

TECHNICAL ARTICLE

Passing EMI Compliance Testing the First Time—Part 1: The Physics

James Niemann, Field Applications Engineer

Abstract

Any product designed today that requires high speed clocks can be troubled with electromagnetic compatibility (EMC) compliance issues. This article outlines an electromagnetic (EM) field-oriented perspective for printed circuit board (PCB) design intended to help the reader pass electromagnetic interference (EMI) compliance testing the first time. The same techniques used for reducing EMI will also mitigate interference, suggesting a universal PCB layout philosophy. This article is presented in three parts. The first article covers the essential physics needed to understand why the techniques described within this article will mitigate EMI on a PCB layout. Part 2 will cover how the material in Part 1 plays into a practical board layout described through examples. Finally, Part 3 will discuss PCB layout strategies for complex boards that will satisfy the presented solutions.

Introduction

Why is it that compliance is often the last thing addressed during the design process? This is often the case because the entire system needs to be available before any of the actual testing can take place. Problems uncovered late in the design may require printed circuit board (PCB) changes, which are always expensive when many people are involved late in the product design cycle. This is unfortunate because electromagnetic compatibility (EMC) compliance (radiated, conducted, or susceptibility) can be confidently designed to pass simply by following the best practices described within this article. The methods described will help keep electromagnetic interference (EMI) compliance in mind during the architecture, design, and layout of PCBs. This article aims to do much more than demonstrate how to design a board to pass EMC testing. The goal is understanding the underlying principles and why they work so that the principles can be leveraged elsewhere.

The fundamentals behind EMI, interference, and susceptibility really are quite simple. Mitigating EMI is all about controlling and confining electric and magnetic fields. Simply put, if electromagnetic (EM) fields are exposed to free space and allowed to expand, interference and EMI will result. How is this field confinement accomplished? All that is needed is to design for zero NET accelerating charge everywhere on the circuit board (and in

any connected cables). This means for each charge that is accelerating, there is another charge very close by that is permitted to accelerate in the opposite direction to the original charge. When these events occur together in the far field, the effects of each charge can and will cancel. Without field energy, there can be no EMI or interference. This will leave all EM field energy confined inside components or the PCB dielectric directly between the signal trace and the ground plane. This is the idea for traces on the boards but in truth, the electrically long antennas with unconfined fields encountered on isolated systems is the bigger problem. This topic will be discussed in a future article.

Physics of Static and Dynamic Forces on Charge

Charges move only when they are exposed to an electric field. The electric field at a given location in space can be static (such as the field from a stationary charge). The electric field at a location can also change with respect to time (such as the field from a moving charge). Once charges are moving, the magnetic field comes into existence. The magnetic field, like the electric field, can be static or can change with respect to time. We know from the Lorentz force, $F = q(E + v \times B)$ that a moving charge in a magnetic field will experience a force either toward or away from the moving charges generating the magnetic field. Since only

an electric field can exert a force on charge, the magnetic field is really a relativistically generated electric field. It is a manifestation of the original coulomb field of the moving charges. The electric and magnetic fields are really one thing that can look different depending on your frame of reference.

When these fields change with respect to time, there is a possibility that radiation or EMI may result. Charges can be static, they can move in space, or they can be accelerated. Radiation results only when charges are under acceleration. This is due to the simple fact that the magnetic field is an electric field on the inside, and an accelerating charge will cause this relativistic field to expose its energy. Accelerating charges allow both Ampere's and Faraday's law to be invoked to create the propagating field in space. The mathematical reason for this can be found by closely examining Maxwell's equations shown below. The third and fourth equations credited to Ampere and Faraday are key to understanding radiation and EMI.

