Preface

*Printed Circuit Board Design Techniques for EMC Compliance: A Handbook for Designers, Second Edition*, is a significant enhancement to the first edition. The first edition was well received within the engineering community worldwide and was translated into international languages. The intent of the present volume is to expand upon concepts presented in the earlier edition, to justify why a specific design technique works, and to show when it is appropriate for use. Additional techniques based on technological changes within the last few years are also incorporated. These techniques and enhancements are based on questions, comments, and discussions received from engineers around the world.

This book presents information never before published within the engineering community, dealing exclusively with printed circuit boards (PCBs). When writing the first edition, it was impossible to anticipate the amount of variations possible, or what the intended audience expected from a book directed toward nondegree engineers. A thorough understanding of the concepts presented herein will assist during the design and layout process. Note the key word here —“concepts.”

*Printed Circuit Board Design Techniques for EMC Compliance* will help minimize the emission or reception of unwanted radio frequency (RF) energy generated by components and circuits, thus achieving acceptable levels of electromagnetic compatibility (EMC) for electrical equipment. The field of EMC consists of two distinct areas:

1. **Emissions**: Propagation of electromagnetic interference (EMI) from noncompliant devices (culprits) and, in particular, radiated and conducted electromagnetic interference.

2. **Susceptibility or immunity**: The detrimental effects on susceptible devices (victims) in forms that include EMI, electrostatic discharge (ESD), and electrical overstress (EOS).

The primary goal of the engineer is to ensure proper operation and performance of a product when used within an intended electromagnetic environment. These design requirements are in addition to making a product function as desired for use within a specific, end-use environment.

Information presented in this guideline is intended for those who design and layout printed circuit boards. EMC and compliance engineers will also find the information presented herein helpful in solving design problems at both the PCB and system level. This book can be used as a reference document for any design project.
The focus of this book is strictly on the PCB. Containment techniques (shielding), internal and external cabling, power supply design, and other system-level subassemblies that use PCBs as a subcomponent will not be discussed. Again, as in the first edition, excellent reference material on these aspects of EMC system-level engineering is listed in the References section at the ends of chapters.

Circuit technology is advancing at a rapid rate. Design techniques that worked several years ago are no longer effective in today’s products with high-speed digital circuits. As such, there is a need for introductory material that presents fundamental concepts in an easy to understand format. With this in mind, Printed Circuit Board Design Techniques for EMC Compliance was written for engineers who never studied applied electromagnetics in school, or who have limited hands-on experience with high-speed, high-technology PCBs related to EMC compliance.

While it is impossible to anticipate every application or design concern possible, this book provides details on how to implement a variety of design techniques for most products, application dependent. The concepts presented are fundamental in nature. Although every design is different, the fundamentals of product design rarely change because electromagnetic theory is always constant. If fundamental concepts of EMC suppression at the PCB level are learned, implementation becomes a simple issue of practice, though every product will be different from previous designs.

This book presents a minimal amount of mathematical analysis and instead concentrates on hands-on techniques that have been successfully applied to many real-world products. Information is presented in a format that is easy to understand and implement. Those interested in Maxwell’s equations, or in the more highly technical aspects of circuit theory related to PCB design should consult my companion book, EMC and the Printed Circuit Board: Design, Theory, and Layout Made Simple (IEEE Press, 1999). The Reference sections and the bibliography also provide listings of publications that discuss EMC theory using rigorous mathematical analysis. These references are provided because a discussion of technical material is beyond the scope of this book.

The companion book, EMC and the Printed Circuit Board: Design, Theory, and Layout Made Simple explains in engineering terms how and why EMI is developed within a PCB. Although these designers may not be directly accountable for the actual PCB layout, they may be responsible for the end product. Engineers generally want to understand technical concepts. This book elucidates a subject that is generally not taught in universities or other educational environments, again using a minimal amount of math.

