Electromagnetic Interference in Printed Circuit Board Design

SENA YILDIZ
Dept. of Electrical-Electronics Engineering
Suleyman Demirel University
Faculty of Engineering
TURKEY

OZLEM COSKUN
Dept. of Electrical-Electronics Engineering
Suleyman Demirel University
Faculty of Engineering
TURKEY

Abstract: - Today, with the widespread use of electrical and electronic systems, it has become important that they can work without interacting with each other. Electronic components and transmission lines generate energy at the radio frequency level because they carry current. In addition, disturbing noises from the mains line or from the air can have a disruptive effect on electronic systems. Electromagnetic compatibility is the ability of the system to work in an electromagnetic environment without creating any noise at the electromagnetic energy level on other systems or devices around it. Electromagnetic interference can be defined as any kind of undesirable disturbance or disturbance frequency effect that occurs as a result of violation of electromagnetic compatibility standards. This can lead to a decrease in the performance of electronic devices or even deterioration. In this study, electromagnetic interference and electromagnetic compatibility issues in printed circuit board are investigated. In addition, solution methods are also presented within the scope of this study.

Key-Words: Electromagnetic Interference, Electromagnetic Compatibility, Electromagnetic Interference Prevention Approaches
Received: April 11, 2022. Revised: February 21, 2023. Accepted: March 24, 2023. Published: May 3, 2023.

1 Introduction

Electromagnetic Compatibility (EMC) standards have been among the mandatory requirements in European Community countries since 1996. Accordingly, no electrical/electronic device should cause electromagnetic emission that will affect the operation of other devices through electromagnetic interference, and it should not be affected by electromagnetic interference (EMI) itself [1].

There are three main application areas of electromagnetic interference prevention techniques, which are applied to prevent problems that may occur on devices as a result of EMI. These; printed circuit board (Printed Circuit Board - PCB), circuits-filters and shielding [2].

PCB can be seen as an internal line of defense. Circuits on the PCB are where electromagnetic interference problems begin and end. As another line of defense, filters and special circuits are used around the printed circuit boards. Shielding, on the other hand, is the outer line of defense and covers the cables and twisted places. Shielding is the basic step in realizing the electromagnetic compatibility of active and passive devices and is one of the most effective methods of preventing electromagnetic interference.

In this study, electromagnetic interference is discussed for PCB designs. Schematic design, PCB layout and line drawings should be planned according to EMC standards [3,4]. Therefore, these standards form the basis of the design.

EMC is the ability of a device, equipment or system to perform its expected functions in its electromagnetic environment, without creating electromagnetic noise at levels that will disturb this environment or other equipment, and without being affected by the interference of other systems in the environment.

First of all, it is necessary to identify all functional groups in the design. High frequency lines take priority. For this reason, these lines should be positioned and directed first. Then the supply voltages and finally the analog circuits and low frequency signal circuits should be positioned and drawn [4,5,6].

2 Electromagnetics

Maxwell proved that electricity and magnetism, which are thought to be two separate structures, are two separate components of a single structure and showed it with his field equations. Maxwell not only developed the basic equations of electromagnetism, but also used these equations to predict the existence of electromagnetic waves and to show that light is an electromagnetic wave [7].

Before the synthesis of the mathematical structure of Maxwell's equations, the basic laws of the fields of electricity and magnetism were Coulomb's law for the electric field E formed by a fixed point charge q, and the Biot-Savart law for the B magnetic field produced by a wire carrying a current i [8].

The laws of electromagnetism in terms of the state they took after Maxwell's Analysis required that these electric and magnetic fields provide a wave equation in empty space [9] and magnetic fields can propagate through space at the speed of light. The radiating fields E and B are therefore solutions to Maxwell's equations. Figure 2.1 shows the directions of these areas relative to each other (they are perpendicular to each other) and the direction of propagation [7].

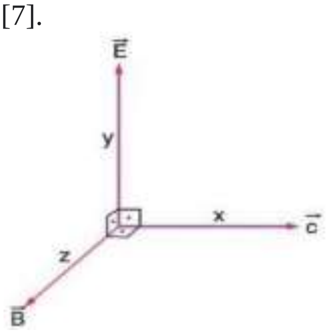

Fig. 1. Fields E and B perpendicular to each other and to the direction of propagation c

Fig 2. shows the array of propagating fields like a sinusoid of fields E and B, giving the wavelength and the direction of motion of the waves.

