1. Introduction

All electrical, electronic and electromechanical devices emit electromagnetic signals of varying amplitude and wavelengths to the environment when energized.

EMI (Electromagnetic Interference): It is the emission of an electromagnetic signal emitted by a device or system that creates a parasitic effect on other devices. It is also called RFI (radio frequency initiative).

EMC (Electromagnetic Compatibility): The ability of a device or system to operate without being affected by the electromagnetic interference (EMI) of other devices and not to generate EMI at levels that cannot be tolerated in its electromagnetic environment (EME).

In short, it is important that the devices and systems to be designed do not go beyond the EMC limits for the long-lasting and healthy operation of both the produced device and the other devices in the environment.

2. Electromagnetic Field

There are many ways to reduce the electromagnetic interference and sensitivity of a device or system to acceptable levels. Before writing these methods into ingredients, let's consider some of the basic issues that cause these effects.

The electromagnetic field is the basis of electromagnetic interference. According to Maxwell's law$\left(\nabla\, x\, B = \mu_0 J - \mu_0 \varepsilon_0 \frac{\partial E}{\partial t}\right)$, an electric field that changes over time causes a magnetic field. Therefore, instead of mentioning electric and magnetic fields separately, often these two are described together as "electromagnetic fields". So, we can think of the electromagnetic field as a combination of electric and magnetic fields. If we need to briefly recall some information;

Electric Field: When a device or system is energized, it generates electric fields in the surrounding environment. Even if the current does not flow, electric fields will occur because of the potential difference between the environment and other devices due to the polarization of the electrical charges.

Magnetic Field: The higher the amplitude and frequency value of the current flowing through the conductors, the higher the EMI value, and the harder it is to stay within EMC limits.

Nearly all PCB-level EMI protection methods are based on factors affecting these two types of fields. For example, basically the intensity of these two types of fields is inversely proportional to the distance between sources. Therefore, the inference that "sensitive transmission paths with low amplitude should not be placed between the circuit paths that will create a high potential difference in PCB design or near the switched circuit paths with high current amplitude" will be correct ***(Figure 4).*** Or if we remember that sharp corners create the leaking fields and fluxes, the inference that in order to the PCB trace to be less affected by itself, to avoid increased sensitivity to surrounding traces and for lower EMI generation in the environment the sharp corners on the PCB trace need to be rounded" will also be correct (***Figure 3***).

3. Importance of EMC Qualifications

Electromagnetic Compatibility (EMC) is the control of electromagnetic interference (EMI) emitted as a signal or transmitted in some way. An inadequate EMC is one of the main reasons that requires a redesign of the PCB. An estimated 50% of the first attempts of each PCB design fail for some reason, because they emit and/or susceptible to unwanted EMI.

The point at which PCB designers make the most mistakes about EMC: Designers of new products which has wireless connections nowadays, do not take into consideration that they are already using an antenna and thus will be sensitive to all kinds of interference in the environment and spread the harmonic parasites in the environment. Even if the products they design do not have wireless connectivity, it is important to note that almost all of the other devices in the working environment of any product have such features!

EMC is an element that must be considered from the beginning of PCB design. EMC deficiency can lead to a decline in the performance of the product and/or surrounding products, leading to the loss of reputation of the brand/product, or even a costly road leading to the product being withdrawn from the market and the design being changed from the very beginning. Therefore, in PCB designs, an acceptable cost budget is reserved for EMC at first and thus the costs that may occur later are avoided.

4. PCB-level EMI Prevention Methods

In each PCB design, attempting to implement all the EMC rules and following an overprotected strategy can lead to long delays in projects, shrinkage in the selection of materials, and increased material costs. The rules should be evaluated separately according to whether they apply to the project we are working on.

