CENG 4480
Lecture 11: PCB

Bei Yu

Reference:
• Chapter 5 of Ground Planes and Layer Stacking
• High speed digital design
• by Johnson and Graham

The Chinese University of Hong Kong
Introduction

• What is a PCB

• Why we need one?
  ✦ For large scale production/repeatable fabrication
  ✦ Reliable: much better than ad hoc bread board
  ✦ Controlled Electrical characteristics

• Many videos showing you how to make one on You Tube
  ✦ https://www.youtube.com/watch?v=e-gMsABCRTI

• Our lecture:
  ✦ Not on how to make one (you will try one later)
  ✦ More concern about issues on reliability; electrical characteristics
How to Make a PCB

1. Cut the board
2. Toner transferring
3. Removing the paper
4. Etching using acid
5. Clean the board
6. Drilling the holes
Modern PCB Design — 1

DDR3 Layout on Layers 2 and 3 [www.mentor.com]
Functions of Ground and Power Planes

• Provide stable reference voltage for digital signal
• Exchange Distribute power
• Control crosstalk between signals

• Note:
  • All formula are approximations
  • In this book, signal trace = tracks on PCB
Current Path

- At low speed: => Follow Least Resistance
- At high speed: Follow Least Inductance

At low speeds, current flows the path of least resistance

At high speeds, current flows the path of least inductance
Return-current density

- A function of H and D:
  - H: height of trace above PCB
  - D: perpendicular distance

- Current density at D is:
  \[
  \frac{I_o}{\pi H} \cdot \frac{1}{1 + \left(\frac{D}{H}\right)^2}
  \]

- Current density at D is proportional to:
  \[
  \frac{1}{1 + \left(\frac{D}{H}\right)^2}
  \]
Crosstalk in Solid Ground Planes

- Magnetic fields => induce voltages in other circuit traces
- Mutual Inductance & Mutual Capacitance
  - inductance effects dominates
- Crosstalk \( \sim \frac{K}{1+(D/H)^2} \)
  - \( K \leq \) rise time & length of trace
  - Faster rise time, higher \( K \)
Simple Crosstalk Experiment

- 26 in Cu track separated by 0.08 in centre to centre
- Ground plane is a solid Cu sheet
Crosstalk in Slotted Ground Planes

- Ground slots increases crosstalk:
  - large loop $\Rightarrow$ higher inductance
  - Overlaps with other signals

- Must not tolerate
Crosstalk on Dense Connection Holes (Vias)

- Slots in ground plane creates unwanted inductance
- Slots inductances slows down rising edges
- Slot inductance creates mutual inductive crosstalk
Crosstalk in Cross-Hatched Ground Plane

- 2-Layer board design
  - (+) Separate power & ground planes
  - (-) At the expense of increased mutual inductance
  - NOT good enough for high speed system

- Need good bypass capacitors
  - Since signals traverse several capacitors
Bypass

• A bypass is a capacitor that shorts AC signals to ground, so that any AC noise presenting on a DC signal is removed, producing a much cleaner DC signal.
### Bypass (cont.)

- **Ideal DC Signal**

  ![DC Voltage Diagram](image)

  - **At low speed:** Follow least resistance

- **Practical DC Signal**

  ![DC Voltage Diagram](image)

  - **At high speed:** Follow least inductance

- **A Capacitor**

  ![Capacitor Diagram](image)
Crosstalk in Cross-Hatched Ground Plane

- 2-Layer board design
  - (+) Separate power & ground planes
  - (-) At the expense of increased mutual inductance
  - NOT good enough for high speed system

- Need good bypass capacitors
  - Since signals traverse several capacitors
Crosstalk with Power & Ground Fingers

- 1 power & ground layer; and 1 signal layer
- (+) Save even more board area
- (-) Worse mutual inductance
  - only for very slow circuit
  - Fatten the ground fingers will not help
Guard Traces

• Appear extensively in Analogue Systems
  ✦ On a two layer board w/o. solid ground plane
  ✦ A pair of guard traces can reduce crosstalk by an order of magnitude

• In general, a trace grounded at both ends will halve coupling

• In general, reduce crosstalk to 1-3% is good enough
**Ex. Guard Trace Calculations**

- **Question:** what’s the estimated crosstalk?

