Aswan University Journal of Science and Technology Volume 1, issue 2

Enhancement of Electromagnetic Interference Control Techniques Using Printed Circuit Boards Design : Review

Mohamed Alkurdi, Ahmed Mohamed Rashwan, Ashraf Mohamed Hemeida

Department of Electrical Engineering, Faculty of Energy Engineering, Aswan University, Aswan 81528, Egypt

Abstract— Electromagnetic Interference (EMI) and Electromagnetic Compatibility (EMC) are highly contributing to today’s modern technology as they refer to the term of handling signal integrity from the point of signal distortion due to the effect of surrounding electromagnetic fields. Our study here aims to control the electromagnetic interference affecting the printed electronic circuit board through layout design.

The following study has two major sections, the first one is the software simulation which analysis the signal value through the IC to the PCB regarding the parasitic components would affect the signal shape, then comes the second study as a practical part with data gathering of the signal shape in different EMI mediums. The Subtract of the simulation analysis from the gathered practical data will result the effect of only EMI on the signal. Repeating the previous approach with signal shielding, plane grounding shapes and board shielding will allow us to study and observe each design technique effect on the EMI control process.

The proposed study shall affect different sections such as areo-space electronic systems, military electronics, and medical electronic systems, as enhancing the EMC within a PCB shall reflect on signal integrity in these devices enhancing accuracy and reducing electromagnetic noise effect.

Keywords: — Electromagnetic Interference in printed circuit boards; PCB Design for EMI/EMC; Advanced layout for electromagnetic compatibility.

1. INTRODUCTION

In the 1940s and 1950s, electromagnetic interference and electromagnetic compatibility, abbreviated as EMI and EMC, became a worry for the first time. This was mostly due to the fact that motor noise was being carried over transmission lines and into sensitive machinery. The purpose of the military's primary concern in EMI/EMC throughout this time frame and throughout the 1960s was to guarantee that electromagnetic systems were electromagnetically compatible. Because of a few notable accidents in which radar emissions led to the accidental discharge of weapons or electromagnetic interference (EMI) resulted in the malfunction of navigation, the biggest priority of the army in terms of EMI/EMC was electromagnetic suitability, particularly within the weapons platform of an aeroplane or boat. [1],[2].

Because of the spread of computers throughout the 1970s and 1980s, disturbance from computing equipment would become a serious issue for the reception of broadcasting outlets, in addition to the transmission of radio signals transmitted by emergency services. [3].

electromagnetic compatibility (EMC) is a very active field of study that is always posing new issues as a result of ongoing advancements in semiconductor technology and the swiftly changing implementation circumstances [4].

Given that the very definition of interference is incompatibility, electromagnetic interference (EMI) is just the

Corresponding author E-mail: Mohamed.Alkurdi@aswu.edu.eg

Received July, 11, 2022, received in revised form, September, 20, 2022, accepted September 22, 2022.

(ASWJST 2022/ printed ISSN: 2735-3087 and on-line ISSN: 2735-3095) https://journals.aswu.edu.eg/stjournal

absence of EMC. EMI occurs when one electronic equipment transmits disrupting electromagnetic radiation to a second via radiated and/or conducted channels [5],[6]. In everyday terminology, the phrase is most often applied to radio frequency (RF) transmissions. EMI can take place in what we know to be the frequency range that is "anything greater than DC to daylight" [7].

Interference emissions and inductive/capacitive coupling are only two of the many EMC-specific issues that can arise from a poorly designed PCB design. Avoiding signal coupling and establishing proper reference grounds are two ways that EMC-compliant designs tackle such problems [8].

Electromagnetic interference (EMI) is caused through either conducting (voltages and/or currents) or radiating (electric and/or magnetic fields) undesired emissions. EMI can be either transitory, spontaneous, or stable in the temporal domain. Elements of EMI can span the frequency spectrum from the reduced power levels of 50, 60, and 400 Hz all the way up to the microwave range [9].

EMI signals can be coherent or noncoherent, and narrowband or broadband, all within the frequency domain. EMI can come from either natural or artificial sources. Within the category of "man-made," it is further distinguished into deliberate and accidental (or inadvertent) EM I source. To aid in the identification of EMI sources, the characterization of EMI receptor susceptibility, the determination of EMI coupling routes, and the facilitation of EMI management strategies, EMI has been organized into a number of different categories [10].

Due to the heightened clock frequency, high density of the devices, and high-power consumption, the electromagnetic environment inside the package and PCB is very complex. This results in lots of potential EMC problems, such as the following:

1. Interconnector delay and loss. For the increased clock frequency, the length of the interconnector is comparable with the working wavelength. The interconnector must be taken as the transmission line, of which the propagation delay cannot be ignored. At the same time, at a high frequency, the skin effect of the current and the dielectric loss greatly increase. The interconnector loss becomes serious [11].

2. Impedance mismatching. At a high frequency, the impedance mismatching between the interconnectors and the circuits and the discontinuities along the interconnectors will result in multireflection of the signal. This degenerates the signal propagation quality [12]. One of the common interconnector discontinuities in PCBs is the through-hole via. The through-hole via provides electric connection between traces at different PCB layers. However, its parasitic capacitance and residual stub also make the impedance of the interconnector discontinuous [13],[14].

3. Simultaneous switching noise (SSN). When lots of digital circuits switch at the same time, a heavy current will be drawn from the power supply. Owing to the parasitic inductance and resistance of the power supply system, there will be fast swing of the supplied voltage. This will result in the unstable operation of the integrated circuit (IC). Things become worse for modern high-speed circuits due to the reduced voltage supply level and noise margin [15],[16].

4. Cross talk (XT). For the dense layout of the circuits and interconnectors, there is strong electromagnetic coupling between them. The signal transmitted on one circuit or interconnector will create interference in another circuit or interconnector. This cross talk increases the bit error ratio of the circuit system [17].

5. Unintended antennas. When the dimensions of the structures in PCBs, such as the traces and the slot on the power–ground planes, are comparable with the working wavelength, they will become effective antennas. Their radiated electromagnetic field will disturb the normal work of the nearby circuit system. One of such unintended antennas is the heat sink on the PCBs [18].

6. Susceptibility or immunity. Above external EMI will induce voltage or current in the IC. This requires that the IC should have certain immunity to protect itself from the external interference or the susceptibility of the IC should be known to make sure that it survives in a complex PCB environment [19],[20]. There are more EMC problems related to the high-speed circuits than what has been listed earlier [21].

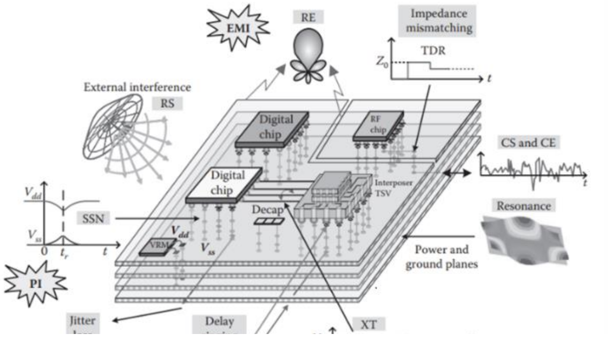


Figure 1 EMC Problems related to high-speed circuit [90].

2. EMI SOURCES

The EMI is relatvent to the change in electrical current or voltage in a conductor in response to time as the broader the range of RF interference, the quicker the change rate of current or voltage in the machinery that is causing it. The more noise voltages or currents there are, the more conducted and radiated emissions there will be **Error! Reference source not found.**,**Error! Reference source not found.**.
Since the commutator switches the inductive coils in the rotor, electric motors produce high voltages and currents with quick rise times. This makes them good instances of powerful sources of EMI **Error! Reference source not found.**.

