Sergi Parera

Small drives PCB layout for EMC compliance

MASTER’S THESIS

To achieve the university degree of

Master in Industrial Engineering

Master’s degree programme: Industrial Engineering

Submitted to

ETSEIB, Universitat Politècnica de Catalunya

Supervisor

Univ.-Prof. Dr.-Ing. Annette Mütze

Institut für Elektrische Antriebstechnik und Maschinen

Dipl.-Ing. Felix Krall

Graz, June 2019

EIDESSTATTLICHE ERKLÄRUNG

AFFIDAVIT

Ich erkläre an Eides statt, dass ich die vorliegende Arbeit selbstständig verfasst, andere als die angegebenen Quellen/Hilfsmittel nicht benutzt, und die den benutzten Quellen wörtlich und inhaltlich entnommenen Stellen als solche kenntlich gemacht habe. Das in TUGRAZonline hochgeladene Textdokument ist mit der vorliegenden Masterarbeit identisch.

I declare that I have authored this thesis independently, that I have not used other than the declared sources/resources, and that I have explicitly indicated all material which has been quoted either literally or by content from the sources used. The text document uploaded to TUGRAZonline is identical to the present master‘s thesis.

Datum / Date

Unterschrift / Signature

Acknowledgements

I would like to thank Prof. Anette Muetze for letting me have the opportunity to work in this project and with her team.

I want to especially thank Felix Krall for being my reference point in this research. Without his help and time this project would not have been possible.

Abstract

This project studies the relevance of following proper design techniques in order to have an EMC compliant PCB for small drives. Small drives usually make use of switching strategies where the highest frequencies involved are of the order of little MHz.

An existing board will serve as reference. It will be redesigned in a EMC friendly way and then tested in a semi-anechoic room in order to compare emissions and see if the layout is really relevant. In order to cut down the production costs, an additional board will be designed with a different set of components with lower prices. The objective is to see how far can the production cost be lowered before having an impact on the electromagnetic behaviour.

After the tests, it has been concluded that following a good design is really important even in boards without high frequency components.

It is hoped that this project will inform the reader about the basics of electromagnetic emissions and how they can be reduced.

Content

List of tables

List of figures

1 CHAPTER I. Introduction

1.1 History

Before 1895 the all long distance communications were based on the telegraph and the telephone. These systems used long cables which were wired to big central communications stations that connected the sender to the destination the message could be delivered. It was in this year when Guglielmo Marconi achieved the first wireless telegraph transmission using radio waves.

The American NAVY saw the potential in this invention and in 1899 they had an operation radio system but when they tried to use it they encountered the first known EMC problem. All devices were tuned to the same frequency so no intelligible information was audible. This problem was called Radio Frequency Interference (RFI).

The way fix the RFI problem was to assign a different carrier frequency to each radio station and they improved the technology and narrowed the bandwidth of the transmitters too. A way to regulate all this wireless connections was needed so the Federal Communications Commission (FCC) was created.

During World War II new types of transmitters and receivers were developed but no extensive tests were performed due to lack of time. This resulted in EMC problems on ships and planes. At the end of the war the Navy issued the first RFI standard and EMC became a respected engineering. [1]

In 1954 the Armour Research Foundation Conference on Radio Frequency Interference debuted. This organization still exists today under the name Electromagnetic Compatibility Society of the Institute of Electrical and Electronic Engineers (IEEE).

This was a brief history of the birth of Electromagnetic Compatibility concept. But there is one thing missing in the previous paragraphs and that is the invention of the microprocessors. Nowadays everywhere you look you can find something which has a microprocessor inside. Not only new things use this technology such as phones, digital cameras and computers. But devices and vehicles that existed before the invention of this microprocessors do too. Now microchips can be found in trains, cars, planes, televisions, radios. All this massive use of microtechnology brought another problem to the EMC.

1.2 Actuality

This sensitive circuits made the FCC and the European Union adapt extensive EMC regulations which every company must follow in order to be able to bring their products to market. That is why Electromagnetic Compatibily is now part of the design of anything involving electronics. Because electromagnetic emissions have to be taken into account from the beginning of the product creation in order to keep costs as low as possible and also the time to market. [2]

According to Moore's law, the amount of transistors in a chip is doubling every N months. At the beginning, when this observation was first said by Gordon Moore in 1965 N was 12 months. Nowadays it has slightly increased to 18-24 months. The relevance of this statement is that not only the number of transistors in microchips is increasing, other structural changes are happening in the chips such as the decreasing of its size. And the increasing of the clock speed. [3]

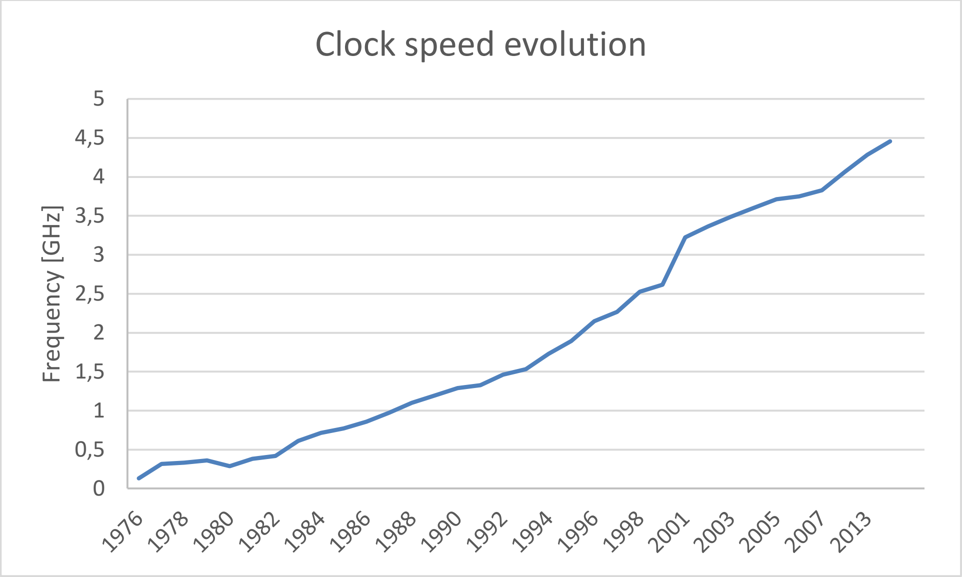


Figure 1: Clock speed evolution [3]

The use of smaller components and higher frequencies is not the best for EMC due to crosstalk between components because of their proximity and a faster switching speed.

More in depth concepts will be discussed in the next chapter.

1.3 Electronics in automotive sector

Since 1969, when Ford introduced the first computer controlled anti-skid system, the number of microprocessors in cars has been increasing. On average, standard cars use around 50 microprocessors. Some of them are used to read and process the signals that the sensors provide so safety systems can be triggered. Others, are the ones that perform control tasks such as control the fuel injection in the motor or the inverter in electric cars. [4]

At the same time, wire harness length has also been increasing. The use of various sensors around the chassis of the car and the need to bring the information to its "brain" and the use of small electric drives to perform different tasks are the reason why more and more wire is used in cars. Nowadays, cars have around 4 km of wires compared to the few hundred that they had 30 years ago. [5]

One of the reasons for this evolution is the new technology called X-by-wire. This technology is a replacement for the old mechanical, hydraulic and pneumatic systems. Instead of having wide hoses or long shafts in order to drive a load, electric cables are used. Electronic control provides better accuracy, faster response and an optimisation of space. For example, heavyduty vehicles use pneumatic brakes. Using the brakes-by-wire technology. Long hoses connecting the brake pedal with the brake clamps are replaced with wires. Then the pneumatic piston is replaced by an electric motor which will push the pad to the disk. [6]

As said before, electromagnetic emissions can cause interference to other systems if they have a way to 'escape' its wiring. Cables can work as two way antennas, they can emit electromagnetic radiation and also pick up radiation from another system. That is another reason why during the design of any electronics systems EMC has to be taken into consideration.

1.4 Case of study. Objectives

The aim of this thesis is to comprehend the effect of the location of the components in a PCB designed to drive small electric drives.

Due to the suboptimal design of the original board layout several prototypes will be manufactured in order to study the electromagnetic performance. Not only the layout will be tested, also a modification in the components in order to reduce the overall cost.

First, a prototype will be design with the exact same components as the original sample and only rearranging its position. This prototype will allow us to see if a proper arrangement of components is relevant in low frequencies PCBs.

In the second prototype some components will be changed in order to reduce the cost and have a better arrangement of components.

The last prototype will have the same layout as the second one but instead of being a 4-layer PCB it is going to have 2-layers which is even cheaper to manufacture.

All these prototypes will allow us to know the effect of the layout in slow frequencies PCBs and if it is really relevant. And if a cheaper PCB is possible to manufacture still complying with the regulations.

The original board is shown in Figure 2.

