FlowCAD

TRAINING

Bei dem hier beschriebenen Training handelt es sich um ein Cadence Standard Training. Sie erhalten eine Dokumentation in englischer Sprache. Die Trainingssprache ist deutsch, falls nicht anders angekündigt.

Course Title	**Understanding High Frequency PCB Design – High-Speed, RF, and EMI -v3.0**
Course Category	**System Interconnect Design – Allegro & OrCAD**
Duration	**5 Days**
Course ID	**ES_86072_3_0**
Product Version	**16.2**

Course Aims

Part 1 of this two-part course applies basic physical principles to develop an understanding of the key issues of high-speed design, to ensure a successful design for signal integrity.
These range from controlling reflections and crosstalk to the design of the power distribution system and the PCB layer structure.
The course is liberally illustrated with examples and "what if" scenarios showing by simulation the effects of varying different parameters, enabling participants to develop an understanding of their relative importance and magnitude.
Helpful guidelines on assessing and implementing best practice are included.
Practical issues are considered throughout.
The basic techniques developed can be applied immediately to improve PCB design, *without* the use of EDA signal integrity tools, but the course also provides a much needed foundation for understanding how to benefit from the use of such tools.

Part 2 of this two-part course builds seamlessly on the principles and practice established in Part 1, extending them to develop techniques for design and test at frequencies above 1 GHz for Gb/s serial transmission and for controlling the generation and propagation of EMI at the PCB level.
Key topics cover signal quality, material effects and EMC from components to backplanes.

Note: This is an integrated course where the concepts and methods developed in Part 1 are applied directly to the topics in part 2.
Delegates to Part 2 of the course are therefore strongly advised to attend Part 1 first.

Learning Objectives

Part 1: Essential high-speed PCB design for signal integrity (3 days)

- **Signal waveforms, frequency components and risetime.** Bandwidths of analog and digital signals. How capacitance and loop inductance on a PCB determine signal behaviour. Current paths on a PCB.
- **Impedance control of the power distribution system.** Controlling induced noise - decoupling networks, PCB planes, bandwidth requirements. Optimizing power delivery.
- **Track impedance, reflections, and line terminations**. Effects of PCB structure, materials, geometry and fabrication. Track impedance testing.
- **Coupled lines.** Odd and even modes – differential and common mode currents. Differential transmission, routing and termination. Unwanted coupling – crosstalk. Near end and far end crosstalk, effects of coupled length, multiple lines.
- **ICs for high-speed design** - I/O characteristics, I/V curves, transition timing. Behavioural device models, IBIS standards.

- **PCB routing topologies.** Branching and non-branching topologies. Constraints. Discontinuity effects – connectors, vias, stubs etc. Equalisation, multiple capacitance loading, clock distribution.

Part 2: PCB design at RF - multi-Gigabit transmission, EMI control, and PCB materials (2 days)

- **High frequency measurement and test** – components and signal paths. Time domain (scope, TDR/TDT) and frequency domain (VNA, spectrum analyser). Probe bandwidth. S-parameters.
- **Gb/s transmission on PCBs** - application of transmission engineering methods. PCB track effects on signal quality (BER, ISI, jitter). Technologies (e.g LVDS, PCI Express). PCB requirements to meet system performance.
- **Frequency-dependent PCB transmission lines.** Waveform degradation due to conductor and dielectric loss. PCB material selection – frequency behaviour, manufacturing and cost tradeoffs, and criteria for acceptable signal performance.
- **EMC control.** EMI mechanisms – what factors can we control? Wave propagation, near and far field impedance. RF field generation on a PCB. Differential to common mode conversion and radiation.
- **Controlling EMI generation on PCBs.** Image planes, stackup, return currents. Grounding schemes, common impedance coupling, partitioning, split planes.
- **EMI from components to systems.** IC package parasitics, ground bounce, component level effects. Filtering, isolation and bridging on PCBs. Interconnections, cables, backplanes, signal routing.

Audience

Design engineers seeking in-depth knowledge of high-speed PCB design, signal integrity issues, high frequency effects and EMC. As the course is built up from basic electrical principles it is suitable for engineers from many areas of application, and also for new graduates.

PCB designers working on digital or mixed signal boards with design rules governing track impedance control, line terminations, routing to minimise noise coupling etc. will also benefit from this course.

Course Agenda

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

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 signaling
- 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

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

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

FlowCAD

    - 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 -λ/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

Related Courses

- Allegro PCB SI Foundations v16.3
- Allegro PCB SI GXL-v16.2

- Allegro PCB SI EMControl v16.2
- Allegro PCB Power Integrity v16.2