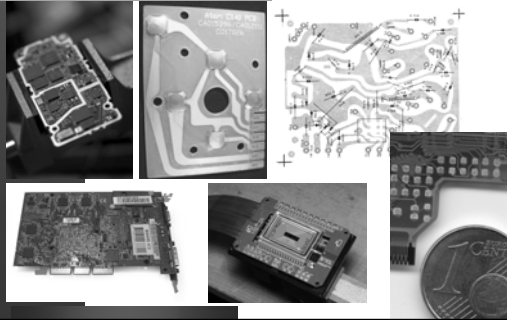# PCB Board Design

Babak Kia
Adjunct Professor
Boston University
College of Engineering
Email: bkia -at- bu.edu

ENG SC757 - Advanced Microprocessor Design

PCB boards

What is a PCB board

- **Printed Circuit Boards are primarily an insulating material used as base, into which conductive strips are printed**
- **The base material is generally fiberglass, and the conductive connections are generally copper and are made through an etching process**
- **The main PCB board is called the motherboard, the smaller attachment PCB boards are called daughter boards or daughter cards**

What is PCB Board Design

- **PCB board design defines the electrical pathways between components**
- **It is derived from a schematic representation of the circuit**
- **When it is derived, or imported from a schematic design, it translates the schematic symbols and libraries into physical components and connections**

Manufacturing Process

Step 1: Film Generation

The film is generated from the design files (gerber files) which are sent to the manufacturing house. One film is generated per layer
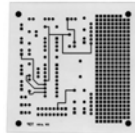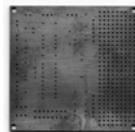
Step 2: Raw Material

Industry standard 0.059" thick, copper clad panel
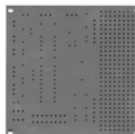
Manufacturing Process

Step 3: Drill Holes

Using NC machines and carbide drills to drill holes according to the drill spec sent to the manufacturing house
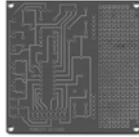
Step 4: Electroless Copper

Apply thin copper deposit in hole barrels
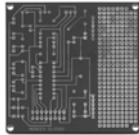
Manufacturing Process

Step 5: Apply Image

Apply photosensitive dryfilm to panel and use a light source and the film to expose the panel
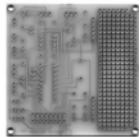
Step 6: Pattern Plate

Electrochemical process to build copper in the holes and on the trace areas. Apply tin to surface

---

Manufacturing Process

Step 7: Strip and Etch

Remove the dryfilm, and then etch the exposed copper. The tin protects the copper circuitry from being etched away

Step 8: Solder Mask

Apply a solder mask area to the entire board with the exception of solder pads

---
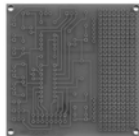
Manufacturing Process

Step 9: Solder Coat

Apply solder to pads by immersing into tank of solder. Hot air knives level the solder when removed from the tank

Step 10: Nomenclature

Apply white letter markings using screen printing process

Source: http://www.pcbpro.com/pcb-tutorial.php

PCB Design Layers

- A PCB design package allows the designer to define and design on multiple layers
- Many of these are physical layers such as:
  - Signal Layer
  - Power plane Layer
  - Mechanical Layer
- And some are special layers such as:
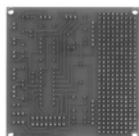 - Solder & Paste Mask Layers
  - Silkscreen or Top Overlay Layers
  - Drill guides
  - Keep-out Layer
- And some CAD specific display-only layers:
  - Ratsnest layer
  - DRC Errors

Signal Layers

- Signal Layers are the tracks that represent where copper needs to be placed
- They are designed in the positive
- There are no conventions for how thick signal traces can be. The width of your signals depend on
  - Electrical nature of the trace (e.g. power traces are generally thicker than signal traces)
  - Routing and space constraints
  - Manufacturing constraings! Not all manufacturing houses/processes can handle fine traces
  - For signals you can use 8 mil trace widths with 8 mil spacing
- Thicker traces provide lesser resistance and inductance