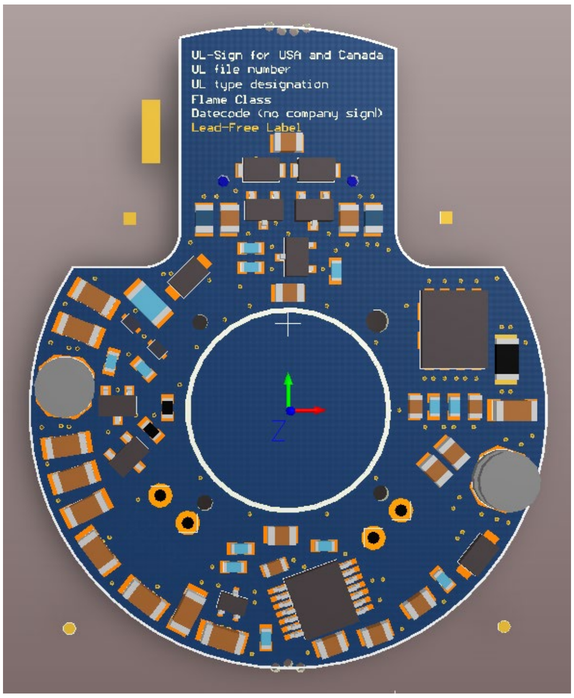


Figure 2: Original board layout

2 CHAPTER II. EMC basics

EMC stands for *ElectroMagnetic Compatibility*. A system or device is considered electromagnetically compatible with closer systems or surroundings when it satisfies the three next points:

- It does not produce electromagnetic emissions that interfere with other systems or devices
- It is not susceptible to electromagnetic interference coming from other systems or devices
- It does not interfere with itself

But how does the system interfere with other devices or the environment?

There are two types of emissions, conducted and radiated.

Conducted emissions: are the ones that propagate from the device to the power system net through the cable harness.

Radiated emissions: are the ones that use the cables, PCB traces or motor windings from the device as an antenna and radiate through the air.

In order to improve the electromagnetic behaviour there are three tasks that can be performed:

- Supress the emission at the source.
- Make the coupling path as inefficient as possible.
- Make the receptor less susceptible to emissions. Which doesn't mean that the emissions are smaller, it just reduces its influence.

2.1 Emissions measurements units

For conducted emissions, the important units are the ones that make reference to the inherent properties of electricity such as voltage and current. So the units used to measure conducted emissions are Amperes [A] (for current) and Volts [V] (for voltage).

On the other hand, radiated emissions are an electromagnetic wave and electromagnetic waves are formed by a magnetic field and an electric field. That is why the units used are [V/m] (for electric fields) and [A/m] (for magnetic fields).

There is one inconvenient in representing the quantities measured using these units though. In some measures, the electromagnetic field can be around 1 μV/m and in others 200 V/m. That is a difference of eight orders of magnitude (10^8). For that reason, every measurement is converted to a logarithmic scale to narrow the dynamic range of the measures and have a more understandable set of data. This new unit is called decibel [dB] and was first use by telephone companies to quantify the noise of its lines.

Basically, there is one formula to convert the “standard” units to decibels.

$$decibels\ (dB) = 20 \log \frac{Measure}{Reference} \tag{1}$$

Decibels: Desired value

Measure: Value measured

Reference: Every conversion to decibels is compared to a reference value with the same units as the measure.

For example:

$$120\ dB\mu V = 20 \log \frac{1\ V}{10^{-6}\ V} \tag{2}$$

Or the same:

$$120\ dB\mu V = 20 \log \frac{1\ V}{1\ \mu V} \tag{3}$$

The units which the decibels make reference to may be different from the microvolts.

For example:

$$60\ dBmA = 20 \log \frac{1\ A}{1\ mA} \tag{4}$$

Converting these units to decibels makes it easier to compare and to plot into a graph if necessary. In a further chapter the legislation will be explained and there the emission’s limits will be shown.

2.2 Common-mode and differential-mode currents

When analysing the emissions of a product it is important to make a difference from the ones created by differential-mode currents and the ones created by common-mode currents. Differential mode currents are the ones that are known and intended. They are calculated during the design using Kirchoff's and Ohm's laws. So these currents are the ones that make the device work as intended.

On the other hand, common-mode currents are not calculated using any basic tool learned in basic electrics or electronics. These currents also appear while the device is running but are not intended so they have to be carefully controlled.

Taking a look at a two wiring diagram like the one shown in the next figure. Let's suppose that there is a load connected to the right end of the wires and a power supply to the left end.

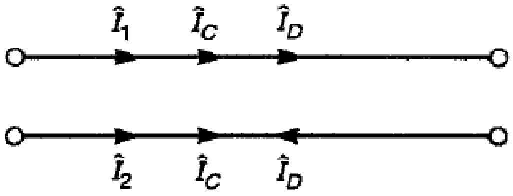


Figure 3: Decomposition of currents into common-mode and differential mode currents [7]

Differential mode currents flow from the positive side of the power supply through the load and go back to the power supply through the lower wire. If the parasitic impedances of both wires are exactly the same, they both have the same value but opposite directions. Common mode currents also have the same value in both wires but they flow in the same direction if the parasitic impedances are equal.

The following equations can be written as a decomposition of the currents flowing through the wires.

$$\hat{I}_1 = \hat{I}_C + \hat{I}_D \tag{5}$$

$$\hat{I}_2 = \hat{I}_C - \hat{I}_D \tag{6}$$

Combining the above two equations two new equations can be written.

$$\hat{I}_C = \frac{(\hat{I}_1 - \hat{I}_2)}{2} \tag{7}$$

$$\hat{I}_D = \frac{(\hat{I}_1 + \hat{I}_2)}{2} \tag{8}$$

If we take a look at the electric field generated by the currents it can be seen that the electric field generated by the differential-mode currents is smaller than the one created by the common-mode currents. That would be true if the currents had the same value which they do not have. But it is interesting to notice that in the differential-mode currents, the electric fields generated by each wire are opposite to each other so apply the superposition theorem, the electric fields have to be subtracted. Otherwise, in the common-mode currents, they flow in the same direction thus the electric field generated by these currents add to each other.

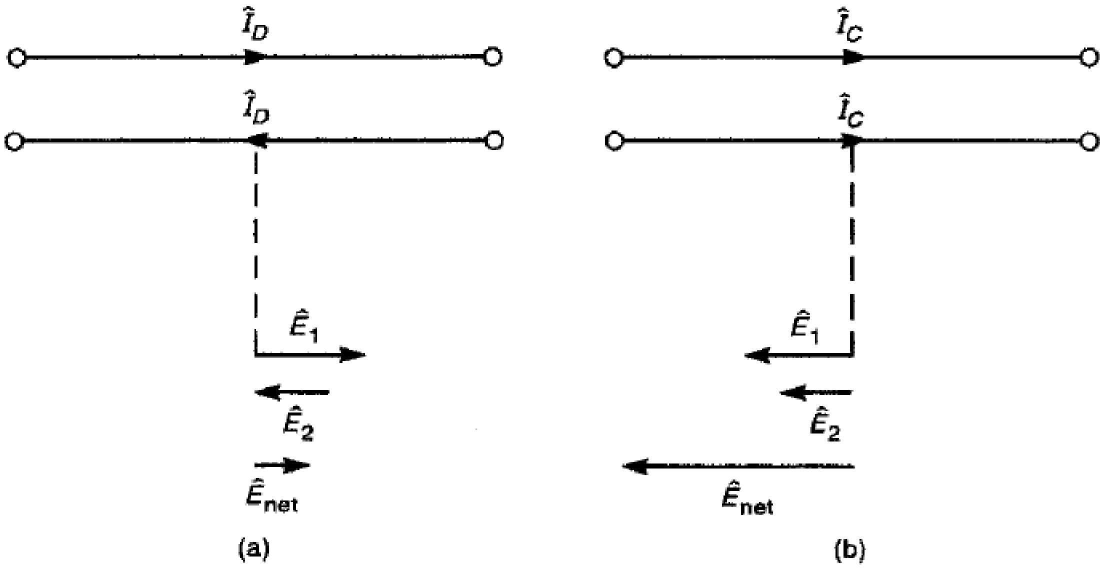


Figure 4: Electric field generated by (a) differential-mode currents (b) common-mode currents [7]

Electromagnetic emissions are proportional to the value of the electric field, for this reason, small common-mode currents can generate equal or even more powerful emissions than differential-mode currents that have a higher value. For example, a common-mode current with the magnitude of microamperes can produce the same level of emissions as a differential-mode current of miliamperes.