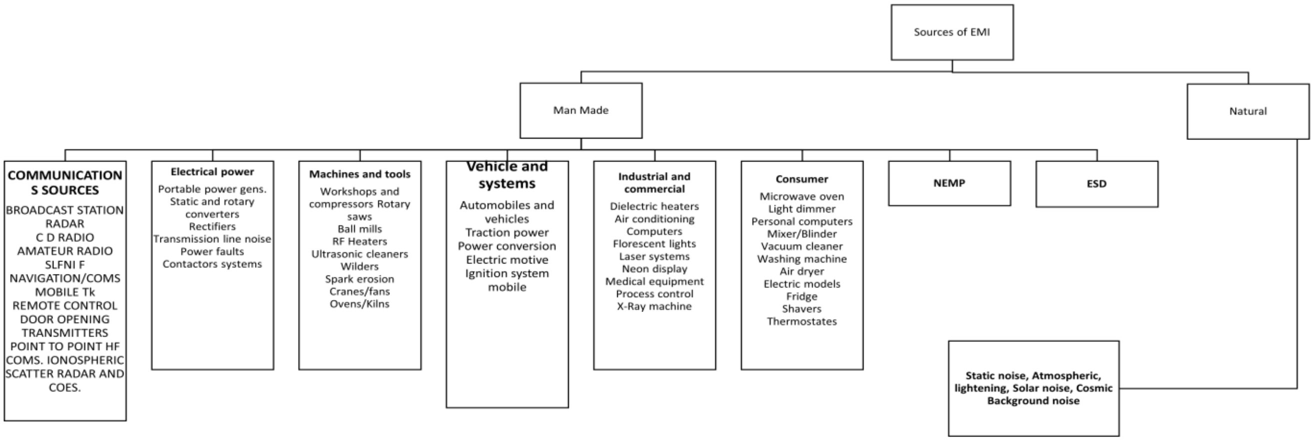
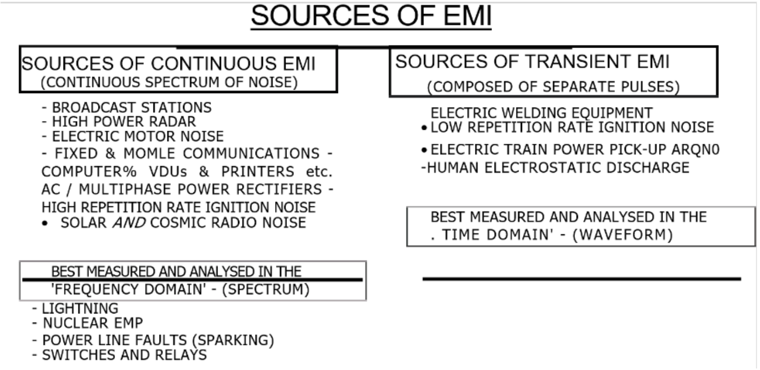


Figure 2. Groups of EMI sources [116].

Figure 2 shows how these components can be clustered by the way they are used. As shown in Figure 3, they can cause interference that lasts all the time or only for a short time. Continuous sources involve radio transmitters and the emissions from RF heaters, in which the signal is a continuous carrier. Pulsed sources include radars and the emissions from digital computers, which have a wide but stable RF spectrum. Narrow-bandwidth scanning receivers or spectrum analyzers are the best tools for measuring and analysing the spectrum of such sources' emission levels **Error! Reference source not found.**,**Error! Reference source not found.**.



SOURCES OF EMI

SOURCES OF CONTINUOUS EMI
(CONTINUOUS SPECTRUM OF NOISE)

- BROADCAST STATIONS
- HIGH POWER RADAR
- ELECTRIC MOTOR NOISE
- FIXED & MOMLE COMMUNICATIONS -
COMPUTER% VDUs & PRINTERS etc.
AC / MULTIPHASE POWER RECTIFIERS -
HIGH REPETITION RATE IGNITION NOISE
• SOLAR *AND* COSMIC RADIO NOISE

BEST MEASURED AND ANALYSED IN THE
'FREQUENCY DOMAIN' - (SPECTRUM)

SOURCES OF TRANSIENT EMI
(COMPOSED OF SEPARATE PULSES)

ELECTRIC WELDING EQUIPMENT
• LOW REPETITION RATE IGNITION NOISE
• ELECTRIC TRAIN POWER PICK-UP ARQN0
-HUMAN ELECTROSTATIC DISCHARGE

BEST MEASURED AND ANALYSED IN THE
. TIME DOMAIN' - (WAVEFORM)

- LIGHTNING
- NUCLEAR EMP
- POWER LINE FAULTS (SPARKING)
- SWITCHES AND RELAYS

Figure 3 Sources of continuous and transient interference [116].

3. EMI EFFECT ON PCB

EMI effect on a PCB appears in the terms of signal integrity and power integrity leading to data lose and in severe cases IC damage **Error! Reference source not found.**,**Error! Reference source not found.**.

3.1.Signal integrity

Within the wider definition, "signal integrity" makes reference to all the disruption created by the connections in high-speed commodities **Error! Reference source not found.**. It has to do with how the electrical characteristics of the interconnects interact with the voltage and current waveforms of the digital signal, and how that might have a bearing on how the performance plays out **Error! Reference source not found.**.

3.2.EMI Emitted in PCBs

Microstrip patch antennas and PCBs both use a time-varying fringing electric field at the board's borders to create radiation. This phenomena has been described in recent works, and its characteristics have been characterized logically and by computational simulations **Error! Reference source not found.**,**Error! Reference source not found.**.

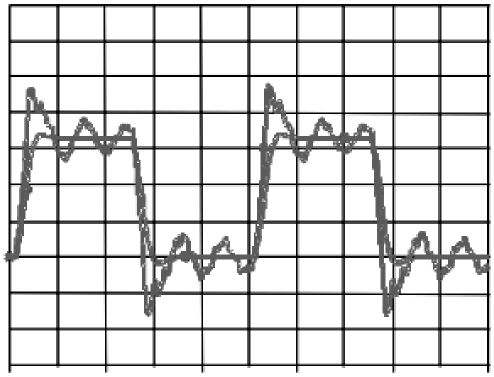


Figure 4 A source-series terminated interconnect line eliminates ringing but has poor signal quality in an unterminated one. There are just two cases where the PCB trace is longer than two inches. Scale is 1 v/div and 2 nsec/div.

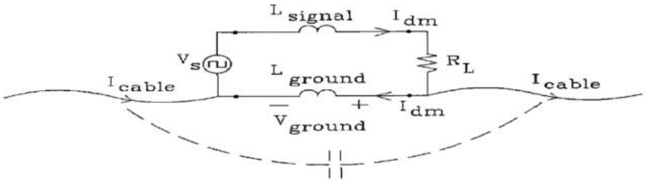


Figure 5. The non-zero impedance (inductance) in a signal return wire may be seen of this sketch, which illustrates the physics involved in determining the source of cable-borne radiated light and electromagnetic interference

The first 8-bit microprocessors, which were manufactured in 1974 and 1975 and served as the spark that ignited the personal computer revolution, ran at nominal clock speeds of 1–2 MHz. At these frequencies, unintended radiation was mostly caused by cables that were departing the electronics; nevertheless, these cables were inefficient radiators due to the fact that they were electrically extremely short. Managing unintended radiation at low megahertz clock rates was frequently a question of shielding and filtration at the conduit contact **Error! Reference source not found.**.
This was the case especially when the clock rate was low. As long as the standards could be satisfied with easy mitigation strategies like as "grounding," shielding, and filtering, the specifics of the electromagnetic physics of the connection between noise source in the designing and the radiated cable were not critical. However, beginning in the middle of the 1980s, the demand for integrating electromagnetic compatibility (EMC) into the product development from the very beginning started to increase. This was due to the fact that design cycles were getting shorter while cost pressures and design densities were getting higher **Error! Reference source not found.**,**Error! Reference source not found.**.