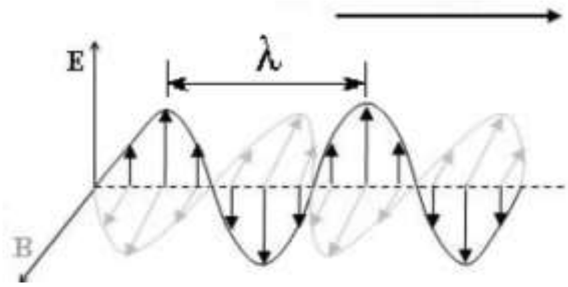

Fig. 2. Fields E and B perpendicular to each other and to the direction of propagation c

Maxwell used the symbols B for magnetism and E for electricity. He also used the symbols $-\frac{\partial}{\partial t}$ for the rate of increase and decrease of something and ∇x for the amount. p corresponds to free charge density and J to current density. dA represents the area element and dS the length element.

Table 1. Maxwell's Equations

$\nabla . D = p_v$	Gauss's law
$\nabla . B = 0$	Gauss's law for magnetism
$\nabla x E = -\frac{\partial B}{\partial t} - J_m$	Faraday's law
$\nabla x H = J_e + \frac{\partial D}{\partial t}$	Ampere's law

Gauss's law is a generalization of Coulomb's inverse square law for point charges and explains how fields are formed from charges. Gauss's Law for magnetism states that there are no magnetic point charges and sources. There is no such thing as a magnetic charge in a magnetic field. Therefore, magnetic field lines never intersect or terminate and always form closed loops.

Faraday's Law of Induction (Maxwell–Faraday equations) explains that the amount of electricity produced by magnetism will be equal to the rate at which the magnetic force increases or decreases. A very rapidly changing magnetic force produces a large amount of electricity, while a slowly changing magnetic force produces a very small amount of electricity. A magnetic force that remains constant over time produces no electricity. Ampère's law describes how fields move around their source. It states that the magnetic field circulates around time-varying magnetic fields according to Ampère's law with electric currents and Maxwell's correction [7].

3 Electromagnetic Interference in PCB Design

Electronic components and transmission lines generate energy at the radio frequency level because they carry current. EMC is the ability of the system to operate in an electromagnetic environment without creating any noise at the electromagnetic energy level on other systems or devices around it. EMI, on the other hand, is the unwanted interference frequency effect that occurs as a result of the violation of electromagnetic compatibility standards [3].

Schematic design, PCB layout and line drawings should be planned according to EMC standards (Yazıcı and Çetinkaya, 2021; Montrose et al., 2004). Therefore, these standards form the basis of the design. First of all, it is necessary to identify all functional groups in the design. High frequency lines take priority. For this reason, these lines should be positioned and directed first. Then the supply voltages and finally the analog circuits and low frequency signal circuits should be positioned and drawn [4,5,6].

3.1 Prevention of Electromagnetic Interference Effect

Electromagnetic interference must be prevented on a PCB for two main reasons. The first reason is to ensure signal integrity. Low noise is the result of signal degradation along a transmission path. The other reason is external interactions related to emissions testing of a product. Electromagnetic interference emissions are caused by harmonics of clock signals or other periodic signals [10,5,11].

The copper tracks on the PCB can act as high-frequency antennas depending on the current carried. As a result, PCB antennas are obtained by calculating the dimensions in a specific way and forming the copper layer in the form of a polygon. Communication lines such as SPI, RS485, CANBUS, UART, USART are high-frequency and the risk of EMI increases as the speed increases. These high-frequency communication circuit sections contain receiver (RX) and transmit (TX) lines.

Microcontrollers are a controller structure consisting of a large number of embedded transistors. For this reason, especially the reset pin and communication pins may be affected by electromagnetic noises. It may prevent the card from working in the desired direction. Circuit segments at high frequencies can affect each other. In case of a PCB antenna that provides wireless communication of the circuit, it should be protected by taking extra precautions against electromagnetic interference.