1- In PCB design, physical properties of traces and components should be considered (***Figure 1***).

   a. The supply inputs of the ICS to be used must be filtered and this filter must be as close to the supply inputs as possible (especially sensitive ICS operating at low voltage, such as the microcontroller).
   b. Traces should be kept as short and thin as possible, both to prevent interference impedances and to reduce EMI sensitivity.
   c. Central systems with dense CLK signals should be kept as close as possible to the connector where sensitive (low-amplitude, high-frequency or low-energy) signals will input and output. If a CLK circuit does not have input and output to the external environment, it must be kept in the center of the PCB.
   d. In high frequency circuit designs, extra emphasis should be placed on length and thickness of traces, and in PCB design, it should be noted that the regions where these signals are located do not stay between the two connectors.
   e. GND voltage must be applied to the external protection of the oscillator and crystals to be used.
   f. Each signal path containing a high frequency (e.g., oscillator signal output, communication input outputs such as USART, I2C, SPI) and/or weak analog signal paths should be kept as short as possible.
   g. Every signal that enters the PCB must be filtered.
   h. The vias to the high and low voltage layers of the filter (Decoupling) capacitors should be as close together as possible.
   i. Filter capacitors should be installed in the remaining part of the PCB area (if not to be cut) to prevent fluctuations in high and low voltage layers.

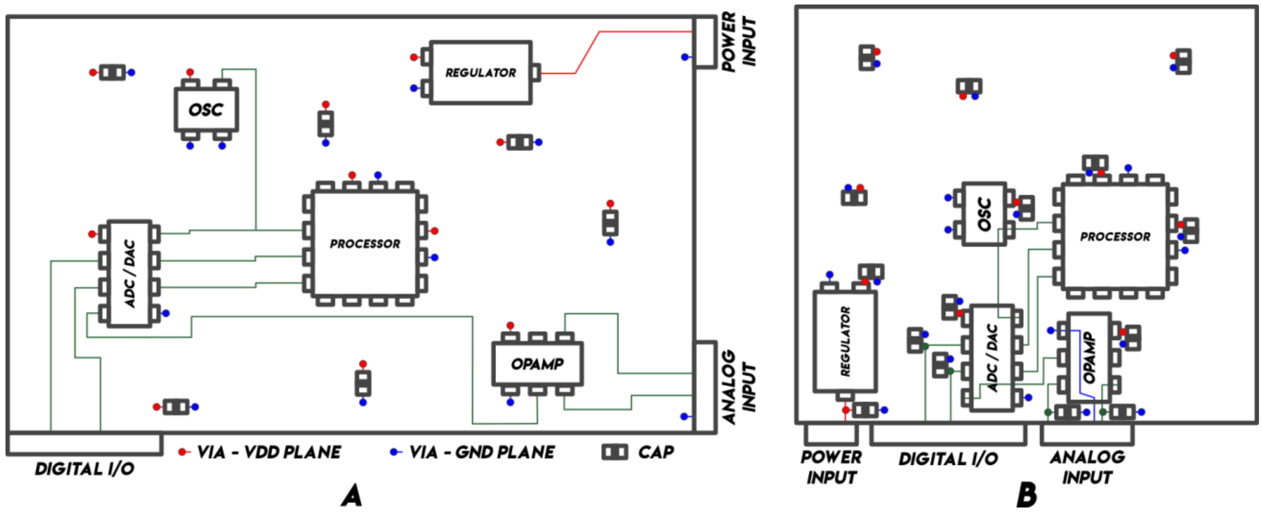

Figure 1: ***(A)*** *4-layer PCB design,* ***(B)****PCB design in accordance with 4-layer EMC rules.*

2- Attention should be paid to switched traces. (***Figure 2)***

a. The rise (t_R) and fall(t_F) times of the switching waves should be kept as high as possible. This prevents high radiation from sudden voltage and current changes. For similar reasons, attention should be paid to the peaks of the switching waves.

b. High-amplitude current switches should be designed with the appropriate charge/discharge capability of the PCB design to prevent sudden fluctuations in the supply current.