The system was delayed as a consequence of the adoption of EMI solutions at the very conclusion of the design cycle, which also resulted in additional costs. The necessity to eradicate the iterative process of EMC retrofits, known as "trial and error," in order to achieve EMI compliance was developing. However, there was a widespread dearth of understanding regarding the physics of EMI coupling which could be directly attributed and statistically to the circuit configuration **Error! Reference source not found.**.

4. PCB DESIGN TO REDUCE EMI

There are several approaches to reduce EMI effect on PCB signal integrity, the following approaches are some of the most common methods used through the design process to achieve the EMI elimination goal:

- Ground Planes
- Electromagnetic Band-Gap Structure
- Signal shielding

4.1. Ground planes

Mutual coupling of trace and ground plane is among the major important variables influencing the EMI interference problem in high-speed digital circuits. Two-layer and four-layer PCB designs are implemented on the clock circuit to mitigate noise and allow for an examination of the radiated emission from PCB ground plane **Error! Reference source not found.**.
The investigation, evaluation, and measurement of radiated EMI that was caused by employing two pieces of PCBs with varying areas and locations of ground planes is the primary emphasis of this research **Error! Reference source not found.**,**Error! Reference source not found.**.

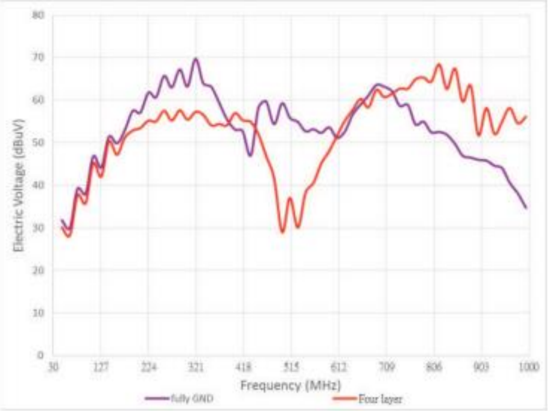


Figure 6 The findings from measurements pertaining to two- and four-layer PCB designs [20].

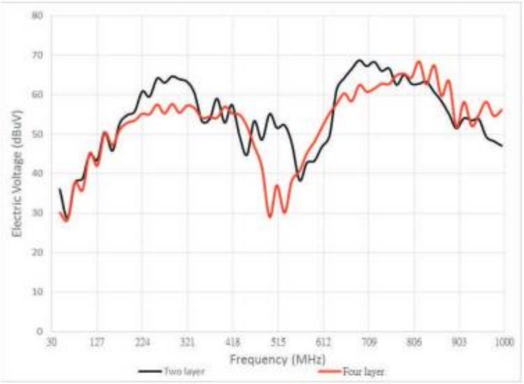


Figure 7 There is a significant difference between the emissions produced by a two-layer PCB and a four-layer PCB system when tested in full ground [20].

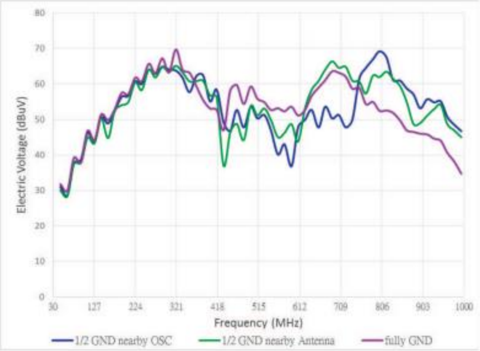


Figure 8 The comparison of emissions in different ground location under the two-layer PCB [20].

4.2.Electromagnetic Band-Gap Structure

EBG structures are part of a large group of man-made materials called "meta-materials," which were first used for antennas because of the unique way they behave. EBG frameworks can meet the perfect magnetic conductor (PMC) circumstance within a specific frequency band and force regular incident waves to have a zero-reflection sequence. This makes them good for implementations like reducing coupling among antennas and improving antenna directivity **Error! Reference source not found.**,**Error! Reference source not found.**.

Even though EBG frameworks have been researched a lot, these research findings have mostly centered on open structures (those that aren't enclosed in contexts like PCBs) and regular incident waves, including antenna applications. This means that the models and theories that have come out of these research findings can only be utilized in these situations **Error! Reference source not found.**.

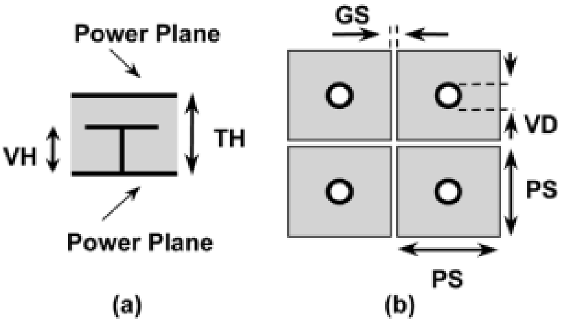


Figure 9. The geometrical characteristics of the EBG architecture. PS denotes patch size, VD via diameter, GS gap size, VH via length, and TH interplanar distance (a) Lateral view. (b) Top view [70].

Figure 9 depicts a common EBG framework, being the relatively simple one. It is made up of metallic patches on top of the vias and periodic vias with a diameter of VD that are connected to each other by a metallic surface. In Figure 9, the shape of such spots is shown as PS. They are detached from one another through gaps of size GS and from the plane they are linked to by a via of with VH length **Error! Reference source not found.**,**Error! Reference source not found.**.

The pattern is periodic with a period of (PS+GS), and it may be found in the space between two parallel metallic plates that are spaced a distance of TH apart from one another. By suppressing surface waves within a frequency range that can be predicted, the entire structure performs the function of a band-stop filter. This frequency range is a function of the geometrical features of the structure (such as periodicity, patch size, gap size, via diameter, via length, and also board thickness), as well as the dielectric material used in the printed circuit board as substrate. For example, periodicity is a function of patch size, gap size, via diameter, and via length. According to the authors' best knowledge, there is currently no equation that can be found in the literature that gives a good relationship between the geometrical features of the structure, on the one hand, and the center frequency and band-stop region, on the other. This is the case even though the authors have done their best to find such an equation. The design approaches that are utilized for EBG buildings are discussed in detail in Section IV **Error! Reference source not found.**.

4.3. Signal shielding

EMC is an extremely significant issue when it comes to high-speed circuits, particularly in the RF circuits. Many such circuits weren't able to realize the purpose that they were presumed to perform, and the schematic designing wasn't the core motive why. The core reason essentially has to do with designers not properly considering EMC, that resultantly triggers a significant disruption of the electro-magnetic field, that then leads to errors with the system. The coupling that occurs among neighboring micro-strip lines is one of the primary factors that contributes to the disruption of electromagnetic fields **Error! Reference source not found.**.

It is evident that when the microstrip line has current passing through it, in the nearby ground plane, a corresponding ground current will then be distributed (as illustrated in Fig. 10) an experiential formula [l] can be used to explain the ground current's distribution:

$$i(D) = \frac{i_o}{\mu H}\frac{1}{1+(D/H)^2} \qquad [1]$$

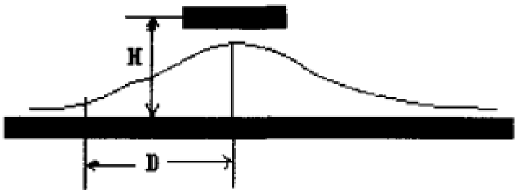


Figure 10. The distribution of the current in the ground plane [41].

