturing

Introduction

As you saw on the video (http://www.youtube.com/watch?v=9x3Lh1ZfggM), photolithography is a way to nanomanufacture very small parts, such as working mechanical devices (gears, bearings, etc…) (MEMS picture) and electronic circuits. Parts that are manufactured on a small scale are often called Micro-electro-mechanical devices (or MEMS for short). Manufacturing MEMS requires very specialized and expensive labs (clean rooms) and lab equipment (photolithography instrument). However, we won't let this prevent us from learning the concepts of the photolithography process! We will manufacture a printed circuit boards (PCB) which uses a similar process to photolithography, but doesn't require specialized equipment or labs.

PCB Background Information (courtesy of Wikipedia)

A **printed circuit board**, or **PCB**, is used to mechanically support and electrically connect electronic components using conductive pathways, or traces, etched from copper sheets laminated onto a non-conductive *substrate*.

PCBs are rugged, inexpensive, and can be highly reliable. They require much more layout effort and higher initial cost than either wire-wrapped or point-to-point constructed circuits, but are much cheaper and faster for high-volume production. Much of the electronics industry's PCB design, assembly, and quality control needs are set by standards that are published by the IPC organization.

Manufacturing

Materials

Conducting layers are typically made of thin copper foil laminated to a non-conductive substrate. The non-conductive layer, known as prepreg in the PCB industry, can be made from different materials, such as FR-2 (Phenolic cotton paper), FR-3 (Cotton paper and epoxy), FR-4 (Woven glass and epoxy), FR-5 (Woven glass and epoxy), FR-6 (Matte glass and polyester), G-10 (Woven glass and epoxy), CEM-1 (Cotton paper and epoxy), CEM-2 (Cotton paper and epoxy), CEM-3 (Woven glass and epoxy), CEM-4 (Woven glass and epoxy), CEM-5 (Woven glass and polyester).

Patterning (etching)

The vast majority of printed circuit boards are made by bonding a layer of copper over the entire substrate, sometimes on both sides, (creating a "blank PCB") then removing unwanted copper after applying a temporary mask (eg. by etching), leaving only the desired copper traces. A few PCBs are made by adding traces to the bare substrate (or a substrate with a very thin layer of copper) usually by a complex process of multiple electroplating steps.

There are three common "subtractive" methods (methods that remove copper) used for the production of printed circuit boards:

1. **Silk screen printing** uses etch-resistant inks to protect the copper foil. Subsequent etching removes the unwanted copper. Alternatively, the ink may be conductive, printed on a blank (non-conductive) board. The latter technique is also used in the manufacture of hybrid circuits.
2. **Photoengraving** uses a photomask and chemical etching to remove the copper foil from the substrate. The photomask is usually prepared with a photoplotter from data produced by a technician using CAM, or computer-aided manufacturing software. Laser-printed transparencies are typically employed for *phototools*; however, direct laser imaging techniques are being employed to replace phototools for high-resolution requirements.
3. **PCB milling** uses a two or three-axis mechanical milling system to mill away the copper foil from the substrate. A PCB milling machine (referred to as a 'PCB Prototyper') operates in a similar way to a plotter, receiving commands from the host software that control the position of the milling head in the x, y, and (if relevant) z axis. Data to drive the Prototyper is extracted from files generated in PCB design software and stored in HPGL or Gerber file format.

Drilling

Holes, or vias, through a PCB are typically drilled with tiny drill bits made of solid tungsten carbide. The drilling is performed by automated drilling machines with placement controlled by a drill tape or drill file. These computer-generated files are also called numerically controlled drill (NCD) files or "Excellon files". The drill file describes the location and size of each drilled hole.

When very small vias are required, drilling with mechanical bits is costly because of high rates of wear and breakage. In this case, the vias may be evaporated by lasers. Laser-drilled vias typically have an inferior surface finish inside the hole. These holes are called *micro vias*.

It is also possible with *controlled-depth* drilling, laser drilling, or by pre-drilling the individual sheets of the PCB before lamination, to produce holes that connect only some of the copper layers, rather than passing through the entire board. These holes are called *blind vias* when they connect an internal copper layer to an outer layer, or *buried vias* when they connect two or more internal copper layers and no outer layers.

The walls of the holes, for boards with 2 or more layers, are plated with copper to form *plated-through holes* that electrically connect the conducting layers of the PCB. For multilayer boards, those with 4 layers or more, drilling typically produces a *smear* comprised of the bonding agent in the laminate system. Before the holes can be plated through, this *smear* must be removed by a chemical *de-smear* process, or by *plasma-etch*.