2.3 Filters

As said before, the most effective way to reduce the emissions is to reduce them at their source. Sometimes this cannot be achieved then filters can be used to reduce them by creating a “resistance” or as it is called, inserting a loss in the circuit. There are various filter topologies, the most used ones are:

- LC filter
- CLC or π-filter
- LCL or T-filter

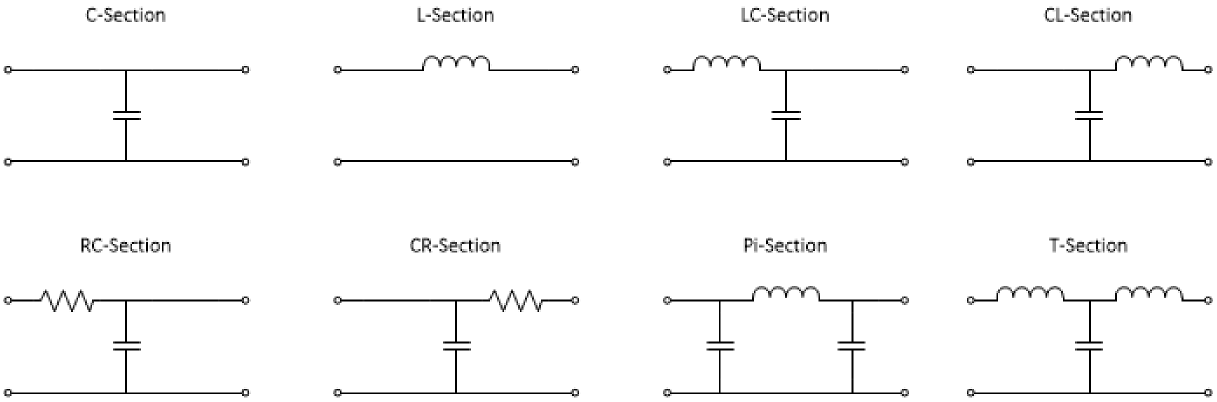


Figure 5: Filter topologies [8]

LC filter

It is a low pass filter, it consists of an inductor in series with the load and a capacitor in parallel with the load. The impedance of the inductor is calculated as following:

$$X_L = 2\pi f L \tag{9}$$

The impedance is higher with higher frequencies that is why higher frequencies are attenuated by this component. On the other hand, impedance of the capacitor is:

$$X_C = \frac{1}{2\pi f C} \tag{10}$$

The impedance is lower with higher frequencies so they find a path of low resistance to ground instead of going through the load.

All circuits have an input and an output resistance. During the measurements the source resistance is fixed to 50 Ω but in real applications it may differ so it is important to know how will the filter behave with different values of loads. [8]

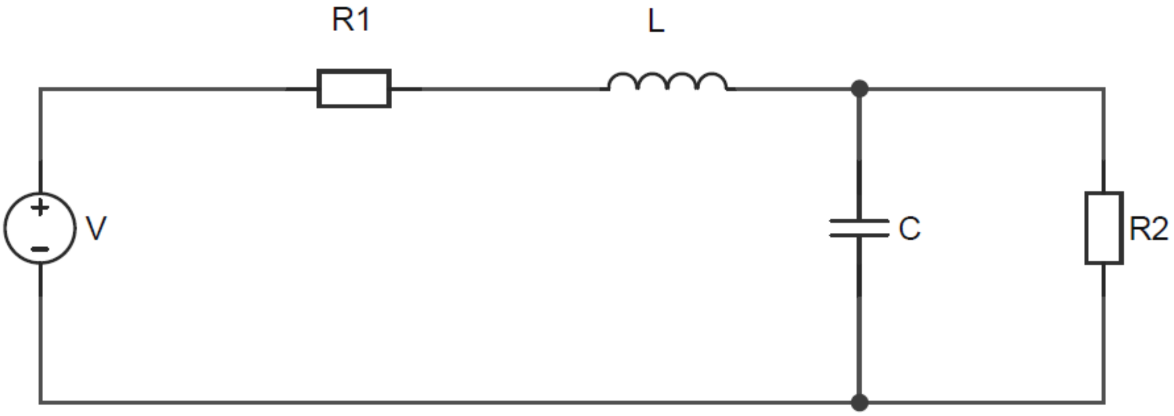


Figure 6: LC filter

A transfer function is going to allow us to study its behaviour. The following transfer functions has been obtained following the FACT method explained in [9].

The transfer function for the input/output voltage of the LC filter is:

$$G(s) = \frac{1}{s^2 \cdot L \cdot C + s \cdot \left(R_1 \cdot C + \frac{L}{R_2}\right) + \frac{R_1}{R_2} + 1} \cdot \frac{R_1 + R_2}{R_2} \tag{11}$$

The resistor R1 is the source resistance and the transfer function has been obtained as if it was part of the circuit. In order to undo the effect of the voltage divider the last part of the equation is needed. Without the term $\frac{R_1+R_2}{R_2}$ the function with balanced impedances (R1 = R2) has an overall attenuation of -6 dB. Which is equal to divide the input voltage by two.

Depending on the size of the components used on the filter, a cut off or corner frequency can be set. Noise and signals above this frequency will be attenuated. The level of attenuation depends on how far these frequencies are from the cut off frequency.

$$f_{cut\,off} = \frac{1}{2\pi\sqrt{LC}} \tag{12}$$

CLC filter

This filter’s topology is like an LC filter with an input capacitor. This capacitor filters the majority of the ripple coming from the source, after this prefiltering the LC filter attenuates the remaining noise. This is why a π-filter is more efficient than a simple LC filter. It also has its drawbacks. It is not suitable for devices where load current varies. In such type of applications, a simple LC filter should be used.

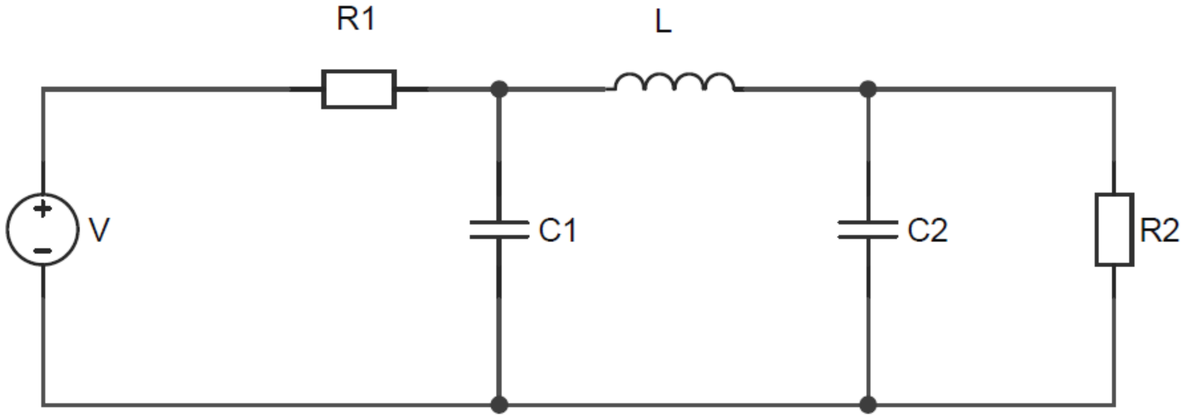


Figure 7: CLC filter

The transfer function of this filter is a third order transfer function which indicates that the slope after the cut off frequency is going to be -60 dB/decade. That is only going to be exactly true if there is a relative high source impedance.

In the following sections the effect of different source and load impedances will be simulated.

The transfer function for the input/output voltage of the CLC filter is:

$$G(s) = \frac{1}{s \cdot C_1 \cdot R_1 + 1} \cdot \frac{\dfrac{1}{s \cdot C_2 + \dfrac{1}{R_2}}}{\dfrac{1}{s \cdot C_2 + \dfrac{1}{R_2}} + \dfrac{1}{s \cdot C_1 + \dfrac{1}{R_1}} + s \cdot L} \cdot \frac{R_1 + R_2}{R_2} \tag{13}$$

In theory, the CLC filter is going to provide better attenuation than the LC filter in the same range of frequencies due to the steeper slope.

LCL filter

It is also a third order filter with two inductors and one capacitors emulating the shape of the letter 'T' as shown in Figure 5. This filter works best with low input and low output impedance. That is why it is usually applied to grid-tie inverters.

For this reason, the T-filter will not be studied in this project and simulations and decisions will be made between the LC filter and the π-filter.

Matlab simulation

In order to simulate the desired attenuation Matlab software will be used. Provided that the switching frequencies of the PWM strategy are 22,89 and 34,33 kHz. The designed filter should have a better attenuation at these frequencies than the previously used.