Figure 11. the principle of coupling [41].

In this equation: Io= total current in the micro-strip line, A;
H= height from microstrip line to ground plane, m;
D =the distance in the horizontal direction, m.;
i(D) = the density of the ground current, A/m.

Once 2 micro-strip lines are adjoining to one another, the charge flow of both lines will be impacted by the opposite line. The explanation for this can be seen in figure 11; To be more explicit, when a current runs in the left micro-strip line, the distributive ground current that is caused by this induces a reactionary current to flow in the correct micro-strip alignment, that will lead to pairing between both the two lines. This allows the coupling to occur. Once this distance among these two lines is sufficient, the coupling will be small enough, and the affinity between lines will be insignificant enough to be disregarded **Error! Reference source not found.,Error! Reference source not found.**. However, as system speeds increase, PCB areas shrink, and distances between transmission lines shorten, we need a new approach to lowering coupling **Error! Reference source not found.Error! Reference source not found.**. Instead of increasing the length in between micro-strip lines, a novel approach is to place a guard trace with vias in between the problematic pairs. Obtaining a simulation outcome of the deployment of the guard traces is something that's happened in several contexts, as a good number of PCBs developers combine guard trace alongside vias to decrease pairing **Error! Reference source not found.Error! Reference source not found.**.

5. EMI MODELING

The simulation procedure consisted of the following actions:
The steps include creating the modelling approach, creating the origin vectors, verifying the DUT's resonance frequencies, verifying EMI using the created vectors, and optimizing EMI efficiency.

5.1. Building the Model

Resonance and electromagnetic interference (EMI) simulations cannot be performed until a model of the entire system is constructed. This includes the entirety of the board, such as capacitors, inductors, etc **Error! Reference source not found.**,**Error! Reference source not found.**.

The lumped model's behavior is unsuitable for our purposes because:

- No radiation is emitted out of lumped models
- Fields increase and decrease simultaneously across the system
- Only reactance may modify the voltage phase or current, and not length

One way to avoid these issues is to construct a distributed model using a complete wave extractor. **Error! Reference source not found.**,**Error! Reference source not found.**.

Capacitors, resistors, and inductors are all discrete parts that must be modelled as distributed systems rather than lumped systems in order to be included in the extracted system **Error! Reference source not found.**.

5.2. Source Vectors

In this case, a spice transient simulation was used to produce the source vectors. Both current and voltage vectors can serve as sources **Error! Reference source not found.**,**Error! Reference source not found.**.

Full wave 2.5D extractor pack and board, a comprehensive spice driver, and the final load are all part of the simulator **Error! Reference source not found.Error! Reference source not found.**.

Long enough simulation times are aimed at obtaining any frequency patterns in the system. Ideally, the platform's production-ready vectors should be used **Error! Reference source not found.**,**Error! Reference source not found.**.

Accuracy may be improved by considering all of the system's parameters. Case in point: the clock phase-locked loop (PLL), which, if it introduces jitter to the clock source, should be modelled in the simulation. Because the PLL introduces jitter into the clock, it works as a spread spectrum, reducing the power at any one frequency by spreading it across a wider range **Error! Reference source not found.**.

All transient outcomes should be converted to the frequency domain because the EMI solver techniques utilized in the frequency domain. The FFT is deployed for this purpose **Error! Reference source not found.**.

At the time of employing the FFT function, you must choose the size and type of the window with care. If the FFT is done with the incorrect parameters, the vectors will be wrong **Error! Reference source not found.**,**Error! Reference source not found.**.

The outcome of converting time to frequency is vectors of the signals in the frequency domain, with phase and magnitude for each important resonant frequencies **Error! Reference source not found.**,**Error! Reference source not found.**.

The driver in the DIE is used to make the source vectors (the source of the signal) **Error! Reference source not found.**.

5.3. System Resonance

No matter what the data pattern or working frequency is, it is extremely crucial to understand what frequencies are being emitted the most energy at. Recognizing the resonance frequency can assist in determining if the interface frequency point needs to be moved or if the system's resonance needs to be changed by changing the way the system is laid out and built. All of these things will help bring down EMI **Error! Reference source not found.**,**Error! Reference source not found.**.

System resonance is acquirable through the following three options, i.e., resonance simulation, s-parameters, near-field simulation through use of vectors with one amplitude used on all frequencies

5.4. Measuring EMI with Generated Vectors

System radiation is determined utilizing 2 distinct radiation solvers and several steps **Error! Reference source not found.**. For a planar, complex system, a 2.5D solver is used to figure out the near field close to the system. A 3D solver then uses the near field as a source to figure out the radiation farther away **Error! Reference source not found.**. In the full wave 2.5D solver, the vectors that were made are connected to the ports. The current source port is the same port that was used to make the system model **Error! Reference source not found.**,**Error! Reference source not found.**.

With current source vectors in the design, the near field for the exact system data pattern can be calculated **Error!**

Reference source not found.,**Error! Reference source not found.**.

The near field is determined by calculating from around the structure on the exterior of an illusionary cube that is extremely near said system. Subsequent to the near field's getting estimated, a full wave 3D solver can be used to figure out the system's radiation at 3, 10, or other ranges **Error! Reference source not found.**. The near field becomes a tool for the 3D solver as a source of radiation; and the structure then, instead, is developing through the same cube that encapsulated it when near field was being measured **Error! Reference source not found.**.

An open boundary step is imperative for the simulation being carried out if the radiation is to be measured irrespective of the distance **Error! Reference source not found.**. For this reason, computational areas are truncated. Within this layout, the PML or perfect matched layer is relevant because of its role as an absorption material of an artificial nature. When PML absorbs the exiting waves from the DUT, the distance becomes irrelevant when radiation is being measured **Error! Reference source not found.**,**Error! Reference source not found.**.

5.5. Signal Properties which Impact EMI

It is greatly impacted because of electrical signals. This is a list of the properties of great significance:

- Pattern
- Periodical/non-periodical
- Strength
- Frequency components
- Duty cycle for periodical signals

When discussing the signal we must remember that one of the core properties is periodical, where signals with an augmented power noted to a particular point of a frequency is by investigating the signal in the frequency domain. To be able to do this a time domain simulation of the full system layout is done and then a FFT is performed on the output signals **Error! Reference source not found.Error! Reference source not found.**.

6. EMI MEASUREMENT AND TESTING

EMI Measurement requires a source of EMI which usually antennas and a receiver that measures the EMI in a controlled environment **Error! Reference source not found.**.

Important transducers, the antennae used for broadband, help radiate or detect electromagnetic fields, in particular for EMC calculations**.** A dipole-like array antenna or LPDA is widely used in civilian RS EMC standards because the patterns on the antennae can be measured quite accurately with equations for electrically resonant and small dipoles **Error! Reference source not found.**.

The normal range for an LPDA is between 300 MHz and 1 GHz, but a modernized LPDA starts at 150 MHz. To improve FU, the single LPDA antenna was made better, and the double LPDA antenna was added. This is a set of two LPDAs that are put together at an angle. Compared to a single LPDA, this solution gives you more gain **Error! Reference source not found.**.