Exposed conductor plating and coating

The places to which components will be mounted are typically plated, because bare copper oxidizes quickly, and therefore is not readily solderable. Traditionally, any exposed copper was plated with solder by hot air solder levelling (HASL). This solder was a tin-lead alloy, however new solder compounds are now used to achieve compliance with the RoHS directive in the EU, which restricts the use of lead. Other platings used are OSP (organic surface protectant), immersion silver (IAg), immersion tin, electroless nickel with immersion gold coating (ENIG), and direct gold. Edge connectors, placed along one edge of some boards, are often gold plated.

Electrochemical migration (ECM) is the growth of conductive metal filaments on or in a printed circuit board (PCB) under the influence of a DC voltage bias.[1][2]

Solder resist

Areas that should not be soldered to may be covered with a polymer solder resist (solder mask) coating. The solder resist prevents solder from bridging between conductors and thereby creating short circuits. Solder resist also provides some protection from the environment.

Lab Exercise

Part 1 *– Hardwire a basic LED circuit*

Given the 9V battery, battery holder, resistor (200 or 330 ohms), and the LED, wire up all of the components is SERIES. NOTE: The resistor is **required** to prevent the LED from burning out. Replacement LED's will cost you 25 cents the first time, 1$ the second time, etc… Also note that LED's are *polarity sensitive* (there is + side and a - side). If you hook up the LED's with the wrong polarity, the LED will NOT light.

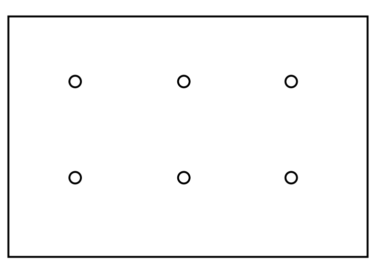
Using the symbols below, draw a complete circuit. Once you have this drawing complete, wire up the circuit. Get your teachers signature once the LED lights up.

Teacher Signature: ________________________________

Part 2 *– Recreate the hardwired circuit utilizing a PCB*

Using the pre-drilled PCB that is provided to you, your goal is to recreate your hardwired circuit, only this time you can't "twist" wires together like you did in part 1. In part 2, you will make a complete circuit using copper **traces**, and you will affix the LED and resistor to the PCB with soldered connections.

A) On the area below, sketch the areas where you will need a copper trace to make a complete circuit (class discussion).



Teacher Signature: ________________________________

B) Once your teacher has approved your sketch on page 4, using a Sharpie marker, draw your copper traces on the PCB. In order to solder your parts onto your PCB, you will also need to outline the drilled holes with the Sharpie. NOTE: The area where the sharpie is drawn is known as a ***MASK***.

C) The ***subtractive*** method: Now clean the "masked" PCB using a sponge soaked with etchant solution (ferric chloride). The etchant will remove all the copper that is NOT covered with a MASK (hence the name, the subtractive method). Once the copper is removed and only your mask remains, clean off your PCB in water and dry with a paper towel.

D) Solder your parts to the PCB and verify that you have a working circuit.

Conclusion Questions:

1) **Masks**
   a. Name and briefly describe three different ways a mask can be made on a PCB:

   b. What is the purpose of the mask?

   c. How is a mask made in the photolithography process?

2) Subtractive Process

a. What is meant by the subtractive process in the manufacturing of a PCB?

b. In making the PCB, what did you use to accomplish the subtractive process?

c. In the photolithography process, what is used to accomplish the subtractive process?

3) Substrate

a. What is a substrate?

b. What is a common substrate used the PCB process?

c. What is a common substrate used in the photolithography process?

4) Photolithography process

a. What specific equipment is required in the photolithography process?

b. What is a "clean room? (obviously not your bedroom, but relating to the nanotechnology industry)

c. Does the photolithography require a cleanroom, why or why not?

Materials List

Basic lab

1) LED's - $0.20 ea
2) 9V battery snaps – $0.19 ea
3) 330 ohm resistors - $3.00 for a package of 200 resistors
4) Blank ½ oz. Copper Board (single sided)* - $17.00 for two 8" x 10" panels
5) Etchant - $17.95 (makes 1 gallon of etchant)
6) Sponges
7) Water
8) Black Sharpie Marker

Advanced lab – items 1 through 7 above, and the following:

9) Laminator* (one required per classroom) - $69.95 ea
10) Toner Transfer Paper* - $14.95 for ten 8-1/2 x 11" sheets
11) Green Toner Reactive Foil* - $8.95 per pack
12) Simulation software
13) Printed Circuit Board software – free software available from http://www.pad2pad.com/
14) Laser printer or copier

Items marked with an asterisk can be purchased at http://www.pulsarprofx.com/PCBfx/main_site/pages/store/_store.html

All other items above can be purchased from Kelvin (http://www.kelvin.com) or a comparable supplier.