Book Review

Rey Perez, Associate Editor

Book: EMC and the Printed
Circuit Board, Design, Theory, and Layout Made Simple
Publisher: Wiley InterScience, 1999
Author: Mark I. Montrose

This review comes somewhat belated for many of you. In reality, I have to confess, that it was several months ago that I got a copy of the book from IEEE Press (now Wiley InterScience). The author told me recently that this book has been translated into several languages, including Chinese and Japanese and is selling well.

This book was written for the general Printed Circuit Board (PCB) designer who wants to understand the EMI issues that exist within a PCB. The book is outlined in nine chapters; each chapter covers an EMC issue in the PCB design process. The focus of the book is mainly on EMC for PCB. Other EMC issues such as shielding are covered only briefly. Each chapter contains a list of useful references.

Chapter 1 addresses EMC fundamentals. It really sets up the fundamental concepts of EMC and its terminology. Such fundamental concepts include the electromagnetic environment, the need for product compliance in that environment, methods of noise coupling (i.e coupling path mechanisms) of radiated and conductive nature, and the nature of interference in a PCB.

Chapter 2 is titled EMC Inside the PCB. The chapter opens with the high frequency behavior of passive components, including PCB traces (which develop reactance elements of considerable magnitude at higher frequencies). The chapter explains, in simple illustrative approaches, some of the fundamental concepts on the theory of electromagnetics, and how RF exists within a PCB. Proper steps used in the minimization of inductive coupling are discussed for a PCB, since it is one of the major sources of EMI in a PCB (the other major ones being common mode and differential mode currents and ground loops). Other frequency effects discussed are skin effects and lead inductance (as it applies specifically to wires). The most important concepts discussed in Chapter 2 are differential mode and common mode currents. I found the explanation of these two concepts very useful for PCB designers, including the conversion between differential and common mode currents.

Chapter 3 is titled Components and EMC. It deals mostly with the RF energy generated by digital signals during transients resulting from switching currents within a PCB. These transients produce RF energy that is directly proportional to the rise and fall times of the switching currents. Table 3.1 of the chapter shows the approximate rise/fall times of different types of components in the logic family. The table also shows the principal harmonic content and the typical EMI frequencies observed from these components. The transient currents are responsible for creating ground noise, which is mostly observed as differential mode noise and it is then converted to common mode noise currents. Common mode current is the main source of radiated RF energy. Another source of current transients is the power supply transition currents to a gate's input ($I=CdV/dt$) which can account for many milliamps. Other important subjects discussed in Chapter 3 are: a) the effect of component packaging in the loop area of the die which can contribute to EMI, b) the current flowing in a loop area between components which can also cause EMI, c) the common mode currents from cabling which can cause the largest source of EMI (analytical approximate expressions are given for the radiated EMI from "b" and "c" current sources), d) ground bounce, e) crosstalk from lead-to-lead capacitance, f) parasitic effects of heat sinks and the proper grounding of heat sinks to avoid becoming un-intentional antennas, and f) power filtering of clock sources (to minimize noise and ground bounce). About four different filtering techniques are discussed.

I find Chapter 4 (Image Planes) the most interesting chapter in the book because it discusses the main reasons for EMI in a PCB. The concept of image planes was originated from the effort needed to provide the best return path for RF currents, considering that there are possible a number of parallel return paths that the return current could follow. The number one return path is referred to as the image plane. A good introduction to the image plane is started by the concept of partial inductance. The author then proceeds to show how only when the RF return path is connected to the power and ground pins of a component will a real image plane exist. The author shows that the separation between the trace and image plane is the main issue for reducing the common mode current in the image plane (by reducing mutual partial inductance). A related concept in Chapter 3 is the discussion of ground and signal loops. The author explains how these ground loops can be formed in a poorly designed PCB. Image planes prevent RF ground loops from being developed because RF currents couple themselves to their source trace without having to find alternate return paths. The author shows in the chapter that the distance between ground stitch locations to a 0V reference should not exceed l/20 to avoid potential RF ground loops.

Image plane violations are discussed in Chapter 4, as well as the proper use of vias. The concept of split planes (i.e. splitting ground planes for providing return paths of analog and digital components in the PCB) is discussed in the chapter. The isolation/partitioning of the actual components in the PCB is also discussed through several approaches (isolation, bridging). Proper RF return paths are also discussed for wiring interconnects. For those PCBs that cannot avail themselves of multi-layer image /ground plane designs (i.e. single and double sided boards), design approaches are discussed. The chapter ends with a brief discussion of localized ground planes.