The double LPDA is a popular way to test RS these days, but it was recently taken out of the AECTP 501 Standards because it had big differences from measurements made with a DRGH antenna **Error! Reference source not found.**. This is a known antenna that works from 200MHz to 1 GHz and offers several benefits, including fewer side-lobes, larger gains, and little effect from the chamber walls and floor. Given that the basic antenna begins at 200 MHz, a longer double-ridged guiding horn (Ext-DRGH) was made that starts at 80MHz. This is essentially the beginning frequency for civilian standards. Researchers have been looking into LPDA and DRG horn antennas for a long time to find ways to make high field strengths, either for measuring or for designing and simulating purposes **Error! Reference source not found.Error! Reference source not found.**.

Due to its poor performance in broadband applications and polarization, a single LPDA is not as good. Even though these things seem to happen often, no publications talk about them or look at the differences or benefits. People liked the LPDA antenna because they thought it would give them dependable outcomes (calculated from basic equations). This is probably why this type of antenna is usually chosen for predisposing experimentation and other types of tests. The double LPDA should make FU better, but there are no published results to back this up. The properties of the extended DRGH are also better than those of the LPDA **Error! Reference source not found.**,**Error! Reference source not found.**.

A good amount of work has been already executed when it comes to RE testing and antennae. However, no papers are devoted to RS testing. Ext-DRGH performs better than single and biconical log-periodic antennae when it comes to FU **Error! Reference source not found.**.

There are a few authors who have also tried to determine a different test approach so that an alternate concept for testing both could be brought forth **Error! Reference source not found.**, in addition to improving the repeatability and reproducibility of RS tests **Error! Reference source not found.**,**Error! Reference source not found.**. However, no analyses or measurement comparisons were carried out for the UFA calibration, nor was the power determined for the different antennas (e.g. ordinary biconical, LPDA, double LPDA, and Ext-DRGH).

In this section, four different transmitting antennas were used to compare the 16 UFA marks and the efficiency of power.

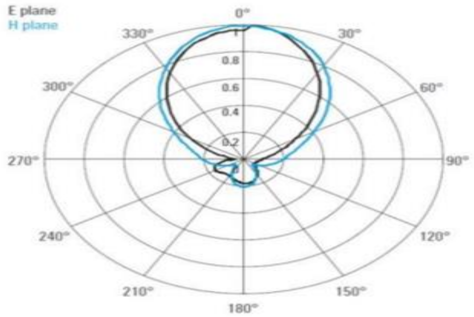


Figure 12 The radiation pattern of a double LPDA (HL 046) at 500 MHz.

This gave enough data to compare them when it comes to power efficacy, FU, and the E-field **Error! Reference source not found.**.

Throughout many situations, a single LPDA transmitter, which is made up of a group of dipoles, is employed to cover a broad frequency range **Error! Reference source not found.**,**Error! Reference source not found.**.

Such forms of antennae help give a low VSWR and decent directivity **Error! Reference source not found.**. LPDA's a frequency-dependent transmitter which utilizes many dipole elements of various lengths. The distances between the dipoles augment proportion to the length from the feed-point. This gives the LPDA antenna a wide frequency range and moderate directivity **Error! Reference source not found.Error! Reference source not found.**.

An enhancement of ordinary LPDA is a double V-shape LPDA or double LPDA. Figure 2.1 The radiation pattern of a double LPDA (HL 046) at 500 MHz, which is a typical field pattern for a double LPDA. The double layout makes it possible for the H-Plane directional pattern to become focused, leading to a typical augmentation of 2- 3 dB as opposed to a typical single LPDA **Error! Reference source not found.**,**Error! Reference source not found.**.

Figure 13 The radiation pattern of a double LPDA (HL 046) at 500MHz **Error! Reference source not found.** This is of utmost significance for susceptibility testing, that calls for the highest possible field strength as well as an excellent FU. Because the beam widths in the E-plane and the H-plane are almost exactly the same, an optimal lighting of the EUT can be achieved with just a tiny change in ground reflection. Double-Ridged Guide Horn (DRGH) Antenna Between the waveguide feeder and empty space, which has an impedance of 120π Ω, the DRGH antenna might be thought of as an impedance match or an RF transformer **Error! Reference source not found.**,**Error! Reference source not found.**.

Simply put, a DRGH transmitter is a horn antenna that has ridges added to it. A rectangular waveguide is packed with a centrally double ridged guide in order to attain a lower cut-off frequency **Error! Reference source not found.**.

The useable bandwidth of a pyramidal horn can be improved if the double-ridged pattern that starts in a waveguide is carried through into the horn itself. An Ext-DRG horn, the likes of which are depicted in Figure 2.2, was constructed and intended to work between 80 MHz and 1 GHz. This was done since the biconical antenna had quite a few issues, particularly in the near field **Error! Reference source not found.**,**Error! Reference source not found.**.

If we make the assumption that both the height (h) and the width (w) are at least one wavelength (), then the gain (G) will be controlled by the dimensions of the horn [100], as indicated in the formula (2):

$$G = 10 * \log(7.5 * h * w) \qquad [2]$$

The horn antenna has the potential to achieve a gain level of up to 20 dB in certain circumstances. In addition to this, it possesses a high degree of directivity, gain, and low levels of sidelobe and back lobe radiation. As a result of these qualities, the DRGH might potentially be appropriate for electromagnetic compatibility (EMC) tests, even with amplifiers that have a lower output power **Error! Reference source not found.**,**Error! Reference source not found.**.

This study used two approaches: An experimental method and a simple calculated (or calculable) method **Error! Reference source not found.**.

Measurements were conducted for four different types of antennas at a standard distance of 3 m. In order to verify the results, the calculable method was used based on a theoretical calculation **Error! Reference source not found.**.

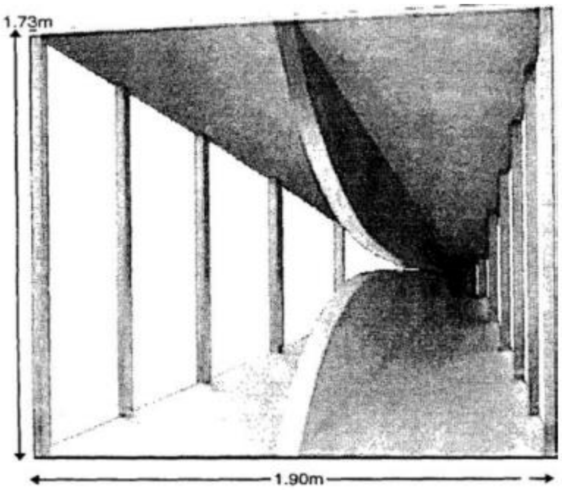


Figure 13. Extended double-ridged guide horn (Ext-DRGH) antenna

6.1. Measurement Setups

UFA measurements were carried out in a fully anechoic chamber using the methods described in **Error! Reference source not found.**. The following antennas were used in the measurements in order to get the results we needed:

- Bi-conical antenna (80MHz–200MHz)
- Single LPDA (150MHz–1GHz)
- Double LPDA or HL046E (80MHz–1GHz)
- Ext-DRGH antenna (80MHz–1GHz)

The LPDA test setup is elaborated in Figure 14 and Figure 15 and the ExtDRGH test setup can be seen in figure 16 Similar conditions were applied in both test setups in order to ensure comparable results **Error! Reference source not found.**,**Error! Reference source not found.**.