Gauss' law for electric fields:

$$\oint_S D \times dn = \oint_S \rho dv \quad (1)$$

Gauss' law for magnetic fields:

$$\oint_S B \times dn = 0 \quad (2)$$

Ampere-Maxwell law:

$$\oint_L H \times dl = \int_S J \times dn + \delta / \delta t \int_S D \times dn \quad (3)$$

Faraday's law:

$$\oint_L E \times dl = -\delta / \delta t \int_S B \times dn \quad (4)$$

These equations suggest that there is more than one way to generate electric and magnetic fields besides using charges. Changing or dynamic fields can also create fields. This effect is responsible for propagating electromagnetic field energy in free space or on the PCB. Poynting's energy vector describes the movement of this energy.

Poynting's vector:

$$S = E \times H \quad (5)$$

Fields are always present wherever voltages and currents exist. The fields are where the information and energy live. It is neither feasible nor desirable to eliminate EM fields. The goal is to control their location so that they cannot exert forces on victim charges (interference) or are exposed to free space (EMI).

The good news is that confining these fields can happen automatically if the PCB is designed correctly.

Any distribution of charges, moving or stationary, will always arrange themselves to minimize the stored or dissipated energy taken (from the distribution of charges).

This statement is true for electrostatic or magnetic energy. The charges involved in the distribution will first arrange themselves to store as little energy as possible. As time goes by, the arrangement will revert to dissipate as little energy as possible. This behavior is simply the result of the force's charges exerted on one another.

How can boards be designed to store the least energy? This can be accomplished by providing a ground plane very close to the signal and power paths. The proximity of the ground plane will permit the copper geometry to store the least electric and magnetic energy. This is accomplished because the electric field will pull charge from the ground plane close to null the net electric field as seen outside this dipole. Similarly, when charges accelerate, Faraday's law will provide a current in the ground plane nulling the external magnetic field. The board needs to be designed to allow this to naturally occur. It's also important to understand that both these effects occur almost immediately, delayed only by the speed C to the ground plane.

The preceding text also describes the attributes of a transmission line. Most engineers know all about the transmission lines and use them every day for high speed circuits. To avoid interference as well as EMI, these transmission line techniques must be used for all PCB applications. Probably the most used structure in PCB design is the microstrip transmission line, basically just a trace over a ground plane. In theory this will confine the electric and the magnetic fields to the space between the trace and the ground plane. Figure 1 shows that in volume A, Maxwell's equations state that the total charge enclosed is equal to the integral of the electric flux crossing the surface of the volume. Since zero charge is enclosed, the net electric field outside the surface will approach zero as the separation of the internal equal and opposite charges co-locate. The transmission line, surface B, will enclose zero accelerating charge. Maxwell's equations state that the line integral of the magnetic field around this surface B will also approach zero if the net current through the surface is zero. Due to Faraday's law, the line integral around surface C will also approach zero.

The space between the transmission line and the ground plane cannot be zero (resulting in zero external field)—the 50 Ω transmission lines are needed to maximize signal bandwidth for RF and fast digital applications. For power, a much lower impedance of only a few ohms is used.

Remarkably, all this physics can be distilled down to three good practices to follow during layout. Each of these is discussed in the following section.

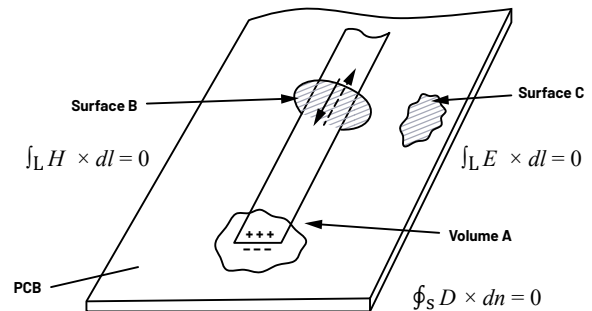


Figure 1. A representation of the electric and magnetic fields surrounding a transmission line.

Good Practices

#1 Strive for zero NET accelerating charge.

This practice encourages designers to be aware of current paths in PCB design. Consider if the board layout allows nature to provide a cancelling current by Faraday's law. Remember, wherever there is current, it must have changed at some point to get to that value.