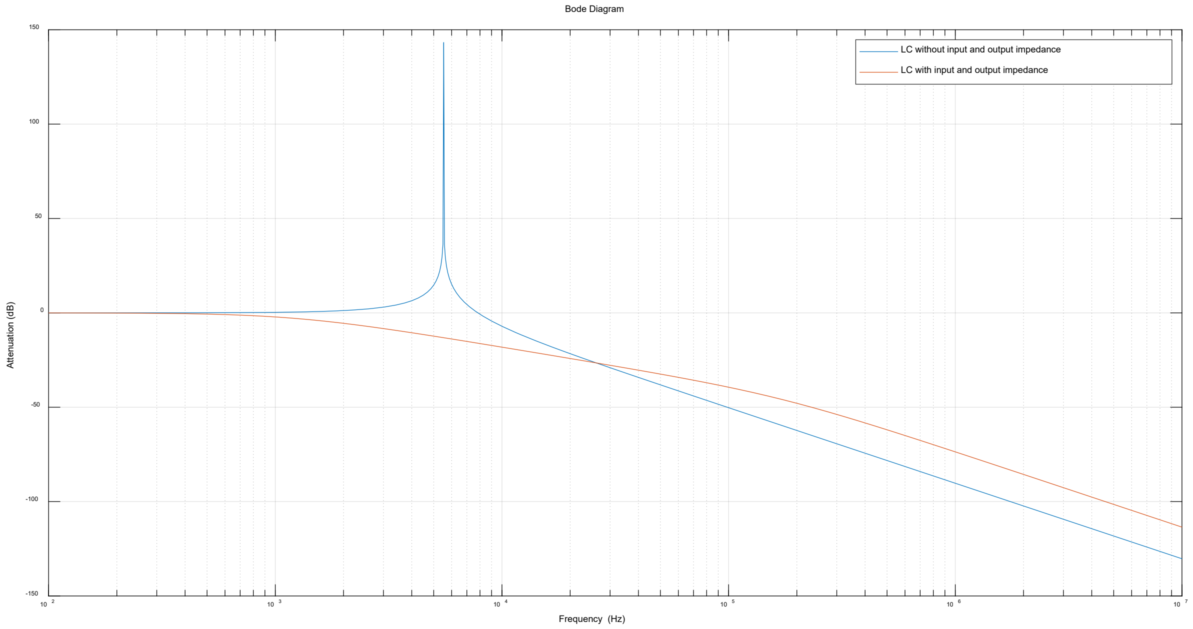


Figure 8: Comparison between different transfer functions for LC filter

Figure 8 compares the transfer function between input and output voltage depending on the transfer function used. In both plots an inductance of 47 µH has been used and eight capacitors in parallel of 2,2 µF. The blue line is the transfer function of only the capacitor and the inductor, without any resistance taken into account. Note that there is a spike and it is when both components resonate. Adding an input resistance of 50 Ω which is the resistance of the net and an output resistance of 8,5 Ω which is the resistance of the winding of the motor we obtain the orange line. It doesn't have any spike but considering the same frequency it attenuates less than the "ideal" one.

This emphasizes the importance of using the most real like transfer function possible and not an ideal one. This way it is possible to simulate more accurately the results and behaviour that will be obtained in the real prototype.

The cutoff frequency of the original LC filter is around 5 kHz and at 150 kHz, which is the starting frequency of the test, it has an attenuation of -44,1 dB. This will be used as a reference for the design of the CLC filter.

CLC filter design

As explained before, the CLC filter is better at filtering because it has a steeper slope than the LC filter. In order to visualize this, in the next figure a π-filter and an LC filter are compared. Both with the same amount of capacitors, eight, but in the π-filter they are arranged as four input capacitors and four output capacitors.

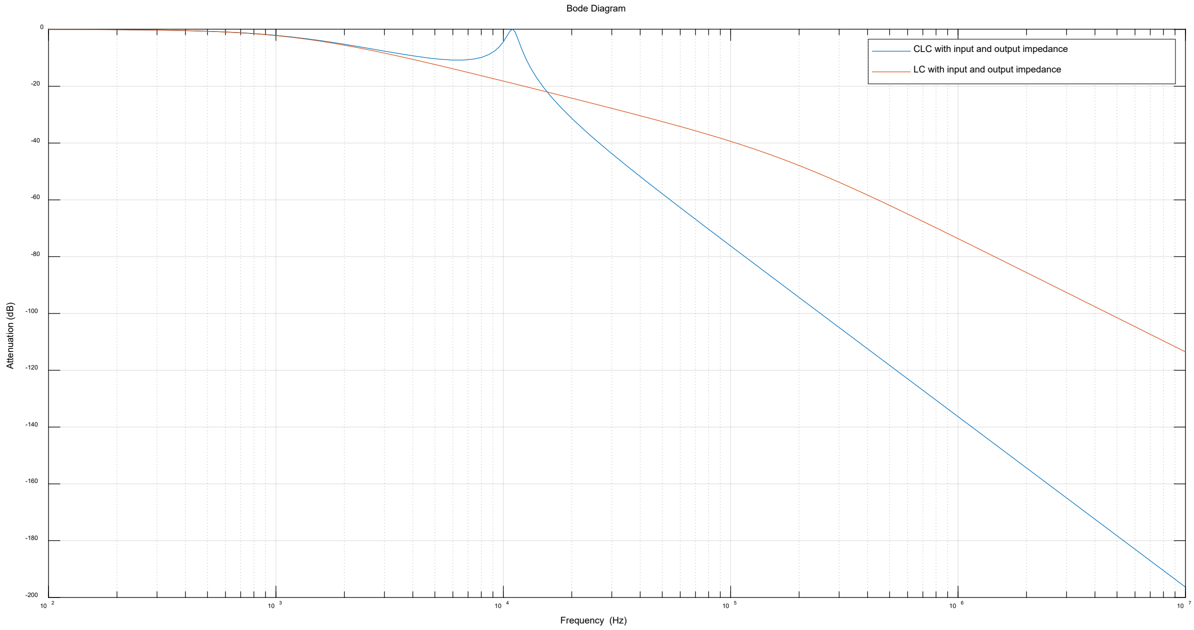


Figure 9: Comparison between LC and CLC filter

It is clearly seen that the CLC filter is better than the LC filter in attenuating, at 150 kHz this π-filter has an attenuation of -86,9 dB. Apparently this filter would be enough to satisfy the requirements but there are a couple of things to observe of this filter. First, some different input and output capacitors will be tested. The PCB will have 4 slots for the capacitors at the input and 4 slots at the output. So we have already simulated with a 2.2 μF capacitor in every slot. Now we will simulate with less capacitors.

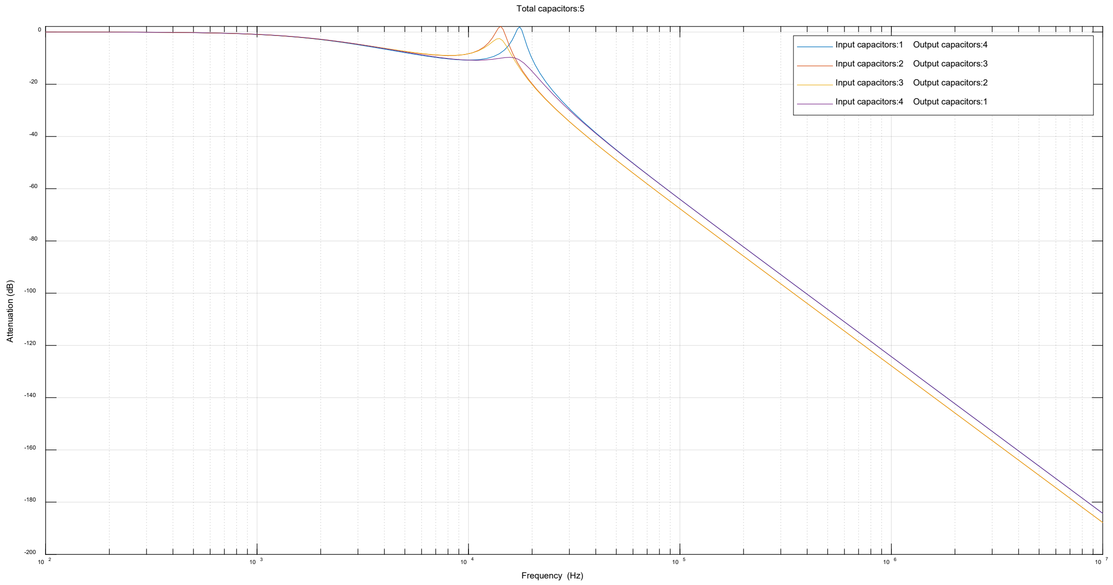


Figure 10: CLC filter with 5 capacitors

In this case, the best distribution is with three input capacitors and two output capacitors. This combination has an attenuation of -78,3 dB which much more than the LC filter but it is still 10 dB away from a π-filter with eight capacitors.