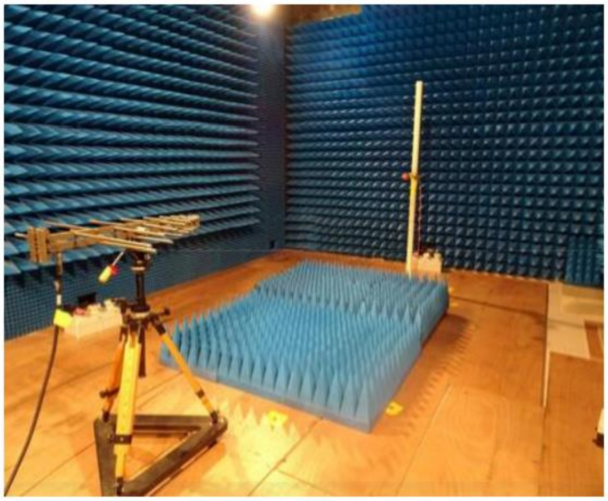


Figure 14 Test setup with single LPDA, originally from reference [120]

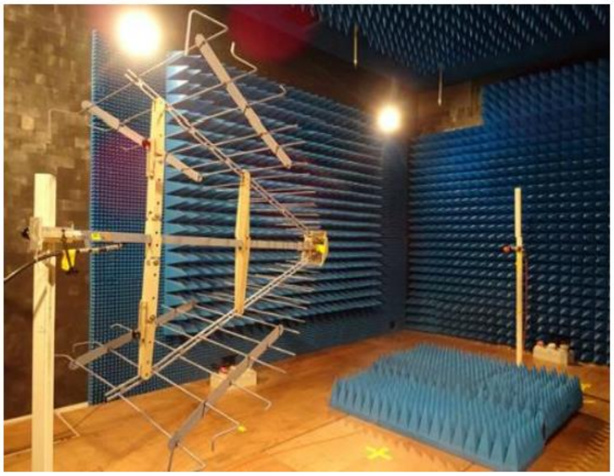


Figure 15. Test setup with single LPDA, originally from reference [120]

Figure 16. Test setup with Ext-DRGH originally from reference [120]

The link shared power introduced into the antenna – or forward power– and the E-field in the far-field in the linear domain, can be calculated with:

$$E\left(\frac{V}{m}\right) = \frac{\sqrt{30PG}}{R} \qquad [3]$$

where P is power, G is the gain of the antenna, and R is the distance between the phase center of the antenna and the UFA (here assumed to be 3 m). Determining the forward power from the generator or amplifier and using the G-value from the antenna calibration specification allows us to obtain the E-field value for each frequency point **Error! Reference source not found.**,**Error! Reference source not found.**.

CONCLUSION

EMI testing has gained importance due to the technological improvements in modern life. Better test techniques are needed to cope with rapidly changing and new challenges.

The study of Electromagnetic Interference effect on the electrical signal through various approaches in electronic printed circuit boards design point out the best practices to design a PCB layout. The experimental technique followed provides a mathematical relationship illustrating the corresponding change in the signal to each case of electromagnetic radiation. This enhances the method of layout routing what is reflected on the signal integrity through the PCB.

Developing systems that have lower levels of electromagnetic interference may not seem complicated, however, it does need $V\ \sqrt{\ }30PG$. The layout starts with $D(\ ) = m\ R$ [3] with the semiconductor components selection, where they produce low levels of radiation. Initial steps in the design process include picking semiconductor elements with a minimal EM radiation output. However, in many instances, other requirements, such as the needed efficiency of the semiconductor part, might be in contradiction to minimal interference. These scenarios may be seen in the electronics industry. The primary responsibility is the development of PCB that does not contain any antennas that emit electromagnetic radiation. Even if this is occasionally possible to do, it is imperative that large signal loops and the ground-return lines that correspond to them that transmit high frequency signals be prevented. In order to realize the benefits of having short interconnecting lines, it is necessary to arrange the electronic components with extreme care.

For our future studies we shall apply the method of shielding signals with ground planes in board stack-up and observe the signal behavior at low and high frequencies, this shall be done through various mediums to measure the amount of change in each case.

REFERENCES

[1] C. M. T. U. S. N. Naoto Oka, "Efficient EMI Reduction in Multilayer PCB Using Novel Wideband Electromagnetic Bandgap Structures," International Journal of RF and Microwave Computer-Aided Engineering, 2011.

[2] Y.-S. L. H.-H. Chou, "Experimental investigation and analysis of EMI in stacked I/O connectors of PCB," *Journal of Electromagnetic Waves and Applications,* 2015.

[3] V. V. Shai Sayfan-Altman, "PCB EMI Simulation," in *IEEE 28-th Convention of Electrical and Electronics Engineers in Israel*, Iseael, 2014.

[4] K. B. M. R. A. M. L. Ž. M. D. M. Kisić, "Performance Analysis of EMI suppressors in Reduction of Ground Bounce on a PCB," in *PROC. 28th INTERNATIONAL CONFERENCE ON MICROELECTRONICS (MIEL 2012)*, SERBIA, 2012.

[5] R. M.-D. M. H.-I. M. V.-M. V. B.-L. P. M.-J. a. A. M.-M. Juan-Jesús Luna-Rodríguez, "Simul-EMI II: An Application to Simulate Electric and Magnetic Phenomena in PCB Designs," in *Trends in Applied Intelligent Systems - 23rd International Conference on Industrial Engineering*, Proceedings, 2010.

[6] A. A. Milad Mehri, "Stochastic EMI Noise Model of PCB Layout for Circuit-Level Analysis of System in IoT Applications," *IEEE,* 2020.

[7] Texas Instruments, PCB Design Guidelines, Texas, US: Texas Instruments Incorporated, 1999.

[8] Geoff Smithson, "Practical RF Printed Circuit Board Design," *IEE,* 2000.

[9] Bruce R. Archambeault, PCB DESIGN FOR REAL-WORLD EMI CONTROL, Research Triangle Park, NC: Kluwer Academic Publishers, 2002.

[10] Stephen C. Thierauf, High-Speed Circuit Board Signal Integrity, Boston • London: Artech House, Inc., 2004.

[11] David L. Jones, PCB Design Tutorial, 2004.

[12] C.-Y. Y. H.-C. L. D.-B. L. H.-P. L. Chih-Hung Lee, "The study of PCB ground area and location on EMI reduction effectiveness," *IEEE,* 2017.

[13] S. K. D. M. Mandeep Kaur, "Electromagnetic interference," *IEEE,* 2011.

[14] D.D.LChung, "Electromagnetic interference shielding effectiveness of carbon materials," *Elsevier Science,* 2001.

[15] O. R. S. Shahparnia, "Electromagnetic interference (EMI) reduction from printed circuit boards (PCB) using electromagnetic bandgap structures," *IEEE,* 1997.

[16] M. B. F. G. C. I. K. S. Grivet-Talocia and P. Kotiranta, "Fast evaluation of electromagnetic interference between antenna and PCB traces for compact mobile devices," *IEEE,* 2008.

[17] P. V. R. M. K. K. S. K. Nageswara Rao, "EMC analysis in PCB designs using an expert system," *IEEE,* 2002.

[18] X.-Q. L. W.-D. X. J.-G. J. M.-X. Z. Q. Y. Q. W. Hui-Sen He, "Efficient EMI reduction in multilayer PCB using novel wideband electromagnetic bandgap structures," *International Journal of RF and Microwave Computer-Aided Engineering,* 2011.

[19] S. C. M. S. H. J. L. D. A. E. R. Bruce Archambeault, "Electromagnetic Radiation Resulting From PCB/High-Density Connector Interfaces," *IEEE,* 2013.

[20] Z. W. C. R. Chen Hanyu, "Electromagnetic Interference Reduction From Printed Circuit Boards," *IEEE,* 2019.

[21] Q. G. X. H. Xiaoqun Liu, "The EMC analysis and design of the PCB," *IEEE,* 2011.