The main differences between my first and second books are as follows:

- **Printed Circuit Board Design Techniques for EMC Compliance.** Provides information for those who have to get a product designed and shipped within a reasonable time frame and within budget. It illustrates that a PCB may exhibit an EMI problem, it briefly explains why the problem occurs, and it shows how to solve the design flaw during layout. For the PCB designer, a show-me-how-to-do-it approach exists, with minimal mathematical analysis.

- **EMC and the Printed Circuit Board: Design, Theory, and Layout Made Simple.** This is a companion book for designers who want to know how and why EMI is developed within a PCB. Although these designers may not be directly accountable for the actual PCB layout, they may be responsible for the end product. Engineers generally want to understand technical concepts. This book elucidates a subject that is generally not taught in universities or other educational environments, again using a minimal amount of math.

Controlling emissions has become a necessity in designing electronic devices for both the civilian and military environment. It is more cost-effective to design a product with suppression designed into the PCB than to “build a better box.” Containment measures are not
always economically justified and may degrade as the EMC life cycle of the product is extended beyond the original design specification. For example, users usually remove covers from enclosures for ease of access during repair or upgrade. In many cases, sheet metal covers, particularly internal subassembly covers that act as partition shields are never replaced. The same is true for blank metal panels or faceplates on the front or rear of a system that contains a chassis or backplane assembly. Consequently, containment measures are compromised, and electromagnetic compatibility with the end-use environment is affected. Proper layout of a PCB, with suppression techniques implemented, assists EMC compliance at the level of cables and interconnects, whereas box shielding (containment) does not.

Why worry about EMC compliance? After all, isn’t speed the most important design parameter as mandated by a marketing specification? Internationally, legal requirements dictate the maximum permissible interference potential of digital products. These requirements are based on experience in the marketplace and are related to emission and immunity complaints. Often, these same techniques will help improve signal quality and signal-to-noise ratio performance.

This text discusses both high- and low-speed designs that require new and expanded layout techniques for EMC suppression at the PCB level. Most techniques used several years ago are now less effective for proper signal functionality and compliance. Components have become faster and more complex. Use of custom gate array logic, application-specific integrated circuits (ASICs), ball grid arrays (BGAs), multichip modules (MCMs), flip chip technology, and other digital devices operating in the sub-nanosecond range present new and challenging opportunities for EMC engineers. Technology is progressing at an incredible rate. It is becoming impossible for an engineer to keep up with the technical details and aspects of various logic devices available for use. Even if an engineer learns everything about a new type of logic family, implementation of these devices on the PCB may be overshadowed by other concerns. These concerns include Input/Output (I/O) interconnects, mixed logic families, different voltage levels, analog and digital components, and packaging requirements. The design and layout of a PCB for EMI suppression at the source must always be optimized, while maintaining systemwide functionality. This is a job for both the electrical design engineer and the PCB designer.

In order to design and create a PCB, use of simulation software is becoming mandatory during the development cycle. Simulation software will not be discussed herein, because the requirements for performance, features, and integration between platforms and vendors frequently change. Where appropriate, simulation tools will be mentioned.

In an effort to keep costs down, design for manufacturing (DFM) concerns must also be addressed. In addition, test points must be provided for all nets. For very sophisticated PCBs, DFM and test points may have to give way to functional requirements. If the PCB designer is not knowledgeable or aware of other facets of PCB layout during the design stage, besides placing components and routing traces, serious functionality concerns will develop. In addition, the PCB will usually fail signal integrity and EMC tests, requiring a forced re-layout of the board.

Significant changes have been made in the second edition of this book, including:

- Restructuring of chapters by moving material elsewhere for a better flow of information, placing related topics together instead of being scattered throughout the book.
- Addition of subject headers to various sections within the text for ease of identification of the topic being discussed. In various places, two items that were blended together in the first edition are now separated to facilitate understanding.
- Inclusion of new and advanced layout techniques based on technology and knowledge that became available after the release of the first edition.
• Major overhaul of Chapter 2 (Printed Circuit Board Basics) and Chapter 3 (Bypassing and Decoupling). These are two of the more important chapters in this book. An expansion on the fundamentals of PCB design is presented for engineers who want a thorough understanding, in simplified form, of why layout techniques work the way they do. Because engineers generally want to understand how and why things work, a primer on PCB theory is now incorporated. New concepts related to bypassing and decoupling techniques mandated a complete restructuring of Chapter 3.