3.2 Electromagnetic Interference Interfaces

The points where an electronic device is exposed to interference are grouped into four groups.

- Radiated Emission (RE)
- Radiated Susceptibility - RS
- Conducted Emission (CE)
- Conducted Susceptibility (CS)

3.3 PCB Design

A PCB should be divided into functional groups: analog, digital, power supply, high noise, and input/output (IO) groups. Conductor traces of one group should not cross into a different group unless there is an interface trace between the groups. Fig. 3 shows exemplary placement of functional groups [12].

Fig. 3. Division of PCB into functional groups

High frequency circuits producing signals leaving the PCB should be placed close to the IO area as in Figure 2.2 to minimize loop areas on the PCB.

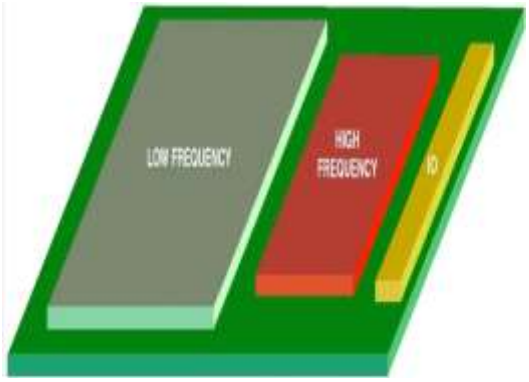

Fig. 4. The order when the high-frequency signal exits the PCB

To minimize electromagnetic interference to IO interconnects, circuits that generate high-frequency signals that do not leave the PCB are placed away from the IO connectors as in Fig. 5.

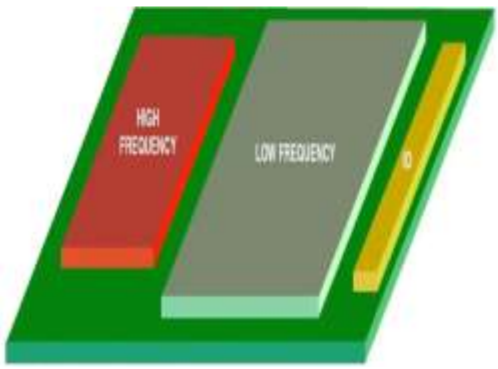

Fig. 5. The order it should be when the high frequency signal stays on the PCB

3.4 Circuit Elements in terms of EMC Emission

Coupling describes the pathways electromagnetic interference uses to reach electronic systems. Electromagnetic energy transfer can be via spatial radiation or electrical transmission.

Two circuit paths with a certain distance between them and current flowing through them may exhibit capacitor behavior, causing uncontrolled electric fields to form on the circuit or voltage injection to the components. Such situations are called capacitive coupling and these resulting elements are called parasitic capacitance [13].

A wire with alternating current flowing at different frequencies can exhibit the behavior of a coil. Any path longer than 1/20 of the wavelength of the signal can exhibit antenna behavior and induce voltage to components around it. Interferences that occur in this way are called inductive coupling. These pathways that exhibit antenna behavior are called parasitic inductances [4,13].

3.4.1 Cables

The characteristic of cables in the radiative propagation band (30 MHz-40 GHz) and partly in the conductive propagation band (150 KHz-30 MHz) is often far from their ideal characteristics. At these frequencies, cable inductance becomes important. In parallel lines, inter-cable capacitance gains importance.

3.4.2 Printed Circuit Elements

Coupling between lines between circuit elements is a major source of problems. Numerical methods are used for coupling modelling. Examples of these are the method of moments (Method of Moments - MoM), the finite element method (Finite Element Method - FEM), the finite difference method (Finite Difference - FD) and the finite difference in time method (Finite Difference Time Domain - FDTD).

Resistors, which are one of the most used elements among circuit elements, may move away from the ideal at high frequencies and their frequency-dependent characteristics may change depending on the construction technique (carbon, thin film, etc.). Capacitors have frequency dependent characteristic.

In addition, connection points, electromechanical devices, digital circuit elements, mechanical switches and solenoids cause electromagnetic emission that will affect the operation of other devices [1].