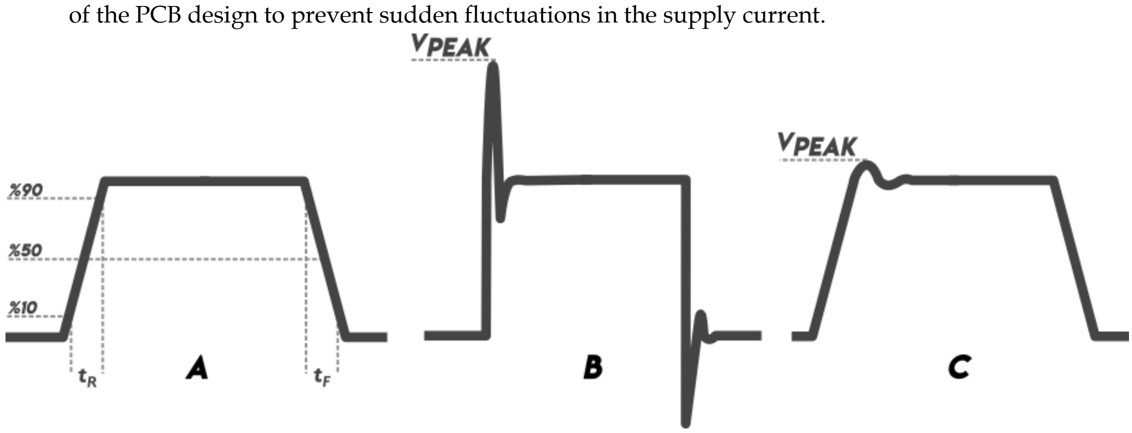

Figure 2: ***(A)*** *Illustration of rise and fall times,* ***(B)*** *square wave, which can cause sudden radiation fluctuations in Switched applications,* ***(C)*** *square wave, which will ensure that the radiation fluctuation is minimal.*

3- The sharp corners on the traces need to be rounded. (***Figure 3)***

Figure 3: *Compliance of transmission lines to EMC, respectively.*

4- Low-amplitude sensitive traces should not be placed between circuit paths that will generate high potential difference or switched circuit paths with high current amplitude. (*Figure 4*)

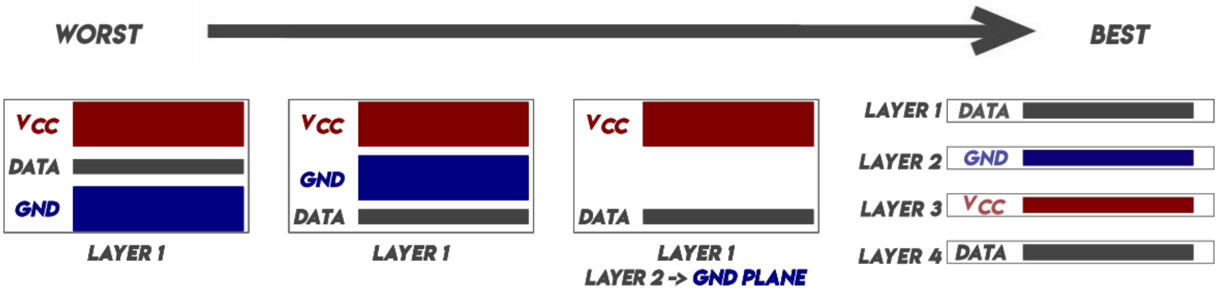

Figure 4: *Placement of sensitive (low-amplitude, high-frequency, or low-energy) traces, respectively.*

5- The GND layer should not be strictly divided. Otherwise, the potential difference will occur from the EMI and/or fluctuations in the area on the PCB. This will result in noise, loss of sensitivity and shortening of product life. In applications where the supply voltage is met from a different PCB, if the distance is far, we can use two of the flat cables as GND instead of one to reduce conduction losses. This may vary from application to application. However, the truth is, in PCB designs, the return routes (GND) should never be broken up. (*Figure 5*)