- **Answer:**

\[
\text{since } K < 1, \text{ crosstalk} < \frac{1}{1 + (D/H)^2} = \frac{1}{1 + 8^2} = 0.015
\]
Experiment Measuring Guard Trace Efficiencies

[Diagram showing a circuit with labels A, B, and C, and a Tektronix 11403 oscilloscope trace demonstrating the effect of guard traces on crosstalk.]
More on Guard Traces

- In Digital System
  - Solid ground plane is preferred
  - No extra benefit using guard trace

![Images showing NEXT coefficient comparisons](image.png)
Near-end & Far-end Crosstalk

- Descriptions so far based on lumped-circuit analysis
  - No good for long transmission lines

- Inductive Coupling Mechanism
- Capacitance Coupling Mechanism
Inductive Coupling Mechanism

- Magnetic coupling (mutual inductance) likes a **transformer**

- A series of blips appear on the bottom line:
  - **Negative** forward blips
  - **Positive** reverse blips
Capacitance Coupling Mechanism

- Positive for both forward and reverse blips
Reflection Diagram

The induced pulse from each transformer in Figure 5.15 propagates in both directions.

Far-end pulses all arrive together at time $T_p$.

Inductive coupling has opposite polarities in the forward and reverse directions.

Derivative of input signal (negative)

Total coupled areas are the same

Rise and fall times same as input signal

Capacitive coupling has the same polarities in the forward and reverse directions.

Derivative of input signal (positive)

Total coupled areas are the same

Rise and fall times same as input signal
Combining Mutual Inductive & Capacitive Coupling

- Generally, over a solid ground plane, inductive and capacitive crosstalk voltages are roughly equal

- Over slotted, hatched or imperfect ground plane
  - Inductive component is much larger
  - Forward cross talk is large & negative
Near-end Crosstalk => Far-end Problem

- In practical applications without source terminators
  - Source is a low impedance driver
  - Reverse crosstalk reflects when it hits the near-end
  - Reflection coefficient = -1
  - Signal seen at the far-end is a copy of the reverse coupling signal at C, delayed by one propagation delay and inverted

![Diagram of near-end crosstalk and far-end effects](image)
Summary (for long transmission line)

- Over solid ground, inductive & capacitive crosstalk are qual
  - Forward crosstalk cancel
  - Reverse crosstalk reinforce

- Over a slotted or imperfect ground plane, inductive coupling exceeds capacitive coupling
  - Forward crosstalk large and negative

- Forward crosstalk is like a square pulse
  - constant height & duration 2Tp

- Reverse crosstalk when it hits a low impedance driver, reflects towards the far-end
Modern PCB Design — 2

Escape Routing
[Tan et al. DAC’2009]
Simultaneously Escape Routing [Luo et al. ISPD’2010]
Untangling Twisted Nets + Length-Matched Routing [Tan et al. ICCAD’2007]
One-Side Untangle Twisted Nets

- Route left side pins to the right side pins
  - Each pin has an ID.
  - Planar routing
  - Only untangle the twisted nets on the left-side

Example:

How to automatically handle extreme large cases?
Stacking Circuit Board Layers

• **Need to specify**
  ✦ Which are the power, ground and signal layers
  ✦ Dielectric constant of the substrate
  ✦ Spacing between layers
  ✦ Desire trace dimensions and minimum trace spacing

• **Power & Ground Planning**
  ✦ Choose solid, hatched or finger ground plane model
  ✦ Use ground & power planes in pair
  ✦ Symmetric pairing in a layer stack helps prevent wrapping
  ✦ Both ground & power planes may be used as low-inductance signal return paths
  ✦ Adequate bypass capacitors between ground and power planes
Selecting Trace Dimensions

• Dense design requires fewer layers but

• Smaller, more closely spaced traces also yield more crosstalk and power-handling capacity problem

• Power-handling capacity depends on
  ✦ Cross sectional area
  ✦ Allowable temperature rise (amount of power dissipated)

• Power is not a problem for a large distribution bus
  ✦ is a big problem for extremely small trace

• High density will lower yield, thus increase cost
  ✦ avoid using minimum attainable line width

• Other factors:
  ✷ Control etching process to avoid wide line width variations to control the impedance
Routing Density versus Layer #

- More layer will cost more but easier to lay
- From experience
  - Divide the circuit into quadrants, half of the wires will stay with a quadrant
  - Same statistics when this quadrant is further subdivided into quadrants
  - Average wire length = spacing between quadrants
Thank You