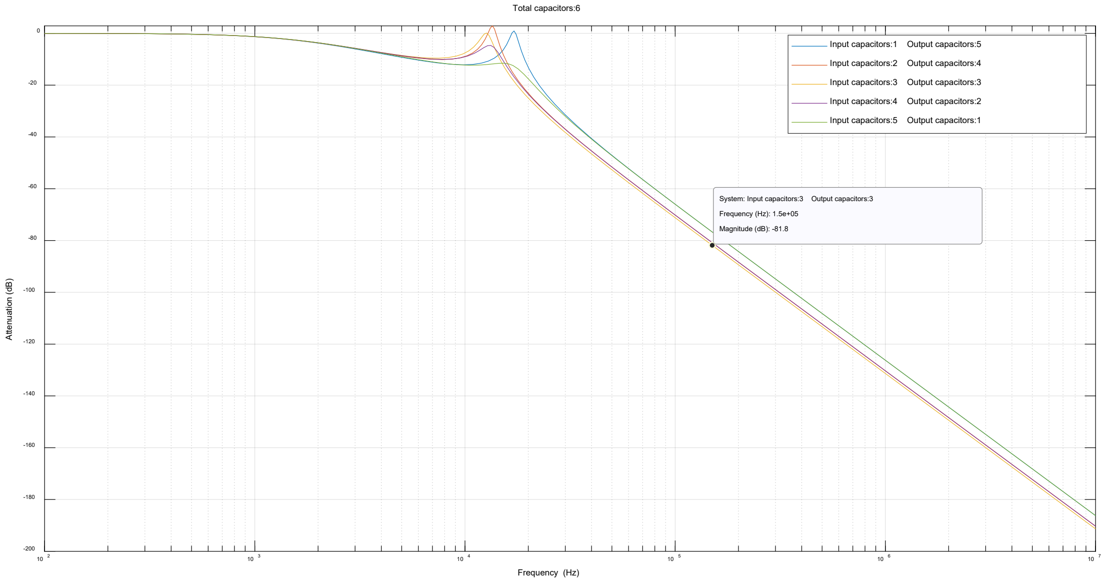


Figure 11: CLC filter with 6 capacitors

With a total of six capacitors, the best configuration is with three at the input and three at the output witch offers an attenuation of -81,8 dB.

With all these simulations, it can be concluded that the CLC filter should be better at attenuating the high frequency components than the actual LC filter if at least five capacitors are used.

2.4 Conducted emissions

Conducted emissions, as described earlier in this thesis, are the ones transmitted through the power cord of the device. Regulatory agencies have set some limits to these emissions. One may think that conducted emissions cannot interact with other devices nearby or affect them in any way but this is not true. There is a large array of wires interconnected to each other and to many power outlets; in a building for example. If the emission levels of one device connected to this grid are too high, they can be transferred to another socket or to the commercial grid and make it not work properly.

In the same way, in order to be considered a good design it is not enough to comply with the emission limits, it is also equally important to be “immune” to noise coming from the grid. For example, when there is a power surge caused by a lightning that strikes the power grid. In this case, the device must be capable of keep working properly and if necessary lead to an automatic shutdown but without losing any data or malfunctioning.

2.4.1 Voltage regulator

The scope of this project is not about voltage regulators but general understanding is needed because they are used in every PCB.

The two large groups of voltage regulators are:

Switching-mode voltage regulator

They use high frequency signals to control one or two MOSFETS (depending if they use a diode or a MOSFET to control the current return path when the capacitor is discharging) thus they are prone to emit more electromagnetic radiation.

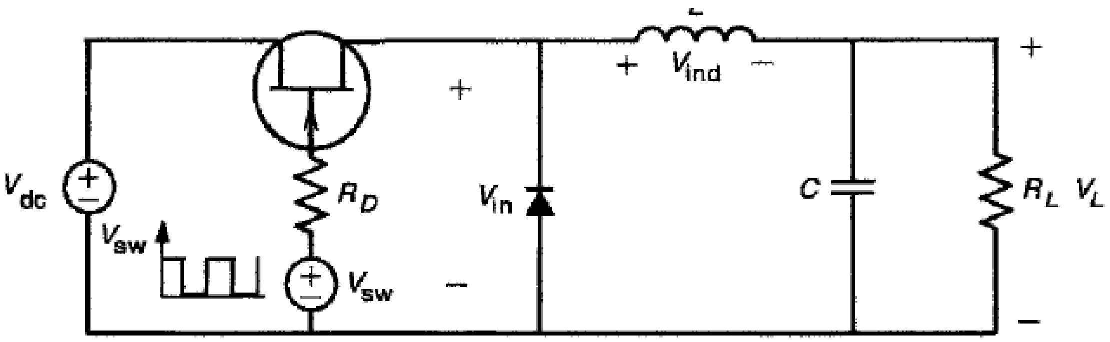


Figure 12: Buck converter [7]

The most common switching mode power supply is the buck converter which steps down voltage. Typically, it consists of a MOSFET, a diode, an inductor and a capacitor. The MOSFET is controlled using a PWM[1] (Pulse Width Modulation) signal. Changing the duty cycle of the PWM signal the output changes too. Different from the transistor in the linear power supplies, the MOSFET only has two states, ON or OFF. When the MOSFET is ON, the dc input voltage is applied to the inductor which creates its magnetic field and charging the capacitor. When it is turned OFF the inductor inverts its polarity according to Faraday’s law and the capacitor discharges. The current flows through the diode and keeps powering the load.

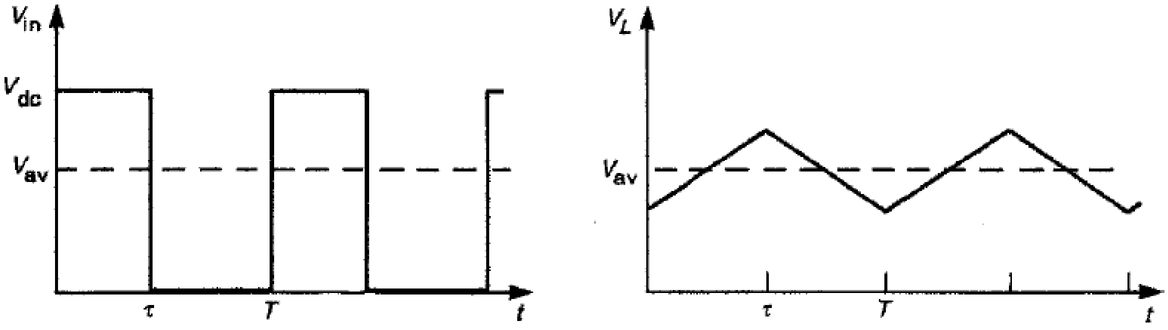


Figure 13: Buck converter. Voltage across the MOSFET (left). Output voltage (right) [7]

The duty cycle that has to be used can be calculated using:

$$Duty\ cycle = \frac{V_{OUT}}{V_{IN}}\ x\ 100 \tag{14}$$

Its efficiency is really high. If the components are well sized for the application that they are needed it can easily reach more than 90% of efficiency.

Linear voltage regulator

The board that is being improved in this project needs to step down its voltage in order to feed the microcontroller. A step down buck converter as explained could do the work but it contains a switching element which might contribute to worse EMI behaviour. In order to solve this, a linear regulator will be used because it doesn’t have a switching component and despite having a much lower efficiency, the power loss will not be important. [10]

---

[1] More about PWM switching strategy in Chapter 3.

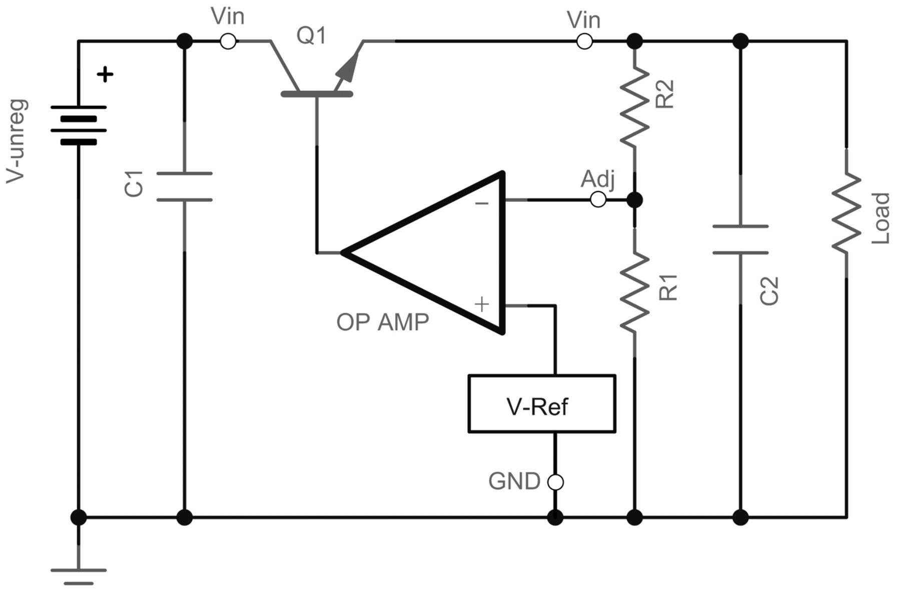


Figure 14: Linear voltage regulator [11]

The working principle of the linear voltage regulator is quite simple. The capacitors C1 and C2 are decoupling capacitors which filter the noise in the input and the output. Q1 is an NPN transistor which, controlled by the operation amplifier, tries to match the voltage between the output and V-Ref. R1 and R2 act as a voltage divider to sense the output voltage.