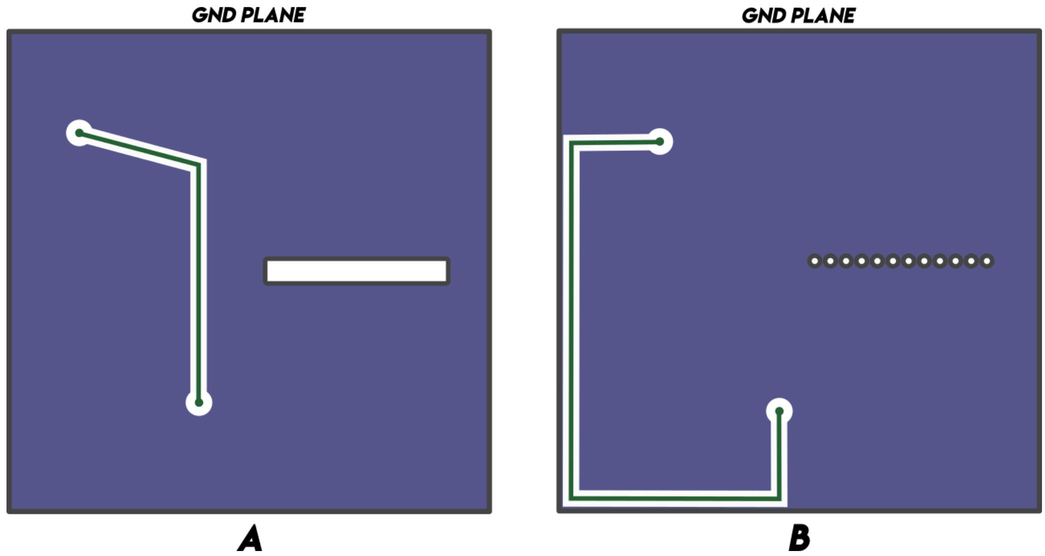

Figure 5: ***(A)*** *A GND layer design not suitable for EMC,* ***(B)*** *Design of a GND layer in accordance with EMC rules.*

6- If high-amplitude current switching circuit design and low-current circuit design need to be located in the same region for some reason, the distance between the high-amplitude switching layer and the other data lines layer should be increased. (*Figure 6*)

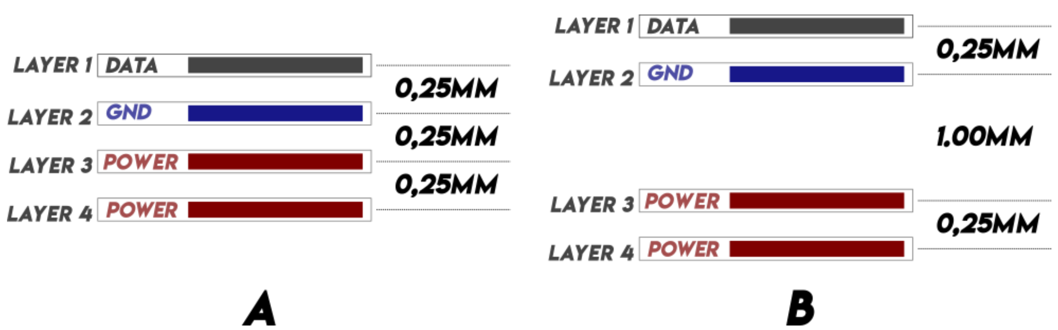

Figure 6: ***(A)*** *Illustration of the distance between layers,* ***(B)*** *Illustration of increased distance between layers.*

7- Some analog signal applications require a reference voltage. In this case, it is necessary to give the reference voltage from a single point. When two or more reference voltages are shown, splits will occur on the return path of the analog signal, causing the signal to receive noise. (*Figure 7*)

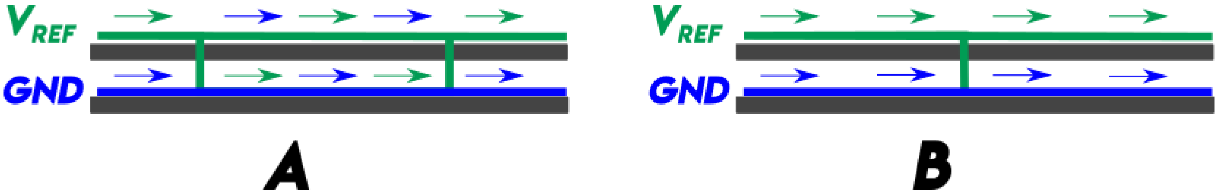

Figure 7: ***(A)*** *Illustration of the same line is referenced in two different points,* **(B)** *Illustration of the signal line referenced from a single point.*

5. Summary

EMC is an element that must be considered from the beginning of PCB design. According to the PCB to be designed, there are rules to be followed to protect us from EMI. And when these rules are examined, almost all of them are based on kirchhoff, maxwell, faraday laws and similar laws. When we act with the awareness of this, it will be much easier to understand why we use the method we used.