Introduction to Signal Integrity

Prof. Daniel Costa Araújo

March 22, 2023

University of Brasília

Outline

Introduction

Signal Integirty, Power Integrity, Eletromagnetic Compatibility

Why study?
In the past, when 10-MHz clock frequencies were common, interconnects were not a major concern for system performance. However, as clock frequencies increased and signal rise times decreased, signal integrity (SI), power integrity (PI), and electromagnetic compatibility (EMC) became important factors in the design of electronic products.

Definition

- Signal integrity (SI) deals with signal distortion;
- power integrity (PI) addresses noise on interconnects and power delivery components;
- electromagnetic compatibility (EMC) involves the product's radiated emissions and susceptibility to external electromagnetic interference.

Main problems in the Field

- Signal distortion on one net
- Rise-time degradation from frequency-dependent losses in interconnects
- Cross talk between two or more nets
- Ground and power bounce as a special case of cross talk
- Rail collapse in power and ground distribution
- Electromagnetic interference and radiation from the entire system

NOTE
Understanding the root cause of these six families of problems and the essential principles behind them helps to find and fix issues in each family, ensuring successful product design.

Signal distortion in net

Understanding a Single Net

- A single net is a group of interconnected metal traces or conductors in an electronic system.
- Electrically connects components (e.g., output pins, input pins) on a printed circuit board (PCB) or within an electronic system.
- Comprises both the signal path and the return path for the signal current.

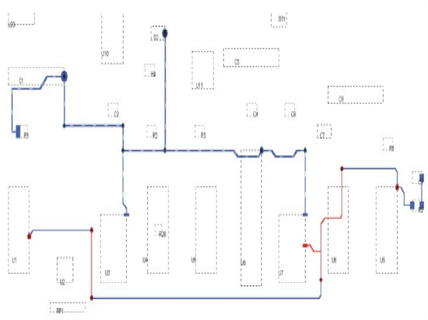

Figure 1:

Understanding Signal Propagation and Impedance Discontinuities

1. Signal leaves output driver as voltage and current.
2. Signal encounters interconnect, perceived as electrical impedance.
3. Signal continuously assesses instantaneous impedance.
4. Constant impedance ensures undistorted signal propagation.
5. Impedance changes cause signal reflection and distortion.
6. Distortions may lead to false triggering.
7. Discontinuities: features that change the net's cross-section or geometric shape.
8. Discontinuities distort the signal from its original form.

Impedance Discontinuities and Signal Integrity

- Features changing impedance include:
  - End of interconnect
  - Line-width change
  - Layer change through a via
  - Gap in return-path plane
  - Connector
  - Routing topology change (branch, tee, stub)
- Discontinuities arise from cross-section changes, routing topology, or added components.
- Common discontinuity: end of a trace (high or low impedance).
- Solution: Engineer the interconnect impedance to be constant.

Best Practices

1. Use a board with constant, or “controlled,” impedance traces. This usually means using uniform transmission lines.
2. To manage the reflections from the ends, use a termination strategy that controls the reflections by using a resistor to fool the signal into not seeing an impedance change.
3. Use routing rules that allow the topology to maintain a constant impedance down the trace. This generally means using point-to-point routing or minimum-length branch or short stub lengths.
4. Engineer the structures that are not uniform transmission lines to reduce their discontinuity. This means adjusting fine geometrical design features to sculpt the fringe fields.

Cross-talk

Cross Talk in Different Environments

Cross talk occurs in two different environments:

- When the interconnects are uniform transmission lines, as in most traces in a circuit board.
- When they are not uniform transmission lines, as in connectors and packages.

In controlled-impedance transmission lines where the traces have a wide uniform return path, the relative amount of capacitive coupling and inductive coupling is comparable. In this case, these two effects combine in different ways at the near end of the quiet line and at the far end of the quiet line.

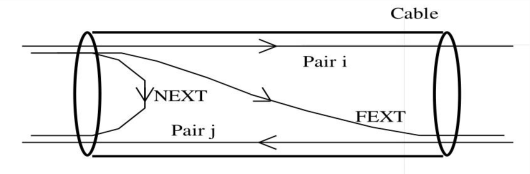

Figure 2: Measured near- and far-end cross talk between two nets in a circuit board.

Cross Talk and Coupled Noise

- Lowest cross talk configuration: wide uniform plane as return path.
- Deviation from this configuration increases coupled noise.
- Inductively coupled noise dominates, causing switching noise, delta I noise, dI-dt noise, ground bounce, SSN, and SSO noise.
- Generated by mutual inductance.
- Occurs mainly in connectors, packages, and vias where return path conductor isn't a wide, uniform plane.
- Ground bounce: special case of cross talk with overlapping return currents in the same conductor, leading to high mutual inductance.

Rail collapse noise (Power supply-induced noise)

Definition

When happens?

- occurs when the voltage rails in an integrated circuit (IC) experience voltage fluctuations due to the simultaneous switching of many digital components within the circuit.
- voltage fluctuations can cause errors, affect performance, or even lead to complete system failure.

How to mitigate

1. Decoupling Capacitors: Place decoupling capacitors close to the power supply pins of the IC. These capacitors act as local energy reservoirs that can quickly supply the additional current required during transient events, helping to stabilize the voltage rail. Decoupling capacitors should be chosen based on their capacitance value, equivalent series resistance (ESR), and equivalent series inductance (ESL) to ensure optimal performance in the target frequency range.
2. Voltage regulator selection: Choose voltage regulators with low output impedance and fast transient response, which can better handle sudden changes in load current and minimize voltage fluctuations. Switching regulators like voltage regulator modules (VRMs) are generally more efficient and have better transient response compared to linear regulators like low-dropout regulators (LDOs).
3. Grounding: Use a solid ground plane to provide a low-impedance return path for currents, which helps reduce noise coupling. Connect all ground points to the ground plane using short, direct paths. For mixed-signal designs, use a partitioned ground plane to separate analog and digital grounds, connecting them at a single point to avoid ground loops