3.5 PCB Assembly in Terms of EMC Emission

Four types of mounting can be made on the PCB. These mounting types vary in terms of EMC emission.With single-sided surface-mount technology (SMT) mounting, surface-mount components are located on only one side or component side of the boards. Because there are no through-hole components, these boards do not need drill holes. Fig. 6 shows single surface-mounted components.

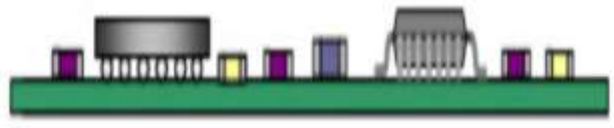

Fig. 6. Surface-mounted components with one-sided mounting

There are both through-hole and surface-mount component bodies on the component side, with single-sided mixed SMT and Through Hole Technology (THT) assembly. Despite being called single-sided, the PCB needs to be double-sided as the through-hole pads have to be on the solder side and the component bodies on the opposite side. Fig. 7. shows one-sided mixed surface-mounted components [14].

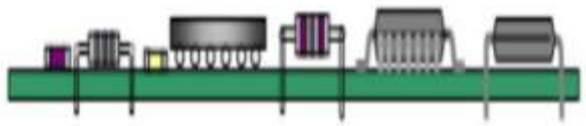

Fig. 7. Surface-mounted components with one-sided mounting

Simple and compact, double-sided SMT cards use both sides to make the most of available space and ensure rigid form factors are met. There is no need for plated holes here. Figure 2.6 shows double-sided surface-mounted components.

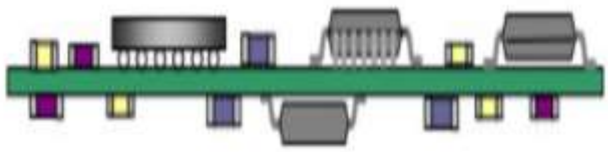

Fig. 8. Surface-mounted components with double-sided mounting

The biggest advantage of wave solder double-sided mixed assembly with THT and SMT on both sides is to maximize the SMT area of the gaps between the long component leads on the solder side. Other than this, it is similar to a one-sided mixed assembly. Fig. 9. shows double-sided mixed surface-mounted components [14].

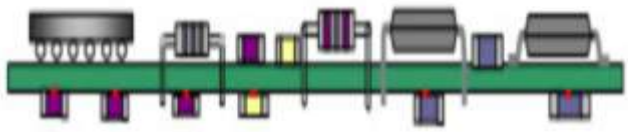

Fig. 9. Surface-mounted components with double-sided mixed mounting

In such multilayer PCBs, the power and ground surfaces act as radiating microstrip patch antennas. Radiation or propagation here is caused by fringe electric fields on the card edges and creates noise on the card.

3.6 Source of Noise and Switching Noise

Switching noise is often caused by the high-speed time-varying currents required by high-performance digital circuits. The flow of these currents through paths between the layers of a PCB causes radiation. Propagating waves use parallel plates formed by power planes [15,16].

Simultaneous switching noise (SSN) is an inductive noise generated when several outputs of a digital circuit are switched simultaneously. SSN cannot be measured precisely because it depends on the geometry of the board and current paths, and various studies have focused on modeling this phenomenon [17,18,19]. SSN status can be described by the following formula:

$$V_{n=}NxL_{eq}\frac{di}{dt} \qquad (1)$$

Where V_n is the magnitude of the noise voltage, N is the number of simultaneously switched outputs (drivers), L_{eq} is the equivalent inductance through which current must flow, and i is the current through each driver during switching [20].

When several signals are switched simultaneously, the power planes connected to the power supply must provide the necessary current that must pass through Leq. The presence of inductance in the current path causes voltage fluctuations in the power planes. This affects the signals at the outputs of the drivers. It also causes faults and incorrect switching across the board. This type of noise, called delta-I noise or ground spatter, is considered a fundamental and critical issue in the design of high-speed PCBs [19].

It is important to terminate the journey of the signal flowing to earth in the shortest distance to complete its circuit. Current loops and magnetic fields occur in areas surrounded by lines carrying power or data signals around the PCB [20].