Chapter 5 is titled Bypassing and Decoupling. Bypass capacitors are used in a PCB to prevent the transfer of energy from one circuit to another, maintaining the constant power distribution among all the PCB components (specially those power hungry). Decoupling assures low impedance power supply during switching operations. Low impedance allows high frequency noise to get diverted away from signal traces, but low frequency noise remains mostly unaffected. Three types of capacitors are covered in the chapter: a) decoupling which removes RF energy injected into the power distribution network from high frequency components in switching power supplies. Decoupling capacitors also provide localized source of DC power for devices and components, eliminating the peak current surges that could occur, b) bypass capacitors are used to divert common mode current from PCB components and cabling, creating a shunt to noise energy from entering susceptible PCB circuits, and c) bulk capacitors are used to maintain a constant dc voltage and current to components during switching operations (low drop out voltage). Because capacitors are not truly passive devices as the circuit frequencies increase (i.e. they become reactive devices with inductive and resistive effects), Chapter 5 goes through all the complexities that arise from this fact in the actual PCB design process. The work is also extended to power and ground plane capacitance. Practical aspects discussed in the chapter are the proper placement of capacitors for the different applications and the proper selection of decoupling capacitors (i.e. the calculation of capacitor values for decoupling applications). The chapter ends with the concept of designing buried capacitance in IC packages.

Chapter 6 on Transmission Lines is written to provide the reader with knowledge and understanding that high speed digital design is really the process of learning how to reduce propagation delays, manage reflections and crosstalk (i.e. signal integrity), and reduce signal losses in PCB where all the interconnects behave as transmission lines. Only the PCB "transmission lines" are discussed. The chapter overviews the different types of transmission lines created during the PCB layout process (microstrip, embedded, single stripline, dual stripline, differential microstrip and stripline) including the appropriate formulas (these are really quasi-static approximations at lower frequencies) for impedance, capacitance, and propagation delays. I found the concept of capacitive loading discussed in the chapter to be the most important section.

Chapter 7 is titled Signal Integrity and Crosstalk. When components operate at higher frequencies, signal edge rates become faster in order to accommodate smaller clock pulse intervals. The behavioral characteristics of source and load devices, the conductor impedance (including the "loading" caused by the components' input impedance), the physical and electrical parameters of the PCB materials, parasitic capacitance and inductance of circuit elements, fast edge rates, and the trace lengths, all play a role in the characteristics, quality and reliability of the signals being transmitted (and this is known as signal integrity). For example, if the trace is electrically long (edge transition time of the signal is less than the time it takes for the signal to travel from source to load and return from load to source) signal integrity issues arise. Transmission line effects such as reflections, overshoot, undershoot, and crosstalk can distort the quality of the transmitted signals. Sources of noise that may affect the signal integrity are: reflections, ground bounce, crosstalk, thermal effects, ground effects, power supply variations, etc. The chapter addresses several of the most important aspects of signal integrity; such as: a) reflection and ringing (identification of the ringing problem, conditions that create the ringing problem, and its theory), b) calculating the correct trace lengths during the PCB layout (avoiding electrically long traces) with several worked examples, c) the effect of loading the transmission line, and d) the concept of crosstalk (coupling of signals among traces). The chapter ends with some design techniques to avoid crosstalk.

Chapter 8 is titled Trace Terminations. Trace terminations are important for guaranteeing signal integrity in some components. The need to terminate a PCB trace is based on several design criteria, of which the most important are electrically long traces. The author states that when the trace is electrically long or when the length exceeds one-sixth of the electrical length of the edge rate, the trace requires termination. Even if the trace is short, termination is required if the load is capacitive or highly inductive to prevent ringing. The book covers resistive series and parallel terminations with examples. These terminations can match trace impedance, reduce ringing and reflections and can also slow the edge rate of the clock signals. The book discusses advantages and disadvantages of a series and parallel terminations. The chapter also discusses alternative destination methods such as Thevenin network and RC network terminations (including advantages and disadvantages).

The last chapter in the book is Grounding. This chapter is typical of similar chapters on grounding from other EMC books. It covers such topics as fundamental grounding concepts, ground loops, safety ground, grounding methods (single point ground, multipoint ground, hybrid), common impedance coupling (and how to avoid it). The chapter ends with such concepts as resonance in multipoint grounding and field transfer coupling of daughter cards to card cage. **EMC**

---

return to contents