Understanding High-speed PCB Design – High-speed, RF and EMI

Part 1: Essential High-speed PCB Design for Signal Integrity

Contents

Module 1 - High-speed design overview
- Design issues for the engineer and for the PCB layout designer
- When is a design “high speed”?
- Industry drivers force high speed
- Signal integrity and the high speed challenge
- Why we need to consider wave propagation and wave properties
- The PCB contribution
- High speed PCB design – key requirements

Module 2 - Fundamental electrical concepts
- Time domain and frequency domain
- Signal bandwidth - analog signals and digital signals
- Digital waveforms
- Clock speed versus edge speed - effect of signal risetime
- Effective operating frequency and knee frequency
- Current, voltage and resistance
- Electric fields, capacitance and dielectric constant
- Magnetic fields and inductance - self inductance on PCBs
- Effect of circuit components on signal waveform - transmission lines
- Current paths on a PCB
- Attenuation of signals on lines - skin effect and loss tangent

Module 3 - Power delivery
- Power requirements
- Coping with changing currents - induced noise
- Board level and component level decoupling
- Practical limitations - bandwidth of capacitors
- Three problems with the traditional approach to decoupling
- Expected versus actual response of decoupling networks
- The alternative approach to power delivery
- Flattening the impedance response
- Supplying charge - component current risetimes
- Power - ground plane resonance
- Summary - two approaches to power delivery

Module 4 - PCB transmission lines
- Transmission line velocity and delay
- Characteristic impedance
- Material and stackup effects
- Geometry and fabrication effects
- Propagation of a voltage step
- Transmission line input impedance
- Reflection from a terminated line - different cases
- Impedance control by line termination
- Series and parallel termination
Module 5 - differential transmission
- Why use differential transmission? (1)
- Differential signalling
- Effects of equal and unequal transmission line lengths
- Differential and common mode currents
- Routing differential tracks close together
- Coupled lines - current, voltage and impedance (odd and even mode)
- Rules for routing differential transmission lines
- Line terminations
- Do we need to terminate for even mode?
- Why use differential transmission? (2)

Module 6 – Crosstalk
- Capacitive and inductive crosstalk
- Dependence on edge rate
- Coupling factors - solid ground plane
- Coupled lines and coupling mechanisms - forward and backward crosstalk
- Where do the coupled signals go?
- Near end and far end crosstalk
- Effect of coupled length
- Other coupling and ground plane effects
- Crosstalk from multiple lines
- Crosstalk induced jitter
- Crosstalk control in PCB design – parts, planes, tracks, connectors, terminations

Module 7 – Modelling drivers and receivers
- IC device characteristics - drivers and loads, bipolar and CMOS
- Simple equivalent circuits and models - device output
- Real devices – modelling input, output and I/O ports
- Behavioural device model
- IBIS - I/O Buffer Information Specification – content and file structure
- Measuring and extracting I-V curves (in principle and in practice)
- Transient characteristics - transition timing
- IBIS standards - evolution and key points

Module 8 - PCB routing topologies
- Transmission line types, nets and buses
- Track routing effects - capacitive and inductive discontinuities
- Discontinuity effects from corners, connectors, vias and microvias
- Serpentine tracks (delay equalisation)
- Incident and reflected mode switching
- Overshoot and ringing
- Topology types - branching and non-branching, stubs, routing constraints
- Multiple capacitance loading
- Clock distribution
- General principles for routing

Module 9 - PCB structure, manufacture and measurement
- Layer stacking effects and principles – power, ground and routing layers
- Effects of PCB fabrication process variables on high-speed designs
- The influence of key PCB materials parameters
- Measurement of transmission line impedance
- TDR testing of PCB track impedance
Understanding High-speed PCB Design – High-speed, RF and EMI

Part 2: High-speed PCB Design for Gigabit Data Rates and EMI Control

Contents

Module 1 - What is “high-speed”? - Part II
- Trends in design and technology
- How do we measure and test?
- Time domain
  - oscilloscope (measure signals)
  - TDR/TDT (measure components/system path)
- Frequency domain
  - spectrum analyser (measure signals)
  - VNA (measure components/system path)
- What do we measure (components/system path) ?
  - S-parameters!
  - Comparison of TDR and VNA measurements

Module 2 - Gb/s transmission on PCBs
- Lessons on signal quality from telecommunications digital transmission
  - digital traffic multiplexing, coding and frequency translation
  - transmission line coder and decoder
  - bit error rate measurement
  - eye diagrams
  - signal skew and data jitter
  - inter symbol interference
- Serialiser/Deserialiser (SerDes) technology
- Gb/s technologies
  - Low Voltage Differential Signalling (LVDS)
  - Current Mode Logic (CML)
- PCI Express
  - an example of a standard (not a technology)

Module 3 - PCB materials for high-speed design
- Material requirements for high-speed PCBs
  - Factors: dielectric constant, dielectric loss, conductor loss, surface roughness
  - Transmission line attenuation due to dielectric and conductor loss
  - Interconnect bandwidth limitation due to line loss
    - effect on signal quality
  - PCB materials for lower loss
    - enhanced epoxy compared to FR-4
    - high performance materials for > 5 GHz bandwidth
- Embedded capacitors and resistors

Module 4 - EMC Control
- EMC concerns
- Why EMC has become a major issue
- Definitions
- EMI mechanism, coupling paths and methods
- The five factors in EMI analysis
- What we can control in digital systems
- EMC guidelines
- Regulatory requirements
Module 5 - Principles of EMI generation
- Electromagnetic wave propagation
- Near field and far field
- Time varying currents and voltages - radiation generation
- RF fields generated on a PCB
- Differential mode and common mode currents and radiation

Module 6 – PCB structure
- Power and ground planes - layer stacking effects
- 20H rule
- Image planes
- Grounding concepts and methods to reduce common mode current loops
- Electrical lengths -\(\lambda/20\) rule
- System partitioning - split planes
- Isolation and bridging techniques

Module 7 – EMC from components to systems
- IC package parasitics
  - ground bounce, mutual capacitance coupling
- EMI from large heatsinks
- Localised ground planes
- Impact of IC technology drivers on EMC control at component level
- I/O connections to/from PCB modules
- Backplanes and plug-in boards
  - RF coupling - PCB to PCB and PCB to chassis
  - Indirect multipoint grounding
  - Backplane connectors and signal routing
- ESD protection
- Summary - designing PCBs for EMC