4 Solution Methods

The healthiest method to reduce the risk of EMI is to design a PCB with 4 or more layers according to the variety of voltage levels [21]. From the layers, the upper layer and the lower layer should be used for all signals, one of the inner layers should be used only for the ground (Ground - GND) and the other inner layer should be used only for the supply voltage levels. In this way, the possibility of coupling noise to the supply voltages of the signal paths is reduced [22].

A PCB design with four or more layers (6, 8, 10, 12, 14, 16, 18, 20) costs more than a two layer PCB design. For this reason, EMI measures can be taken with different methods on cards that do not have many high-frequency lines.

The most basic known application is to reduce the ground inductance by expanding the ground areas in the PCB [23,24]. When the ground terminals of the signals are connected to each other and flow to the ground over a single or several lines, it creates high ground inductance and causes interference, which is a poor design [25]. Each signal must be connected directly to the large earth polygon. The ground polygon should also be checked, the looped parts may also act as antennas and affect the working status of some circuit parts. In the connection of shielded communication signals, the RX and TX lines should be very close to each other and their starting points should go to the terminal to which they will be connected by the shortest route. There should be no other high-frequency lines on the route of these lines. The bottom of the microcontroller should be set as a homogeneous ground plane. Capacitors should be positioned correctly around the microcontroller that will drain the unwanted electromagnetic energy to the ground. Recommended capacitor values are available in the reference designs in the technical documentation.

The shape of the conduction paths on the PCB can also cause electromagnetic interference; Where the direction of the transmission path is to change, 45° transitions should be used rather than a 90° transition. On the PCB, the paths from the top layer and the bottom layer should not be drawn parallel to each other, this will cause interference. They should be drawn perpendicular to each other [26,27,28]. The PCB should be separated for different voltage regions in order to avoid crosstalk, conflict and interaction between different supply voltages. Commonly used voltage levels on PCBs are 24V, 15V, 5V, 3.3V, 1.1V.

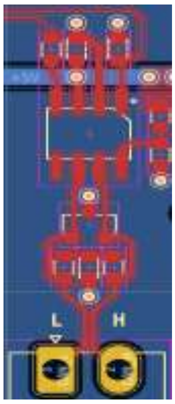

Fig.10. A CAN-BUS interface circuit conforming to EMC standards

The circuit is designed as two layers in terms of cost-effectiveness and the lower layer is determined as GND, keeping the ground area wide, the ground terminals of the components are directly connected to the ground via vias. The CANBUS transmission lines CANL and CANH are differentially plotted.

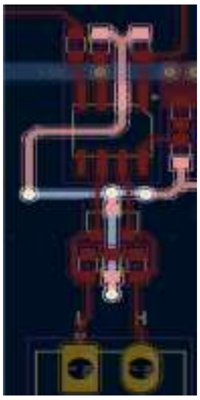

Fig. 11. A CAN-BUS interface circuit that does not comply with EMC standards

In the figure, the ground terminals of the components are not directly supplied to a low impedance ground area, but are formed by jumping over a single or several lines and connecting to each other. In this way, the ground path will create an antenna effect and distort the signal carried on the CANL and CANH lines.

Fig. 12. Lower and upper layer transmission paths running parallel to each other

As seen in the figure, if there are signal paths that need to progress from the lower layer, these paths should be drawn perpendicular to the ones in the upper layer. The design of the output line signal paths in the figure is wrong, the information may distort each other.

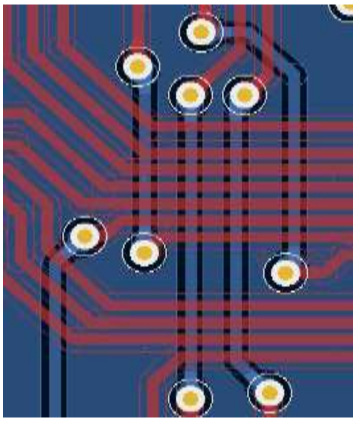

Fig. 13. Upper and lower layer transmission paths running perpendicular to each other

The paths in the upper and lower layers in the figure are drawn perpendicular to each other, minimizing the shortest distance from each other.