[22] M. L. S. Shantala, "ANALYSIS OF ELECTROMAGNETIC INTERFERENCE RADIATIONS FROM THE PCB TRACES," *IEEE,* 2019.

[23] C. R. Paul, "Modeling electromagnetic interference properties of printed circuit boards," *IBM,* 1989.

[24] M. C. S. S. A. P. V. V. A. M. G. A. F. V. R. M. A. B. Filippo Pellitteri, "A prototypal PCB board for the EMI characterization of SiC-based innovative switching devices," *IEEE,* 2020.

[25] S.-T. C. J.-N. H. Y.-H. L. Tzong-Lin Wu, "Numerical and experimental investigation of radiation caused by the switching noise on the partitioned DC reference planes of high speed digital PCB," *IEEE,* 2004.

[26] P.-A. D. D. R. B.-L. N. Pei Xiao, "A Hybrid Method for Calculating the Coupling to PCB Inside a Nested Shielding Enclosure Based on Electromagnetic Topology," *IEEE,* 2016.

[27] X. C. Tong, Advanced Materials and Design for Electromagnetic Interference Shielding, Boca Raton: Taylor & Francis Group , 2008.

[28] L. Z. B. W. S. L. C. P. Lanyong Zhang, "Hybrid Prediction Method for the Electromagnetic Interference Characteristics of Printed Circuit Boards Based on the Equivalent Dipole Model and the Finite-Difference Time Domain Method," *IEEE,* 2017.

[29] P. S. V Anupama, "Analysis of different techniques for reduction of SI and emission from PCB trace," *IEEE,* 2013.

[30] D. R. P. X. P.-A. D. Yin-Shuang Xiao, "An Equivalent Modeling Method for the Radiated Electromagnetic Interference of PCB Based on Near-field Scanning," *Applied Computational Electromagnetics Society Journal,* 2019.

[31] M. L. B. S. E. M. K. Aygun, "Analysis of PCB level EMI phenomena using an adaptive low-frequency plane wave time domain algorithm," *IEEE,* 2000.

[32] K. P. Y. P. W. K.-N. S. N. K. Aunchaleevarapan, "Classification of PCB configurations from radiated EMI by using neural network," *IEEE,* 2000.

[33] G. L. J. D. Xiaoning Ye, "Investigation of PCB layout parasitics in EMI filtering of I/O lines," *IEEE,* 2002.

[34] M. O. W. K.-n. F. C. C. C. H. G. Kye Yak See, "Impact of PCB layout design on final product's EMI compliance," *IEEE,* 2006.

[35] T. S. Hiroki Funato, "A Study on Correlation Between the PCB Layout and EMI from Chassis," *IEEE,* 2007.

[36] C. L. X. W. M.H. Pong, "EMI due to electric field coupling on PCB," *IEEE,* 2002.

[37] B. G. M. K. Prakash Chand, "PCB layout design for EMI/EMC compliance," *IEEE,* 2006.

[38] S. W. Z. L. Juntao Yao, "Radiated EMI Reduction by Layout Improvement in Power Converters in Automotive Applications," *IEEE,* 2021.

[39] T. H. Hubing, "Printed Circuit Board EMI Source Mechanisms," *IEEE,* 2003.

[40] W.-D. T. C.-H. W. T.-H. Y. T.-H. H. Han-Nien Lin, "Root Cause Analysis and Defect Ground Effect of EMI Problem for Power Electronics," *IEEE,* 2019.

[41] H. W. Shim, Development of Radiated EMI Estimation Algorithms for PCB EMI Expert System, Rolla: University of Missouri, 2004.

[42] Mark I. Montrose, EMC and the Printed Circuit Board: Design, Theory, and Layout Made Simple, Piscataway, NJ: IEEE, 1999.

[43] Y. Han, Interactive PCB Layout Design System for EMC/EMI Minimisation, Nanyang Technological University, School of Electrical and Electronic Engineering, 1997.

[44] R. S. Khandpur, Printed Circuit Boards: Design, Fabrication, Assembly and Testing, New Delhi: Tata McGraw-Hill, 2005.

[45] D. G. William D. Kimmel, Electromagnetic Compatibility in Medical Equipment: A Guide for Designers and Installers, IEEE, 2018.

[46] Martin O 'Hara, EMC at Component and PCB Level, Elsevier, 1998.

[47] C. B. S. C. Bruce Archambeault, "Review of Printed-Circuit-Board Level EMI/EMC Issues and Tools," *IEEE,* 2010.

[48] L. Zhai, Electromagnetic Compatibility of Electric Vehicle, Springer Nature, 2021.

[49] X.-C. Wei, Modeling and Design of Electromagnetic Compatibility for High-Speed Printed Circuit Boards and Packaging, U.S: Taylor & Francis Group, 2017.

[50] M. P. C. L. Z. Q. Wu Xin, "Reduction of EMI by electric field method," *IEEE,* 1999.

[51] J. DeMarinis, "The antenna cable as a source of error in EMI measurements," *IEEE,* 1988.

[52] A. C. S. S. Saswati Ghosh, "Loaded Wire Antenna as EMI Sensor," *PIER,* 2005.

[53] G. C. Godo, Time-Domaint EMI Measurement System for Conducted Emissions Measurements, Universitat

Politècnica de Catalunya. Escola Tècnica Superior d'Enginyeria de Telecomunicació de Barcelona, 2015.

[54] D. Mandaris, High Strength Electromagnetic Field Generation for Radiated EMI Measurements, University of Twente, 2020.

[55] P. S. Crovetti, Electromagnetic Interference and Compatibility, MDPI, 2021.

[56] W. Y. J. S. M. Z. Z. M. Yang Zhao, Electromagnetic Compatibility: Principles and Applications, Springer Nature, 2022.

[57] D. Morgan, A Handbook for EMC Testing and Measurement, IET, 1994.

[58] J. Zheng, Electromagnetic Compatibility (EMC) Design and Test Case Analysis, John Wiley & Sons, 2019.

[59] M. L. Wheeler, Spread Spectrum Receiver Electromagnetic Interference (EMI) Test Guide, National Aeronautics and Space Administration, Marshall Space Flight Center, 1998.

[60] E. M. N. Mark I. Montrose, Testing for EMC Compliance: Approaches and Techniques, John Wiley & Sons, 2004.

[61] D. Weston, Electromagnetic Compatibility: Principles and Applications, Second Edition, Revised and Expanded, CRC Press, 2017.

[62] D. R. J. White, A Handbook Series on Electromagnetic Interference and Compatibility: EMI test methods and procedures, Don White Consultants, 1980.

[63] B. K. Daryl D Gerke, Edn Designers Guide to Electromagnetic Compatibility, Newnes, 2002.

[64] N. Violette, Electromagnetic Compatibility Handbook, Springer, 2013.

[65] O. M. R. C. B. Bruce R. Archambeault, EMI/EMC Computational Modeling Handbook, Springer Science & Business Media, 2013.

[66] K. Malaric, EMI Protection for Communication Systems, Artech House, 2010.

[67] W. R. Free, "Radiated EMI Measurements in Shielded Enclosures," *IEEE,* 2016.

[68] T. D. P. R. Stephan Braun, "A Real-Time Time-Domain EMI Measurement System for Full-Compliance Measurements According to CISPR 16-1-1," *IEEE,* 2008.

[69] Tasuku Takagi, "Research and development on EMC/EMI measurements and technologies in Japan," *Jhon Wiley & Sons ,* 1999.

[70] M. P. S. Ç. M. Ç. F. S. M. A. Azpúrua, "Improving time-domain EMI measurements through digital signal processing," *IEEE,* 2015.