#2 In layout, keep the E and B fields confined to a dedicated small space.

This is really the result of the application of the first practice. If the layout allows for zero NET accelerating charge, the E electric field should also be confined. It is important to keep this result in mind separately during layout. Both the electric and the magnetic fields are needed to move energy.

#3 Think about EM fields instead of voltages and currents.

Energy and information move in the fields, not in the copper on the PCB. A zero external field will never be achieved with an imperfect microstrip. The larger the transmission line impedance becomes, the less confined the fields will be. Fortunately, the higher impedance lines (50 Ω for maximum bandwidth) generally operate at lower currents with initial values limited to the voltage divided by the 50 Ω characteristic impedance. Power planes are lower impedance (they should be, which is good) but run at higher currents (not so good) and are required to be under damped (possessing minimal real resistance), or losses would be too large. These high Q circuits can cause significant radiation, which will be discussed in the next article. Nature is working for you when it comes to EMI rather than against you, always arranging charges to take the least energy, stored or dissipated from the circuit.

PCB layout can cause EMI and interference due to either magnetic or electrostatic coupling (mutual inductance and mutual capacitance). This kind of coupling happens because the structures used to move energy and information on the PCBs are not perfect; they do not completely confine fields as well as a coaxial cable. Space (the dielectric) between the waveguide (the traces and the ground plane) will lead to interference and EMI unless the geometry can completely confine the field.

The next article discusses the actual board layout, exposing common sources of radiation found from imperfect layout techniques as well as how to improve results. The following situations in PCB layout will be covered.

On the PCB, field confinement is commonly lost when:

- Signals transition between layers
- Signals share the same volume over a common ground plane
- Signals cross one another over a shared ground plane
- Signals are run in parallel
- Field fringing occurs
- Signals propagate down a microstrip or other imperfect transmission line

Conclusion

This article covered the essential physics and thought process needed to lay out a board for low emissions and EMI compliance testing. So far this has uncovered that only electric fields act on charges and that the magnetic field generated from moving charges is in fact a relativistic electric field. Provided charges are accelerating, Maxwell's equations and Poynting's vector show how fields can propagate through free space. All PCB copper geometries can be considered transmissions lines that can take advantage of how fields influence charges to confine fields to dedicated areas on the PCB. Part 2 of this article will build on this to illustrate how to lay out a PCB to minimize unconfined fields, which are the source of interference and EMI. Unconfined fields can come from two places: the circuit itself or the outside world (susceptibility). Each of these will be considered separately in the next article.

The key points and all that physics can be implemented simply by following the three good practices below.

- **Good practice #1:** Strive for zero NET accelerating charge.
- **Good practice #2:** In layout, keep the E and B fields confined to a dedicated small space.
- **Good practice #3:** Think about EM fields instead of voltages and currents.

Any distribution of charges, moving or stationary, will always arrange themselves to minimize the stored or dissipated energy taken (from the distribution of charges).

References

- Feynman, Richard P., Robert B. Leighton, and Matthew Sands. *The Feynman Lectures on Physics*, boxed set: The New Millennium Edition. Basic Books, January 2011.
- Johnson, Howard W. and Martin Graham. *High-Speed Digital Design: A Handbook of Black Magic*. PTR Prentice Hall, April 1993.
- Morrison, Ralph. *Fast Circuit Boards: Energy Management*. John Wiley & Sons Publications, January 2018.

About the Author

[James Niemann](#) joined Analog Devices in March 2020 and is currently a field application engineer in Cleveland, Ohio. James has 35 years of combined experience designing test and measurement equipment and working as an FAE at ADI. James has 14 patents.

Engage with the ADI technology experts in our online support community.
Ask your tough design questions, browse FAQs, or join a conversation.

ez.analog.com

 **ADI EngineerZone™**



analog.com

For regional headquarters, sales, and distributors or to contact customer service and technical support, visit analog.com/contact.
©2025 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners.

TA25796-3/25