4.1 EMI Prevention Approaches on PCB

Before starting the PCB design, it should be considered how the different circuit topologies of the system should be determined and how the circuit elements should be positioned on the PCB. This process is called Partitioning.

Electrostatic Discharge (ESD) is a temporary discharge of charge that occurs when static charges accumulated on a person or an object encounter a grounded surface. Even if they are low energy, they can easily match the capacitances and inductances in electronic systems through the signals of different frequencies and amplitudes they contain. Precautions should be taken while designing, as these matches may cause unforeseen malfunctions [29].

Circuit elements, in the circuit diagram;

• Analog circuit groups (Opamps, Comparators etc.)
• Digital Circuit Groups (Clock Pulse, PLL, I2C, SPI, UART, CAN lines etc.)
• Driver Circuits (Mosfet, IGBT driver ICs, Snubber circuits etc.),
• Isolation Circuits (Optical Couples, Isolation Transformers etc.),
• Power Electronics Circuits (Buck, Boost, Buck – Boost Converters etc.),

Considering the physical conditions of the system, they should be placed at points where they will not act as noise sources.

The thermoelectric effect is the direct conversion of heat into electrical energy or vice versa. Voltage occurs when a temperature is different on each side of a thermoelectric device. On the contrary, when voltage is applied to a device, a temperature difference occurs. The temperature gradient applied at the atomic scale causes the charges in the material to spread from the hot side to the cold side. The Seebeck effect is the direct conversion of temperature differences into electricity.

For this reason, the data power transmission paths on the PCB should be drawn with the least amount of resistance to electron flow. Wide-angle curves should be preferred at the points where the lines turn. In this way, the thermoelectric effect of the heat generated during the passage of electron flow through narrow sections on the PCB with different types of metals, in other words, the Seebeck effect is prevented [29].

PCB lines carrying high current should be kept as wide as possible. A conductor with a narrow cross-section has a higher resistance to current than a conductor with a large cross-section. Therefore, a lot of heat energy emerges on it. The impedance of the circuit paths carrying out data transmission should be equalized as much as possible. If the impedance of the data transmission lines is equalized, the electromagnetic fields on the lines will be of equal intensity and the possibility of cross talk problems will decrease [13].

In order to minimize current loops, ground planes are used in the design of the printed circuit board. Suitable spaces on the PCB are filled with copper surfaces consisting of one piece. Ground line connections of circuit elements are placed at the closest points to these planes.

The parts with high-frequency broadcasting circuits in the circuit design can be used to prevent in-circuit interferences by means of Faraday Cages specially designed for PCB applications [13].

A Faraday cage is created by grounding the entire periphery of the PCB and not sending any signals beyond this limit. This mechanism limits the emission and interference from PCB to another PCB inside and outside the boundary defined by the cage.

The sheath types of the components used are also effective in preventing electromagnetic interference in the PCB. For example, surface mount technology (SMD) type surface mount components are close to the surface, making it easier to transfer electromagnetic interference to the ground line plane. Since the majority of SMD circuit elements contain ceramics, they provide good EMI isolation. The amount of parasitic capacitance and inductance is low. Although they have various advantages, the majority of SMD components cannot be used in high power applications [29,13,30]
.

4.2 Performing Electromagnetic Analysis Using MATLAB

Using MATLAB, the RF PCB Toolbox™, high-frequency PCBs can be designed, analyzed and prototyped. RF PCB Toolbox™ accurately calculates S parameters, impedance, current and load distribution with full wave 3D electromagnetic analysis using components such as Transmission lines, couplers, filters, inductors and capacitors. It allows the use of behavioral models to accelerate the analysis of traces, coils and capacitors. RF PCB Toolbox™ is used to determine metal and dielectric material properties, analyze the effect of thickness, conductivity, permeability, and loss tangent on PCB performance, etc. Helpful for situations.

Performing PCB electromagnetic analysis using the RF PCB Toolbox™ is shown in Fig. 14.