• Clarification on when a particular technique is appropriate and not appropriate for use. Justification on how various techniques work is now provided.

• Greatly enhanced information related to single- and double-sided stackup assemblies.

• Restructuring and extensive enhancement of Chapter 4, focusing on all aspects of transmission line (traces) within the PCB in one complete chapter, without having to jump to different areas for a sequential flow of information.

• Clarification and expansion on I/O and interconnects (Chapter 5).

• Significant embellishment of Chapter 6, with sections on additional prevention, layout, and control techniques.

• Restructuring of Chapter 7, including new chapter headings to describe contents as appropriate. Material presented in the first edition was grouped together under one subheading, and new layout techniques were added.

• Chapter 8 is significantly different, as unique layout techniques are now grouped into one chapter. Also, technical details on right angle corners and the 20-H rule are presented. This information has never been published in any written form worldwide, until now.

Information in this edition will assist in PCB design and layout, with the intent of meeting North American and international EMC compliance requirements. Many different layout design methodologies exist. This book illustrates generally applicable layout methods for EMC compliance, along with a justification of why the technique works. The concepts presented will vary for each particular PCB design.

Engineers may focus on analog, digital, or system-level products but, regardless of specialty, whatever they produce must be suitable for production. Frequently, more emphasis is placed on functionality than on system integration, with system integration usually assigned to product engineers, mechanical engineers, or others within an organization. Design engineers must now consider other aspects of product design, including the layout and production of PCBs for EMC compliance. Considerations include recognition of the way the electromagnetic fields transfer from circuit boards to the chassis and/or case structure. In addition, cost must be minimized during design, test, integration, and production.

If a product fails to meet regulatory compliance tests (EMC and product safety), redesign or rework may be required. This redesign significantly increases costs such as the following:

1. Engineering resources (along with administrative overhead).
2. A new PCB layout, including artwork and gerber files.
3. Prototyping material required for a new product build.
4. System integration and testing to validate functionality and compliance.
5. Procurement of new or additional components for quick delivery (very expensive).
6. New or modified in-circuit test fixtures and documentation.
7. Retesting for EMC compliance and product safety.

These costs are in addition to loss of market share, delayed shipments, loss of customer faith in the company (goodwill), a potential drop in stock price, anxiety attacks, as well as many other issues not detailed in the list. Personal experience as a consultant has allowed me the opportunity to witness several times such problems faced by start-up companies.

Not only must a design work properly, it must also comply with international regulatory requirements. Engineers who specialize in regulatory issues must evaluate products based on different standards. The present guide describes techniques that will alleviate existing conflicts among various layout methods.

A great deal of technical information related to PCB design and layout is available commercially, as well as from public domain documents. Typically, these sources provide only a brief discussion of how to implement a layout technique to solve an EMI problem without justifying the need, use, or application. (Several sources are listed in the Bibliography.)

My focus as a consultant is to assist and advise in the design of high-technology products at minimal cost. Implementing suppression techniques saves money, enhances performance, increases reliability, and achieves first-time compliance with emissions and immunity requirements, in addition to having the product function as desired.

This industry has allowed me to participate in state-of-the-art designs as we move into the future. Although my focus is on the technology of the future, one cannot forget that simple, low-technology products are being produced in ever increasing numbers. Although the thrust of this book is toward high-end products, an understanding of the fundamental concept of EMC suppression techniques will allow any PCB being designed to pass EMC tests. When one does not understand fundamental concepts, compliance and functional disaster may be the result.

Mark I. Montrose
Santa Clara, California