Internal Layers

- Internal Layers are generally used for Power Planes
- They are drawn in the negative, in other words tracks placed on this layer represent the void
- A signal name such as VCC or GND can be assigned to an Internal Plane and the CAD tool will automatically connect pads to the plane, greatly simplifying PCB routing
- Internal planes can also be split into sub planes or split planes

Mechanical & Keepout Layers

- **Mechanical Layers are special in that they can be added or viewed in any layer**
- **Used to provide drill and cut-out guides during the manufacturing process**
- **Keepout layers generally follow the mechanical layer to indicate to the CAD tool an area should not be routed over. This is useful as an auto-routing constraint.**
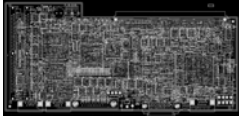
Solder Mask Layer

- **Solder mask is usually the green coating on a PCB board which is designed to insulate and protect the underlying copper traces from enviornmental factors, and is also used to prevent bridging (shorting) traces during wave soldering**
- **Solder mask usually covers everything on the PCB board except for pads and vias, though it is good practice to cover vias, especially if dealing with BGA components. This process is called *tenting* the vias**
- **Solder mask is shown on the CAD tool as a negative image. I.e. where there is solder mask "shown" is where there will be *NO* soldermask.**

Other Layers

- **Paste Mask**
  - **Paste mask is similar to solder mask, except that it is used to create solder paste screens which can then be used to solder SMDs in a hot re-flow soldering process**
- **Multi-Layer**
  - **Multilayer is a simple way of adding an object such as mechanical drawing or a pad to ALL the PCB layers. Any object on this layer will automatically be added to all the PCB layers**
- **Drill Layer – There are two drill layers**
  - **The Drill Guide Layer is used for legacy equipment and is generally not in use today**
  - **The Drill Drawing Layer provides coded plots of board hole locations**

Silkscreen Layer

- The silkscreen layer is also known as Overlay. Top Overlay refers to the silkscreen on top of the board, and Bottom Overlay refers to silkscreen on bottom
- This is the layer onto which the component designators are printed (R1, R2, …) so as to identify individual components during component placement of the board
- They are also used during the PCB routing process to indicate the outlines of your components. This helps you in placing (or not placing) components too close to one another, or too close to the edge of the board.
- Make sure your silkscreen doesn't run over any exposed copper (such as pads)!

Multi-layer design

- Multi-layer PCB boards are the norm for complex designs, in particular those which have size constraints
- The simplest of multi-layer boards is a 4 layer board which provides two internal power planes and two outer signal layers
- More complex multi-layer boards can reach up to 24 layers!
- Some PCB material limit the number of layers, such as flexible PCB boards, but the advantages of using flexboards can

Example of Multiple Layers
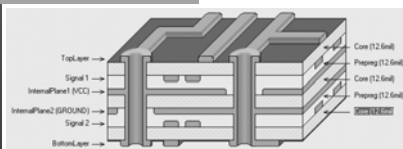
- By using dedicated power and ground planes, you drastically reduce the complexity of your routing, not to mention reducing inductance and impedance across the board
- Prevent ground loops from hapening

Grounding and Bypassing

- Providing good grounding is critical to the functionality of a PCB board
- On a multi layer board, one of the layers should be dedicated ground and all ground signals should have vias into this ground plane.
- Avoid chaining grounds in order to prevent ground loops from occurring
- Use bypass capacitors to smooth out power spikes by components that suddenly draw significant current
- Bypass caps should be placed as close to the component's power pins as possible
- You should use 100 nf caps as the norm, with lower capacitance for higher frequencies, and higher capacitance for lower frequencies

Layer management

- Viewing all layers in a PCB document at once can quickly get out of hand
- In order to aid the designer, CAD tools provide several options for managing and showing active layers
- One technique is to view the PCB in a draft mode. In this mode objects are drawn only with their outline
- Another technique is to view them one layer at a time
- Some CAD tools have a "see through" mode enabling you to view multiple layers at the same time
- Finally you can enable/disable each layer

Units

- PCB Boards are primarily designed in imperial units (inches) as opposed to metric units (mm).
- A thousands of an inch is called mil (not to be confused with mm), where:
  - 100 mils = 0.1 inch = 2.54 mm
- The reason for using imperial units in a PCB document is because most of the components were manufactured according to imperial pin spacing. The practice continues even today!