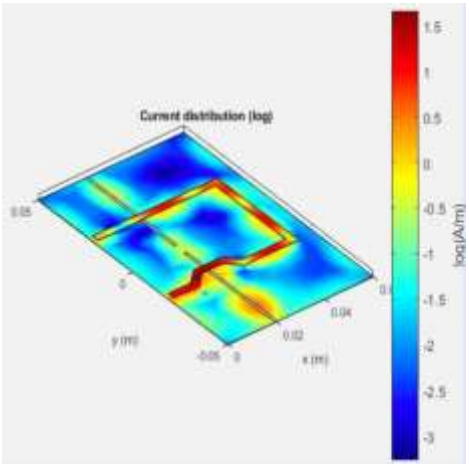

Fig. 14. PCB electromagnetic analysis using MATLAB

4.3 Performing Electromagnetic Analysis Using

EMPro

One of the challenging aspects of PCB design is minimizing EMI effects and providing EMC. EMPro allows to simulate electromagnetic emissions from the PCB at different frequencies as well as to simulate the sensitivity of the design to external sources of electromagnetic emissions. EMPro makes it possible to analyze the effect of placing the PCB design inside different metal enclosures.

Fig. 15 shows the use of electromagnetic simulation technology in EMPro to analyze the EMI and EMC of a PCB inside a metal enclosure.

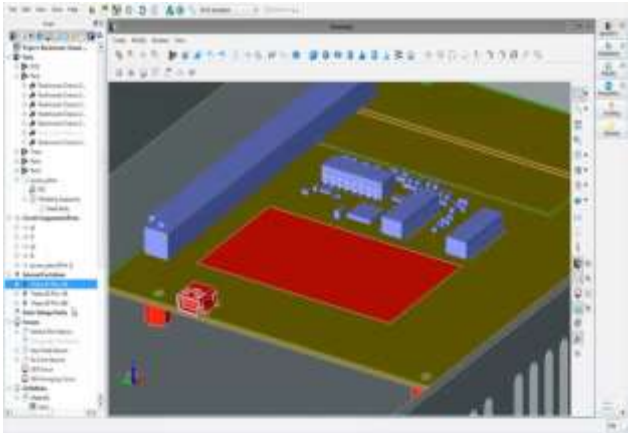

Fig. 15. PCB electromagnetic analysis using EMPro

5 Conclusions

The purpose of this study is to prepare a guide for the design of all kinds of PCBs and wiring with reduced EMI and high-frequency circuit layer. In the study, some of the design rules have been proven experimentally and some of them theoretically. Important points of electromagnetic theory related to EMI and EMI-based design are mentioned. Such theoretical knowledge of electromagnetic field theory should be considered before the design procedure is established, some of which should be verified with theoretical knowledge. Basically, design procedures are defined such as which functional blocks should be placed in which place on a PCB.

Some restrictions on overlapping and interconnecting different circuits are noted to avoid EMI radiation. Sometimes you can use a circuit as ground plane etc. should be distinguished from the others.

Restrictions on PCB orientation, layout of circuits on the PCB, and must-haves for reduced EMI are specified. If these are not taken care of in the design, unpredictable errors may occur in the system and it may not be understood where the problem originates. In addition, the EDA program to be used as PCB Layout in electronic design is also important. For example, while KiCAD software does not have a tool for electromagnetic compatibility, Altium Designer and EMPro guide the designer by simulating the parts that will create an antenna effect on the card.

References:

[1]Dural, G., EMC/EMI", METU EEMB, 2008.

[2]Johnson, HW., Graham, M., High-speed digital design, Prentice Hall PTR, pp.20-23, 1993.

[3]Yazici, Osman., Çetinkaya, Ö., EMI and EMC in high speed electronic circuit boards, 10th International Scientific Research Congress, 2021.

[4]Montrose, Mark I., 2004 EMC and the printed circuit board: design, theory, and layout made simple, John Wiley & Sons, 2004.

[5]Ogut, A., PCB design and ınternal wiring with reduced emı (electromagnetic ınterference) for flat tv applications, Phd Thesis. DEU Institute of Science and Technology, 2009.

[6]Archambeault, Bruce R., Drewniak, James., PCB design for real-world emi control, Springer Science & Business Media, 2013.

[7]Uçar, S., An evaluation of maxwell's area equations. Philosophy Archive 51, 261-270 2019.

