SILENT SOLUTIONS
EMC TRAINING

Unique courses on recognizing, solving, and avoiding the toughest EMI problems, from Silent Solutions LLC

10 Northern Blvd, Suite 1, Amherst, NH 03031-2328 USA

T: +1 (603) 578-1842 www.silent-solutions.com

Printed Circuit Board Design for EMC

This two-day class provides a unique blend of theory, applications, and numerous hardware demonstrations to describe effective and sometimes not-so-effective PCB design advice that we often hear from electrical engineers and EMC "experts." The real-time demonstrations use a spectrum analyzer, oscilloscope and signal generators to illustrate inductance, common-impedance coupling, and ground loops in PCBs, cables, and systems. Specific examples of single-point, multi-point, "good", and "bad" grounds will be discussed. Examples of integrated circuit application notes that give bad EMC design advice will also be studied during the class.

After Attending This Class, You Will Be Able To:

- Place decoupling capacitors to obtain best performance for a given stackup, based on the latest university research
- Explain the pros and cons of different PCB stackups, and know where to route and not to route high frequency noise sources
- Understand and explain the noise problems shown in the many in-class live demonstrations of functioning PC boards
- Control trace inductance for signal integrity and low noise design
- Locate connectors to reduce emissions and improve immunity
- Explain the problems that split ground planes cause and how to use them correctly
- Identify mutual inductance and improve the effectiveness of filter capacitors
- Choose connectors and assign signals for lowest crosstalk, best signal integrity, and lowest EMI
- Place decoupling capacitors to obtain best performance for a given stackup, based on the latest university research
- Identify good and bad design practices when viewing actual PCB layout screenshots

SILENT

Solutions for your noisy world.

SILENT SOLUTIONS
EMC TRAINING

Unique courses on recognizing, solving, and avoiding the toughest EMI problems, from Silent Solutions LLC

10 Northern Blvd, Suite 1, Amherst, NH 03031-2328 USA

T: +1 (603) 578-1842 www.silent-solutions.com

Printed Circuit Board Design for EMC

Day 1: Attend Day 1 of Electronic Product Design & Retrofit

Section 1: Measuring and Inducing Noise

1. Electromagnetic Compatibility
2. Radiated emissions & associated measurements + DEMONSTRATION
3. Uncertainty in measurements. Underlying problems in predicting results
4. Conducted emissions—electrical schematic and the purpose of LISNs
5. Function and purpose of immunity tests with simplified schematics

Section 2: Predicting and Solving Noise Problems

1. Capacitance—in ESD, PD boards, decoupling networks, filter networks, cables + DEMONSTRATION
2. Inductance—in PC boards, connectors, ICs, high speed signal paths, decoupling networks, filter networks
3. Behavior of current paths at low and high frequencies + DEMONSTRATION
4. Develop a customized source/victim/coupling-factor list of your company's designs
5. Improving your skills – additional topics

SILENT

Solutions for your noisy world.

SILENT SOLUTIONS
EMC TRAINING

Unique courses on recognizing, solving, and avoiding the toughest EMI problems, from Silent Solutions LLC

10 Northern Blvd, Suite 1, Amherst, NH 03031-2328 USA

T: +1 (603) 578-1842 www.silent-solutions.com

Printed Circuit Board Design for EMC

Day 2

Section 3: PCB Noise Models

1. Review of the noise coupling model
2. Review of the four noise coupling paths
3. Emissions and immunity

Section 4: Capacitance, Inductance and Current Paths in PC Boards

1. Good and bad capacitance
2. Good and bad inductance
3. Current loops
4. Low versus high frequency current paths
5. Inductance and low versus high frequency current paths + DEMONSTRATION
6. "Ground plane" splits - appropriate and inappropriate uses

Section 5: Signals on PC Boards

1. Which signals are important?
2. What do they look like? + DEMONSTRATION
3. Transmission lines, characteristic impedance, terminations + SIMULATIONS
4. Harmonic content versus duty cycle + DEMONSTRATION

Section 6: Power Distribution

1. Functions of PCB "grounds"
2. Vcc noise
3. Decoupling and filtering
4. Board layer stack-ups
5. Funny design ideas, current research, new design applications

Section 7: General Design Techniques and Examples

1. Component placement
2. Signal routing
3. Bad application notes
4. Connectors, cables, and I/O wires connected to the PCB

SILENT

Solutions for your noisy world.