Pads

- **Pads come in all sorts of shapes and sizes. They can be through-hole or SMD but they all follow the same general guidelines**
- **Through-hole pads are used for leaded components like resistors and capacitors are round, with about a 60 mil inner diameter**
- **Dual in-line Packages (DIP) components like ICs fair better with an oval shaped through-hole pad, with 60 mils in one dimension and 100 mils in another**
- **Surface mount (SMD) components adhere to the chip specs and are generally rectangular**
- **Pin 1 of any chip footprint should always be a different shape: rectangular for DIPs and oval for SMDs**

Vias

- **Vias are special pads which connect electrical signals from one side of your board to another**
- **In special circumstances, from one layer to another without crossing all layers (blind or buried vias)**
- **Blind and buried vias are to be avoided at all cost, they are difficult to debug and rework**
- **Vias are made of conductive material which are called Plated Through-hole.**
- **There is really no difference between Vias and Pads except that the CAD tools manage them differently to allow more complex operations on vias.**
- **Vias are generally much smaller than pads**

Component placement

- **Component placement is an extremely important function of the designer**
- **Components should be placed according to their connections to other components, thermal considerations, mechanical requirements, as well as signal integrity and routability**
- **Components which have connections to each other should be placed in the same vicinity**
- **For example, a processor should be placed very close to the RAM and Flash ICs on which it relies**
- **Components should also be placed on a grid, usually a 100 mil grid, in order to provide for a symmetric flow of routing where tracks and components are lined up**

Auto Router

- CAD tools provide auto-router and board wizard functionality
- In reality, PCB designers don't use an auto-router
- The technology behind an auto-router has a science of its own, drawing from disciplines such as artificial intelligence, heuristic algorithms, and ultimately attempting to solve the traveling salesman problem
- The irony is, only a very experienced PCB designer can take advantage of an auto-router, and even then, it is only used for a fraction of the board

Manufacturing Guidelines

- Internal layers should be designed at least 15" away from the edge of the PCB board. This is to ensure that the manufacturer of the board does not cut into the copper or short the planes. You can employ the keepout layer for this purpose.
- Each manufacturing house has their own manufacturing guidelines for minimum space width, copper-to-edge distance, via and hole sizes, annular ring, etc.
- Many assembly houses require that you provide your PCB boards in a panelized format. Panelizing a design means fitting multiple boards on a standard panel. Once the assembly is done, they are cut into individual PCB boards

Gerber files

- Gerber, otherwise known as RS-274-D, is the industry standard photo plotting language
- It is used by photo plotter equipment which use light to "draw" a line
- It was developed in the 1980s, and has since been modified in many ways
- It is an ASCII format file, which instructs the photo plotter to perform one of four basic functions:
  - Move to location X,Y
  - Select the correct aperture tool
  - Open, close, or flash shutter
  - End of line instruction (the '*' character)

What to send to a manufacturer

- In order for a board manufacturer to build the PCB boards, they need a set of gerber files
- The PCB CAD tool is capable of generating gerber files for every PCB layer
- However, an aperture library and sometimes a drill file must be generated separately
- As discussed, gerber files instruct the equipment where to go and what to do. The aperture list specifies which tool to use. Aperture files come with different resolutions, it is best to send the highest resolution aperture list possible to the manufacturer
- Excellon drill files which may also be sent separately are used to tell the equipment what size holes to drill and where

Starting a new PCB

- First create a Mechanical outline for the board by using one of the Mechanical Layers
- Then create an electrical outline of the board by placing a Keepout Layer
- The Keepout Layer aids the CAD system in automatically placing and routing of the components
- Generally the Keepout Layer and the Mechanical Layer are one and the same

Testing

- Babak's Test of Futility
  - Take a bare PCB board and an ohm-meter, and ohm out the power and ground signals. If they are shorted, the Test of Futility has been completed and you have a useless board! Needless to say, do not plug it in. This can be caused by manufacturing errors, or by corrupt gerber files which were sent to the PCB house