[8]Cushing, J., 2006. T. philosophical concepts in physics. Translated by Ozgur Sarioglu. Istanbul: Sabanci Uni. Publications, 2006.

[9]Dereli, T., Verçin A., Quantum mechanics. Ankara Turkish Academy of Science, 2014.

[10]Saracoglu, OG., filter design for the prevention of electromagnetic ınterference at the printed circuit level, 2022.

[11]Karaman, AB., Experimental Investigation of Electromagnetic Interference Behaviors of PCB

Coolers According to Feeding Point in 1-10 GHz Band, Gazi University Faculty of Engineering and Architecture Journal, 38.2: 733-742, 2022.

[12]NXP Semiconductors. A guide to designing for ESD and EMC (AN10897), 2010.

[13]Henry, WO., John Wiley & Sons, 2009. “Electromagnetic Compatibility Engineering, New Jersey, 2009.

[14] ahmetturanalgin. PCB layout and layout. https://www.ahmetturanalgin.com/pcb-yerlesim-ve-duzeni-1 (Available online: 2023).

[15]Van den Berghe., S., Olyslager, F., Zutter, De D., Moerloose, De J., and Temmerman, W., Study of the ground bounce caused by power plane resonances, IEEE Trans. Electromagn. Compat., vol. 40, p. 111–119, 1998.

[16] Leone, M., The radiation of a rectangular power bus structure at multiple cavity-mode resonances, IEEE Trans. Electromagn. Compat., vol. 45, p. 486–492, 2003.

[17]Chun, S., Swaminathan, M., Smith, L. D., Srinivasan, J., Jin, Z., & Iyer, MK., Physics based modeling of simultaneous switching noise in high speed systems, in Proc. 50th Electronic Components Technology Conf., Las Vegas, NV, pp. 760–768, 2000.

[18]Chun, S., Swaminathan, M., Smith, LD., Srinivasan, J.,nJin, Z., and Iyer, MK., Modeling of simultaneous switching noise in high speed systems, IEEE Trans. Adv. Packag., vol. 24, p. 132–142, 2001.

[19]Ramahi, O. M., Subramanian, V., and Archambeault, B., Simple and efficient finite-difference frequency-domain algorithm for study and analysis of power plane resonance and simultaneous switching noise in printed circuit boards and chip packages, IEEE Trans. Adv. Packag., vol. 26, p. 191–198, 2013.

[20]Shahparnia, S., Ramahi, Omar M., Electromagnetic interference (EMI) reduction from printed circuit boards (PCB) using electromagnetic bandgap structures, IEEE Transactions on Electromagnetic Compatibility, 46.4: 580-587, 2004.

[21]Paul, CR., Scully, RC., Steffka, Mark., Introduction to Electromagnetic Compatibility, John Wiley & Sons, 2022.

[22]Mercer, Sean, Eng, C., Minimizing RF PCB electromagnetic emissions, RF DESIGN, 22: 46-85, 1999.

[23]Ott, Henry W., Noise reduction techniques in electronic systems, New York: Wiley, 1988.

[24]Walter, R. L., Identifying EMI parameters on printed circuit boards, In: Proc. EMC EXPO 86. P. T18. 12-16, 1986.

[25]Taki, M., Identification and simulation of critical interconnect paths with respect to transient noise on PCB-level, PhD Thesis, 2008.

[26]Brooks, D., Signal integrity issues and printed circuit board design, Prentice Hall Professional, 2023.

[27]Ardizzoni, Jo., A practical guide to high-speed printed-circuit-board layout, Analog Dialogue, 39.9: 1-6, 2005.

[28]Shahparnia, S., & Ramahi, OM., Electromagnetic interference (EMI) reduction from printed circuit boards (PCB) using electromagnetic bandgap structures. IEEE Transactions on Electromagnetic Compatibility, 46(4), 580-587, 2004.

[29]EMC/EMI noise reduction and protection, shielding, grounding and pcb design, Translation: Cigdem Osşar, Aydın Bodur, TMMOB Chamber of Electrical Engineers Publications, 1st Edition, 2008.

[30]Mark, I. M., Edward, MN., Testing for EMC compliance - approaches and techniques, IEEE, 2004.