For example, if the voltage reference is 2,5 V and the desired output voltage is 5 V, R1 and R2 must have the same value so they divide 5 V in half to match the 2,5 V of the reference. The power loss of this device is because the transistor usually works in its linear region creating the voltage drop needed for the output voltage.

$$Power\ loss = Voltage\ drop \cdot Current \tag{15}$$

2.5 Radiated emissions

Radiated emissions, as explained in the beginning of this chapter, is the unintended emission of electromagnetic energy which travels through air and can affect nearby devices even though not being physically in contact. Wires and/or tracks act as antennas that can send or pick up non desired signals. [12]

During radiated emissions tests, only the radiated emissions coming from the device and its power cord are measured. But in real life, radiated emissions can appear in the power grid too. Just imagine that a device connected to the grid has a high level of conducted emissions, these emissions can use the wires of the net as antennas and radiate.

2.5.1 Assumptions

Near field vs far field

When studying the behaviour of the electric fields there is a difference to be established. It is important to take into account if the point of interest is inside the near field or in the far field. Different equations model its behaviour depending on which field it is. For this reason, it is important to determine where is this boundary.

In order to decide if an antenna is **electromagnetically short or long**, the wavelength of the frequency it carries has to be calculated. The highest frequency that is tested in this device in order to decide if it complies with the regulation is 30 MHz.

$$\lambda = \frac{c}{f} = \frac{3 \cdot 10^8\ m/s}{3 \cdot 10^7\ s^{-1}} = 10\ m \tag{16}$$

It is considered an electromagnetically short antenna if it is shorter than half of the wavelength. The PCB is around 6 cm long, for this reason, any PCB trace that could be acting as antenna will be considered an electromagnetically short antenna because it is below half of a wavelength.

For this type of antenna, the near field goes from the source to a wavelength away. Then, there is a transition zone between 1 and 2 wavelengths and anything further away 2 wavelengths is considered the far field.

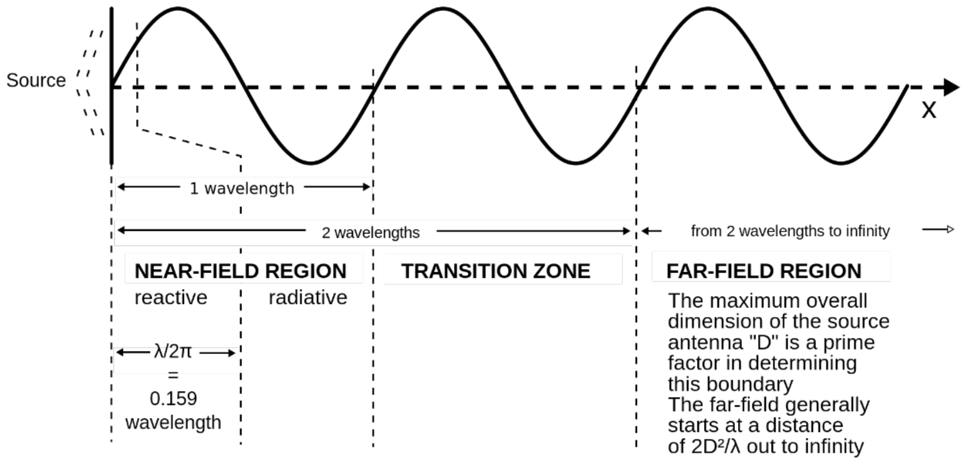


Figure 15: Near field and far field transition zones in antennas [13]

2.5.2 Antenna

Antennas are nothing else than conductors that create and electromagnetic field which travels through space. There are mainly two types of antennas that can appear in PCB designs:

- Dipole antenna
- Loop antenna

Dipole Antenna

Is one of the simple and widely used antennas. It consists of two conductors that change polarities in a concrete frequency. When used in intended communications, this frequency is set by the transmission protocol. For example, the frequency of 2.4 GHz is broadly used. With this frequency, the length of each cable should be a fourth of a wavelength. [14]

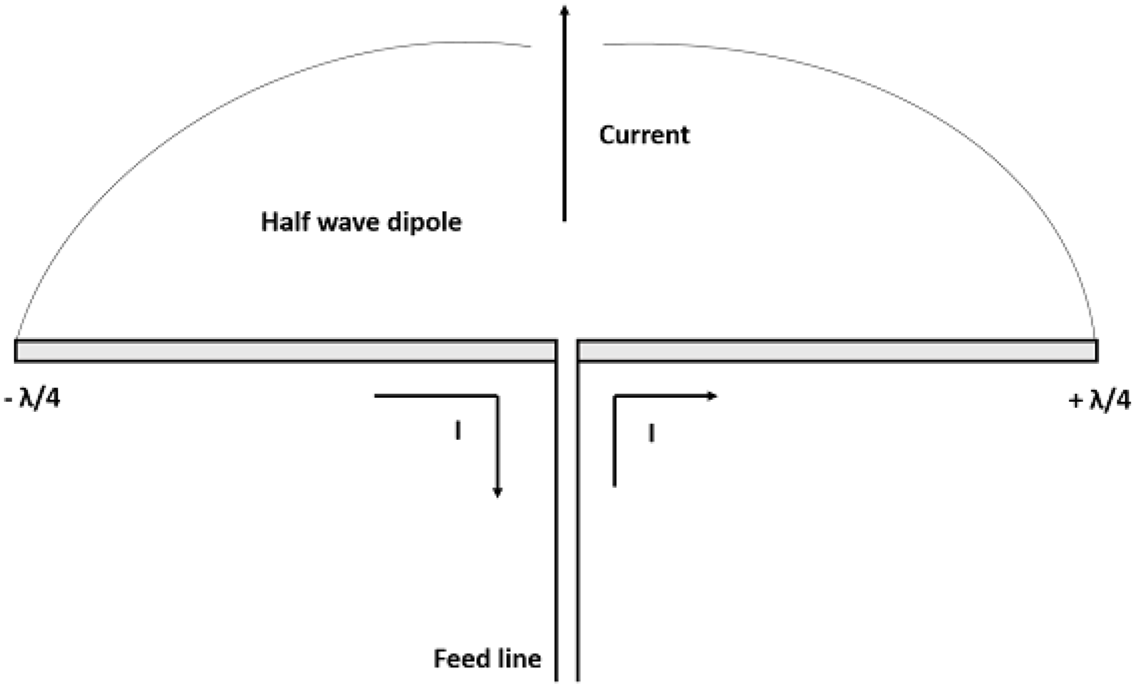


Figure 16: Half wave dipole antenna [14]

It is important to have the right length of antenna so it can electrically resonate at the desired frequency. Resonance enhances the performance of the antenna making the transmitted signal more powerful.

When designing the PCB layout, it is important not to leave any traces that carry current or signals without connecting so they cannot perform as a dipole antenna.

Loop antenna

This loop antenna consists of a conductor shaped in a circular loop. There are small and big loop antennas. This thesis is about rearranging the layout of a PCB where the components and trace dimensions are small. It looks obvious that small loop antennas are the ones that are worth study.

The difference between small or large loop antennas is the longitude of the loop. If it is less than a tenth of a wavelength it is considered a small loop antenna.

Considering a clock frequency of 20 MHz, a tenth of its wavelength is 1.5 m which means that, clearly, in a PCB we can expect small loop antennas. It is worth mention that small antennas also take into account far field equations.

These antennas don't need to have an exact circular shape. Any closed loop shape is enough to emit electromagnetic waves. This type of antenna cannot be avoided in PCB because current need a return path to its origin.