[71] P. R. Arnd Frech, "Real-time ambient noise cancellation for EMI measurements on open area test sites," *IEEE,* 2012.

[72] S. B. P. R. Arnd Frech, "Time-domain EMI measurements in the presence of ambient noise," *IEEE,* 2009.

[73] J. L. D. S. R. J. N. T. H. H. R. E. D. T. V. D. Min Li, "An EMI estimate for shielding-enclosure evaluation," *IEEE,* 2001.

[74] P. R. F. Krug, "Signal processing methods for time domain EMI measurements," *IEEE,* 2003.

[75] M. D. F. W. Tan Yan, "Recent research development in PCB layout," *IEEE,* 2010.

[76] R. Hu, PCB Design and Layout Fundamentals for EMC, Independently Published, 2019.

[77] J. S. Jens Lienig, Fundamentals of Layout Design for Electronic Circuits, Springer Nature, 2020.

[78] M. Tooley, Electronic Circuits: Fundamentals and Applications, Routledge, 2019.

[79] D. Brooks, Signal Integrity Issues and Printed Circuit Board Design, Prentice Hall Professional, 2003.

[80] T. Granberg, Handbook of Digital Techniques for High-speed Design: Design Examples, Signaling and Memory Technologies, Fiber Optics, Modeling and Simulation to Ensure Signal Integrity, Prentice Hall PTR, 2004.

[81] T. Hubing, "PCB EMC design guidelines: a brief annotated list," *IEEE,* 2003.

[82] T. H. Hubing, "Common PCB layout errors that cause products to fail to meet automotive EMC requirements," *IEEE,* 2019.

[83] R. S. B. G. F. H. R. S. M. S. B. R. F. H. I. P. M. E. S. V. K. W. D. R. R. H. P. L. E. S.-S. J. M. F. M. D. J. W. John B. Anderson, "Printed Circuit Board Design Techniques for EMC Compliance," *IEEE,* 1996.

[84] E. B. Fatih Üstüner, "An EMC experimental kit for demonstrating the effect of printed circuit board layout on

the radiated emissions," *IEEE,* 2011.

[85] W. Q. S. C. Li Zhi, "Application of guard traces with vias in the RF PCB layout," *IEEE,* 2002.

[86] R. Perez, "Steps for the proper development of an EMC Control Plan," *IEEE,* 2002.

[87] O. H. Khaled Yazbek, "Practical EMC considerations in designing PCB for RF and microwave communications systems," *IEEE,* 2008.

[88] C. F. B. D. G. D. K. L. M. V. R. Vijay Devabhaktuni, "A New ANN-Based Modeling Approach for Rapid EMI/EMC Analysis of PCB and Shielding Enclosures," *IEEE,* 2012.

[89] J. S. G. F. M. Foissac and C. V. D. Frey, "System simulation for EMC network analysis," *IEEE,* 2011.

[90] S. S. S. Muralikrishna, "An overview of digital circuit design and PCB design guidelines - An EMC perspective," *IEEE,* 2008.

[91] D. V. M. K. Indrek Roasto, "EMC Considerations on PCB Design for a High-Power Converter Control System," *IEEE,* 2007.

[92] M. Z. Jian Cui, "Transient Analysis of PCB EMC Problem," *IEEE,* 2008.

[93] H. W. Ott, Electromagnetic Compatibility Engineering, John Wiley & Sons, 2011.

[94] T. Williams, "A demonstrator for EMC design training," *IET,* 1992.

[95] S. H. Russ, Signal Integrity: Applied Electromagnetics and Professional Practice, Springer Nature, 2022.

[96] F. M. Spartaco Caniggia, Signal Integrity and Radiated Emission of High-Speed Digital Systems, John Wiley & Sons, 2008.

[97] S. C. Thierauf, High-Speed Circuit Board Signal Integrity, Second Edition, Artech House, 2017.

[98] D. Brooks, PCB Signal Integrity, Prentice Hall, 2013.

[99] S. C. Thierauf, Understanding Signal Integrity, Artech House, 2010.

[100] B. A. M. M. Samuel Connor, "The impact of common mode currents on signal integrity and EMI in high-speed differential data links," *IEEE,* 2008.

[101] P. D. T. S. H. C. Mohammad Ali Khorrami, "Employment of microwave absorbers for EMI/RFI mitigations from high speed digital buses with signal integrity considerations," *IEEE,* 2016.

[102] V. Ricchiuti, "Power bus signal integrity improvement and EMI mitigation on multilayer high-speed digital PCBs with embedded capacitance," *IEEE,* 2003.

[103] Z. Z. A. M. Z. M. J. M. A. Alshargabi, "High speed PCB & spiral with patch EBG planar integration for EMI reduction," *IEEE,* 2015.

[104] S. L. C. H. C. H. D. Schröder, Far-field prediction combining simulations with near-field measurements for EMI assessment of PCBs, Croyton, UK: The Institution of Engineering and Technology (IET), 2020.

[105] M. Umekawa, "Simple modeling method of EMI simulation for PCB," *IEEE,* 2017.

[106] H. I. Y. K. M. Y. Ikuo Yoshihara, "Simultaneous Improvement to Signal Integrity and Electromagnetic Interference in High-Speed Transmission Lines," *IEEE,* 2016.

[107] Rick Hartley, "Controlling Radiated EMI Through PCB Stack-up," *Printed Circuit Design Magazine,* 2000.

[108] Y. T. M. N. Kouji ICHIKAWA, "Experimental Verification of Power Supply Noise Modeling for EMI Analysis through On-Board and On-Chip Noise Measurements," *IEICE TRANSACTIONS on Electronics,* 2007.

[109] S. M. G. D. A.K. Ghose, "PCB design with low EMI," *IEEE,* 2002.

[110] L. C. L. T. I. R. J.-M. R. A. Richelli, "Measurements of EMI susceptibility of precision voltage references," *IEEE,* 2017.

[111] C. C. H. Tzu-Chiang Tang, "EMI suppression for high-speed PCB," *IEEE,* 2009.

[112] W. S. J. C. Soonyong Lee, "The Effect of Via Spacing on the Signal Integrity," *IEICE,* 2010.

[113] M. O. H. I. Yoshiki Kayano, "Weak-coupled cross-sectional differential-paired lines with bend discontinuities for SI and EMI performances," *IEEE,* 2014.

[114] D. G. C. M. M. Vikram Jandhyala, "Toward building full-system EMI verification and early design flows through full-wave electromagnetic simulation," *International Journal of RF and Microwave Computer-Aided Engineering,* 2011.

[115] D. H. S. Radu, "An investigation of PCB radiated emissions from simultaneous switching noise," *IEEE,* 1999.

[116] D. H. F. S. J. D. T. H. T. V. D. D.S. Britt, "Effects of gapped groundplanes and guard traces on radiated EMI," *IEEE,* 1997.

[117] M. I. Montrose, Printed Circuit Board Design Techniques for EMC Compliance—A Handbook for Designers, Second Edition, New York, NY: THE INSTITUTE OF ELECTRICAL AND ELECTRONICS, 2000.

[118] E. Bogatin, Signal and Power Integrity - Simplified, Pearson Education, 2010.

[119] Z. Z. H. W. X. L. Lijuan Huang, "Effect of high-frequency PCB laminate on thermal cycling behavior of electronic packaging structures," *Emerald Publishing Limited,* 2022.

[120] R. S. D. R. Y. P. K. Nireeshma Ulkundakar, "Design, Simulation and Fabrication of Non-Spiral Based Fluxgate Sensor on Printed Circuit Board (Pcb)," *MAGMA-D-22-00236,* 2022.