PCB Design Guidelines for Reduced EMI

Guided By:
Prof. Ruchi Gajjar

Prepared By:
Shukla Jay (13MECE17)
Outline

• Power Distribution for Two-Layer Boards
• Gridding Power Traces on Two-Layer Boards
• Ferrite Bead
• Board Zoning
• Signal Traces
• Different Noises
• Common Mode Noise
• Differential Mode Noise
• Shielding
• Conclusion
• References
Power Distribution for Two-Layer Boards

1. Single-Point Distribution
   • Each active component has its own separate power and ground
   • Remain separate until they meet at a single reference point
Power Distribution for Two-Layer Boards

2. Multi-Point Distribution

- Connections in daisy-chain fashion with 0-V reference points
- Potential for common impedance coupling
Power Distribution for Two-Layer Boards

3. Star Distribution

• Much like single point
• All points reference the same fixed point
• Which is located centrally by about the same length of traces
• The point where the separate traces begin is near the center of the board, and each trace goes in its own direction, with the resulting trace length equal to that of all the others
Power Distribution for Two-Layer Boards

- **Trace Type**
  - **Power**
    - Single Point: Best
    - Star: O.K.
    - Multipoint: Worst
  - **Signal**
    - Single Point: O.K.
    - Star: Best
    - Multipoint: Worst

- **Source “Star”**
Gridding Power Traces on Two-Layer Boards

• Gridding: The network of orthogonal connections between traces carrying ground
• Same noise reduction as on four-layer boards
• Emulates the ground plane of a four-layer board by providing a ground return path under each of the signal traces
• Lowers the impedance between the microcomputer and the voltage regulation
• Gridding is done by expanding any ground traces and using ground-fill patterns to create a network of connections to ground across the PCB
Gridding Power Traces on Two-Layer Boards

- PCB has most of the topside traces running vertically and most of the bottom traces running horizontally
Ferrite Bead

- A passive electric component that suppresses high frequency noise in electronic circuits

Fig. 1. Ferrite-Bead Placement Closest to the Noise Source
Board Zoning

• Basic meaning as board floor planning, which is the process of defining the general location of components on the blank PCB before drawing in any traces
• The process of placing like functions on a board in the same general area, as opposed to mixing them together
• High speed logic nearer to power supply
• Less chances to pollute other signal traces
• Power supply-high speed logic-slower components-analog components
Chassis harness, microcomputer regulator are all in the same area. Wide ground fields interconnect all three, forming single-point ground reference.
1. Capacitive and Inductive Crosstalk:-
   • Keeping RF-noise-carrying traces that are connected to the microcomputer away from other signals so they do not pick up noise
   • Signals that may become victims of noise should have their return ground run underneath them, which serves to reduce their impedance, thus reducing the noise voltage and any radiating area
   • Never run noisy traces on the outside edge of the board
   • If possible, group a number of noisy traces together surrounded by ground traces
Signal Traces

• Keep non-noisy traces away from areas on the board where they could pick up noise, such as connectors, oscillator circuits, relays, and relay drivers

2. Antenna Factor Length Rules:
   • For Federal Communication Commission (FCC) limits, trace length becomes important when it is greater than 1/10 of the wavelength

3. Series termination, Transmission lines:
   • To provide critical damping to achieve the highest possible data transmission rates with the least-possible overshoot
Different Noises

1. Common Mode Noise
2. Differential Mode Noise
Common Mode Noise

• Common-mode noise is a big problem in cables, but the fault does not lie in the cable, it lies in the connections on the board that the signals and returns tie to that form the common impedance.

• Common-mode noise is corrected either at the source, by reducing the impedance of the common node, or reduced by placing a ferrite bead around the entire cable.
Differential Mode Noise

- Differential-mode noise (the useful noise of an edge transition) should first be reduced to the maximum (slowest) allowable rise and fall times and should occur at only the minimum needed frequency.
- The noise radiated is due to the loop of the signal and its return.
- This loop is minimized by having as many returns as possible and by twisting each signal and return pair.
- The latter causes field cancellation at some distance away, in the same manner as routing power over ground.
Fig. 2. Mounting Filter Capacitors for External I/Os

- **GOOD**
  - Large ground on both sides reduces impedance, a good starting point for the 0-V reference point for the power supply.

- **BAD**
  - Narrow traces increase impedance, which reduces filtering effectiveness.
Conclusion

• Selection of semiconductor components that produce low electromagnetic radiation
• Large loops of signal and corresponding ground-return lines that carry high frequencies must be avoided
• Filtering of critical lines, such as the supply line, ensures that high-frequency currents do not leave the PCB
• Eliminates antennas that can radiate electromagnetic energy
• Shielding of the total system is not required if we follow this rules
References
