MIXED SIGNAL SYSTEMS

SIGNAL INTEGRITY FUNDAMENTALS

COURSE OVERVIEW

Understanding Signal Integrity is a key skill for every hardware and PCB layout engineer. This 3-day course provides engineers with the Signal Integrity knowledge and skills they need to design modern electronic systems.

The skills learned in this course allow engineers to design both digital and mixed-signal PCBs with confidence. Engineers employing these skills may benefit from fewer board revisions, improved system performance, fewer EMC issues and reduced time to market.

WHO SHOULD ATTEND

Hardware Engineers.
PCB Layout Engineers.
Engineers that participate in PCB Layout Reviews.
Engineers wishing to learn Signal Integrity design techniques.
Engineers wishing to resolve Signal Integrity and EMC issues.

WHAT YOU WILL LEARN

Principles of Signal Integrity.
Principles of Power Integrity.
Practical Design Techniques.

PREREQUISITES

A working knowledge of electronic hardware design is required. Knowledge of electromagnetic theory would be beneficial but is not necessary.

COURSE DURATION

Three Days

COURSE DATES AND LOCATIONS

Please contact us for upcoming course dates and locations. On-site, team-based training is available on request.

PRICE

Please contact us for pricing information. Discounts are available for early-bird registrations and team bookings.

ABOUT MIXED SIGNAL SYSTEMS LIMITED

Mixed Signal Systems provides design and technical consultancy services to a global client base in the technology, electronics and semiconductor industries.

COURSE OUTLINE

Introduction

What is Signal Integrity? • What is Power Integrity? • Classification of Signal Integrity Problems • Signal Integrity and EMC/EMI

Properties of Electrical Systems

Voltage • Current • Resistance • Capacitance • Inductance • Impedance • Path of Least Impedance • Bandwidth • Lumped versus Distributed Systems • Spectral Content of Digital Signals • Bandwidth and Rise Time • Knee Frequency

PCB Fabrication

Elements of a PCB • Copper • Laminate • Prepreg • Vias • Design Rules

PCB Stack-Up

Definition • Design Principles • Signal Return Paths • Classic PCB Stack-Ups

Transmission Lines

Ideal Transmission Lines • Propagation Delay • Characteristic Impedance • Popular Transmission Line Configurations • Lossy Transmission Lines • Performance Regions

Reflections

Consequences of Reflections • Effects of Source and Load Impedance • Reflections at Impedance Changes • Controlling Reflections

Termination Techniques

When to Terminate • End Termination • Source Termination • Termination Strategies • Rise Time Implications • Power Dissipation

Crosstalk

Coupling Mechanisms • Common Path Noise • Inductive Coupling • Capacitive Coupling • Reference Plane Splits • Near-End Crosstalk • Far-End Crosstalk • Guard Traces • Connector Crosstalk

Integrated Circuits

Package Types • Lead Inductance • Ground Bounce • Synchronous Switching Noise (SSN) • Input Buffers • Output Drivers • On-Die Termination (ODT) • IBIS Simulations

Differential Signalling

Differential Pairs • Advantages • Return Current Distribution • Differential Impedance • Common Impedance • Termination Techniques

Power Distribution Networks (PDNs)

Power Distribution • Voltage Reference Distribution • Frequencies of Interest • Target PDN Impedance • Multi-Layered PDN Model • Planes • Bypass Capacitors • PDN Design Methodology

Bypass Capacitors

Capacitor Types • Ceramic Capacitors • Electrolytic Capacitors • Planar Capacitance • Equivalent Series Resistance • Equivalent Series Inductance • Placement • PCB Layout

Grounding Strategies

What is “Ground”? • Ground Plane Topologies • Single versus Split Ground Planes • Ground Loops

Mixed Signal Systems Limited
4th Floor, 115 George Street, Edinburgh, EH2 4JN, UK.

www.mixedsignalsystems.com
enquiries@mixedsignalsystems.com