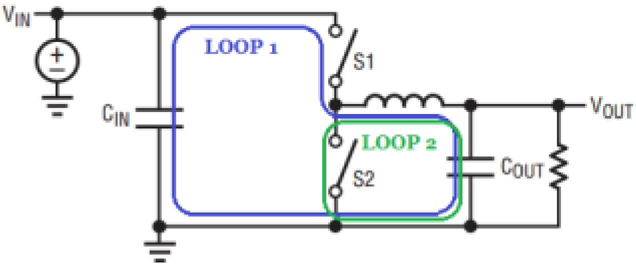
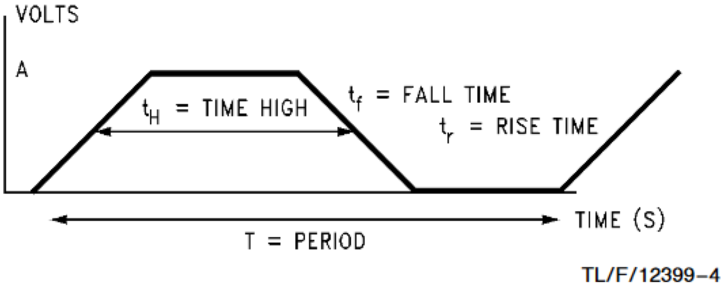


Figure 17: Loop area in a buck converter [15]

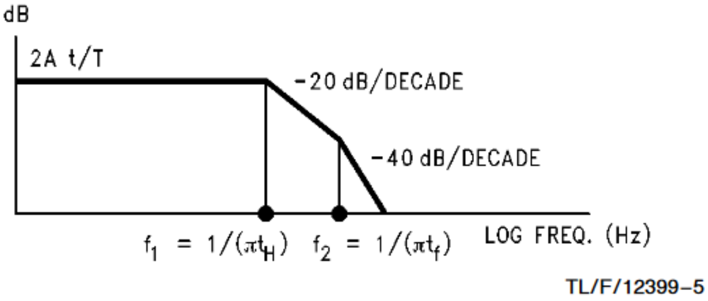
The previous figure illustrates the current loops that a circuit has. When arranging the components in the PCB it has to be taken into account and try to make the loop as small as possible so the emissions are reduced.

2.6 Importance of frequency rise/fall times

It is understood that a higher clock frequency implies a higher amount of high frequency components existing in the board. And with an increased clock frequency, the rise/fall times are also smaller – or faster.



Time Domain



Frequency Domain

Figure 18: Time to frequency domain [16]

This figure shows the time and frequency domain of a trapezoidal waveform. The first slope of -20 dB/dec is caused by the clock frequency and has nothing to do with the rise/fall times. Otherwise, the second slope of -40 dB/dec is caused by the rise/fall times. If they are increased, the second corner frequency is reduced and with that, there are less high frequency components.

For this reason, it is important to control this rise/fall times and make them as high as possible without disturbing the functionality of the device. For example, the value of the gate resistors of the MOSFETs can be changed in order to increase or decrease the gate current and then increase its rise/fall time.

Increasing the rise/fall times has a drawback, it increases the switching losses which are in form of heat. For this reason, the switching speeds can only be modified if the component can withstand the increment of temperature.

$$P_{sw} = \frac{1}{2} \cdot V_{in} \cdot I \cdot (tr + tf) \cdot f \qquad (17)$$

As seen in the equation [17], the power losses are directly related to the rise/fall times.

2.7 Crosstalk, coupling and decoupling

2.7.1 Crosstalk

In a PCB it is usual to have parallel traces, if the distance between them and with the ground plane is not enough, crosstalk can appear. In order to have crosstalk between traces, at least three traces are required, the source one, the victim and the reference. Both source and victim can have two types of coupling. Inductive coupling which occurs when there is a time-varying current flowing through the source conductors generating a magnetic field. This field induces current to the victim trace. [18]

The gap of non-conductor material between two traces can be seen as a dielectric and both traces as the conductor plates of a capacitor. Which means that there can also be capacitive coupling between both traces.

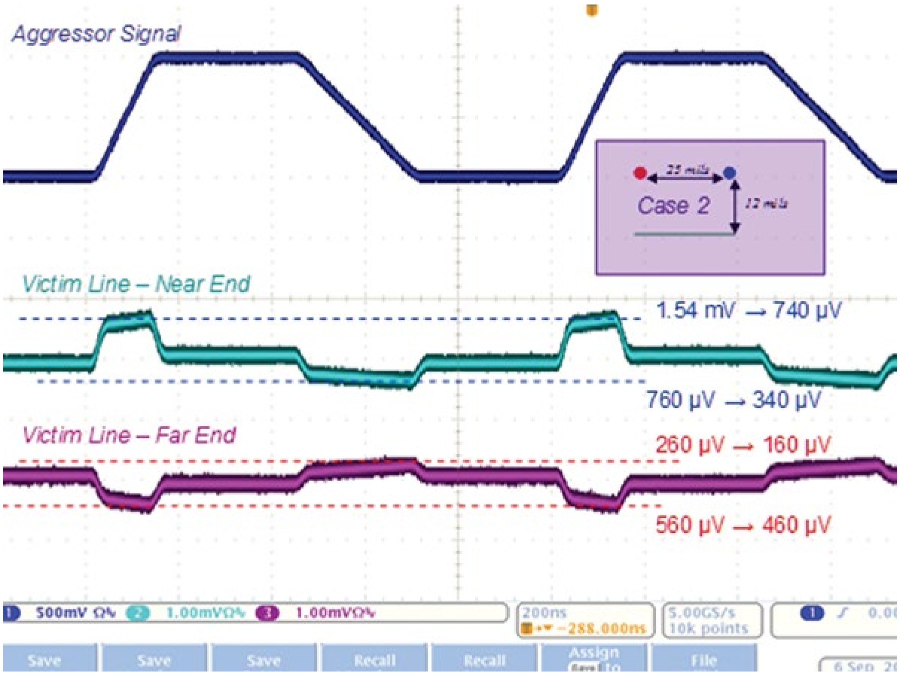


Figure 19: Induced voltage due to crosstalk [18]

In Figure 19 it can be seen that the voltage spikes produced by the source in the victim trace appear only when there is a voltage level variation. The slower the voltage change, the lower the crosstalk level. That is another reason to try to keep rise/fall times as slow as possible other than its own frequency components.

2.7.2 Decoupling capacitors

Microcontrollers and other sensitive IC need a constant voltage to operate. In order to guarantee this decoupling – or bypass – capacitors are placed near the voltage pin of this ICs.

Once they are charged, its function is to absorb any voltage spikes produced by noise, undesired coupling or just a power surge. They can also provide power to the integrated circuit if the input voltage drops.

These capacitors are incredibly necessary, without them, microcontrollers might skip some instructions and behave abnormally. [19]

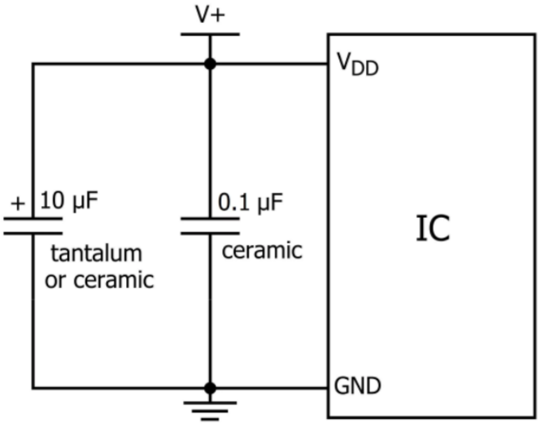


Figure 20: Decoupling capacitors [19]

Due to the frequency response of different capacitor types and capacities the most common way to place these capacitors is to place a low capacity ceramic capacitor near the IC's pin and a larger capacity in parallel. The smallest capacitor has a good high frequency response and is going to decouple high frequency noise or variations. The larger capacitor is better at low frequencies; this way they complement each other.

2.8 Non ideal behaviour

Capacitors and inductors don't have an ideal behaviour. They are not a pure capacitance or a pure inductance. They have two leads and made of conductor, that is why in reality they are a combination of resistance, capacitance and inductance. The difference is that in capacitors, for example, the capacitance component is much higher than the inductance or resistance.

The next step is to visualize which implications has this non ideal behaviour of the components

Non ideal capacitors

As stated before, a real capacitor should be modelled as a resistance, an impedance and a capacitance in series. The resistance is called ESR (Equivalent Series Resistance) and the inductance ESL (Equivalent Series Inductance). The equation of the impedance of the circuit would be the following:

$$Z = \sqrt{R^2 + \left(2\pi f L - \frac{1}{2\pi f C}\right)^2} \tag{18}$$

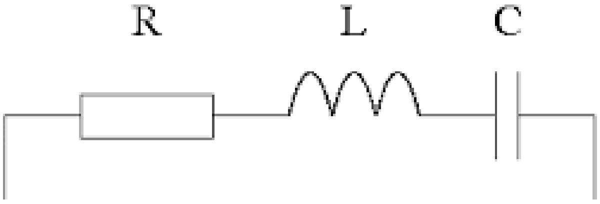


Figure 21: Equivalent circuit of a real capacitor

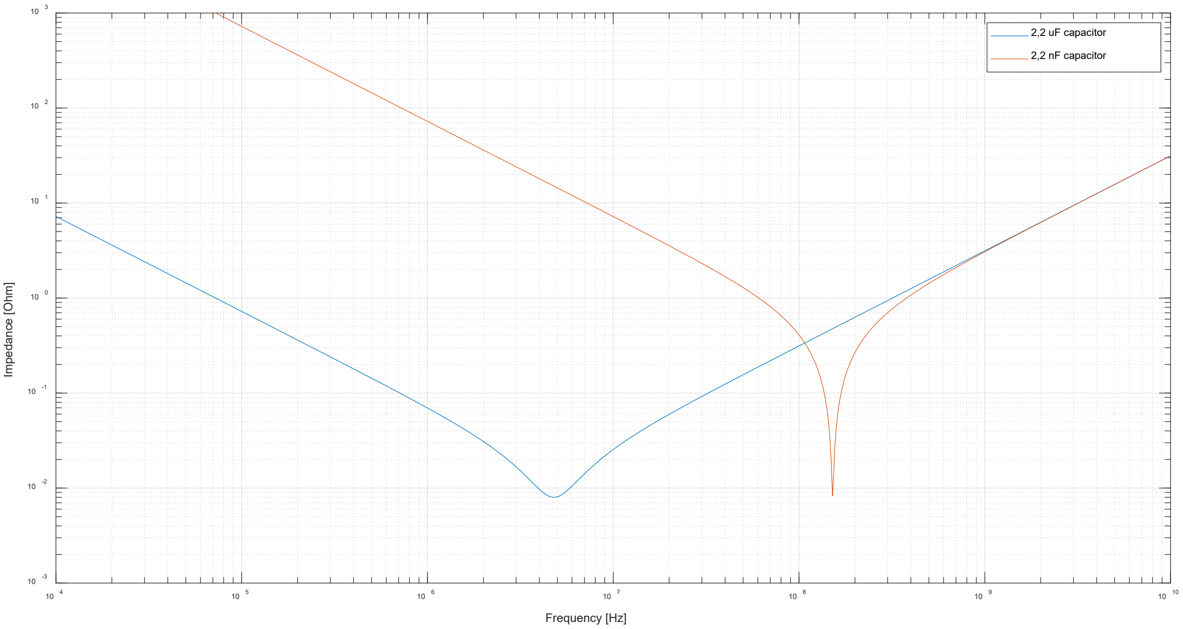


Figure 22: Impedance of two capacitors of different size

As seen in the graph, there is a frequency where the impedance starts to rise, this is due to the inductance. This should be taken into account when designing any circuit, especially on high frequency applications. This increasing of the impedance will mean that these high frequency components no longer have a low impedance path to “escape” so they will not be filtered out.

Note that depending of the capacitance of each capacitor, the resonance frequency varies. The higher the capacitance, the lower the resonance frequency. If it is important to have a high resonance frequency, multiple capacitors in parallel should be used.

Also, it is important to know which is this the so called resonant frequency of the circuit. The frequency where the impedance is minimum is:

$$f = \frac{1}{2\pi\sqrt{C \cdot L}} \tag{19}$$

The impedance from the capacitor and the impedance from the inductor cancel each other. Which means that the impedance of the circuit is at its minimum. If there is a constant voltage source, by the Ohm´s law, the current will be at its maximum; which could be dangerous. [20].

Non ideal inductors

Similarly to capacitors, inductors are not purely an inductance, they also have a capacitive and a resistive component. There are basically two different models to simulate the real behaviour of an inductor; the old and simple model and the new and more accurate model but also more difficult.

For the sake of this project the easy one is enough to get an idea of how does the non-idealistic behaviour of the component is reflected in the real world appliance. More information about the newest model can be found in [21].

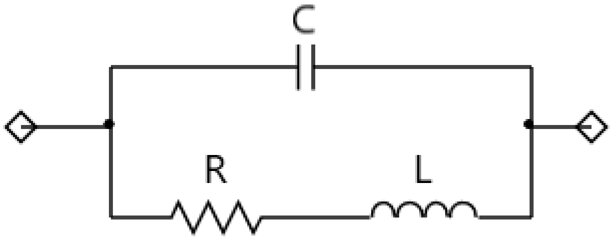


Figure 23: Real inductor equivalent circuit [21]

The model discussed here is going to be a model with three elements, the inductor with a series resistance and a capacitor in parallel.

In this case the formula that describes the impedance is derived from Ohm’s law:

$$Z = \frac{1}{\frac{1}{R + 2\pi f L} + 2\pi f C} \tag{20}$$

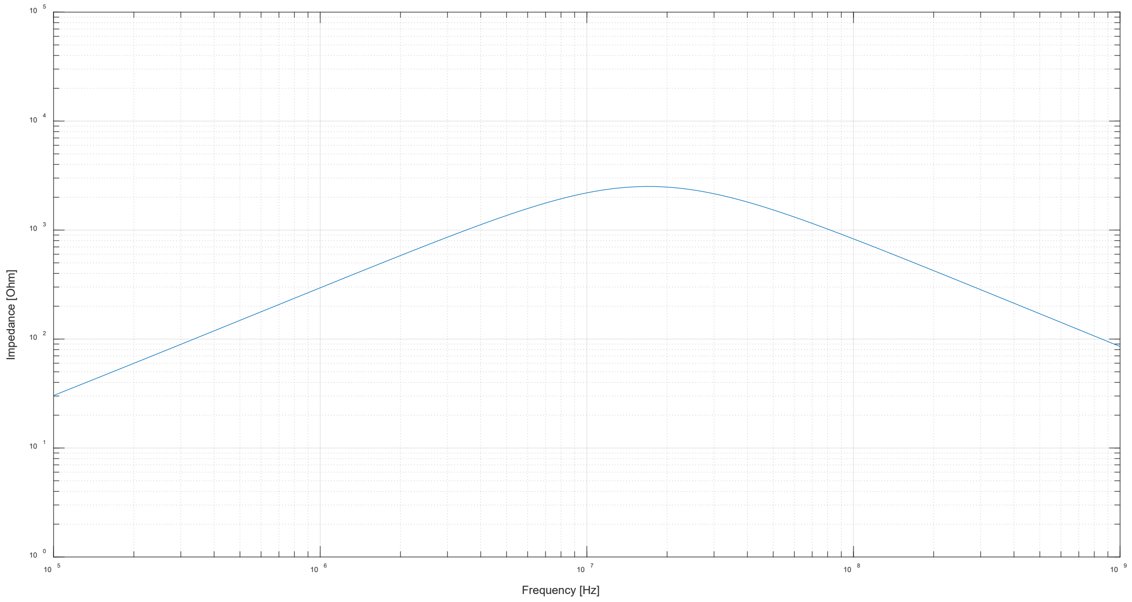


Figure 24: Simulation of a real inductor impedance. Simple model of a 47 µH inductor

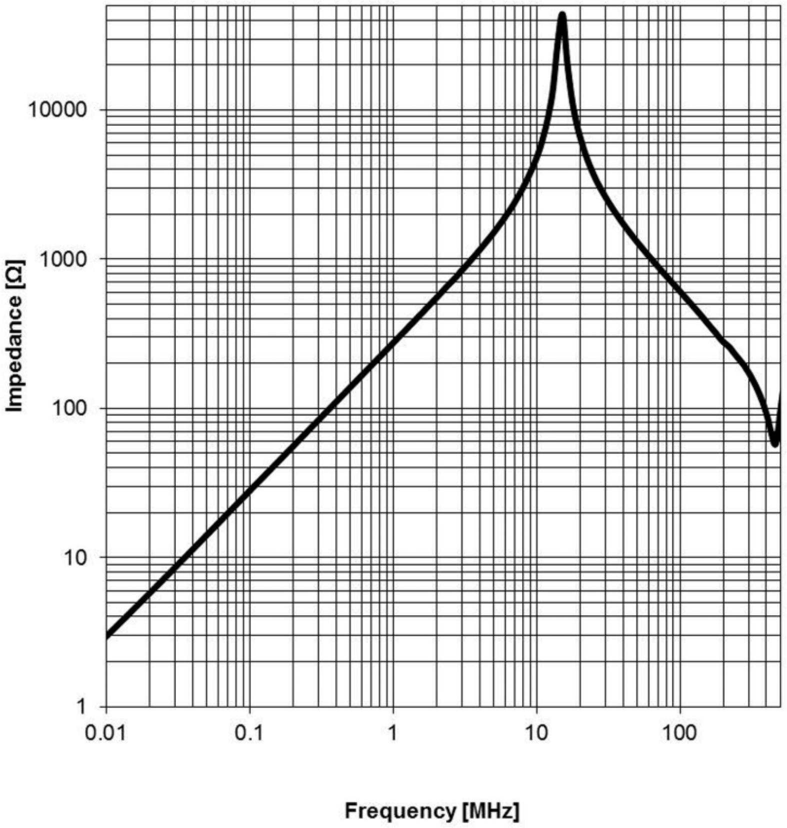


Figure 25: Real impedance from the 47 µH datasheet [22]

The graph shows that this model doesn't really predict the spike of the resonance as the reality but it can give an approximation of how does the impedance vary and why is important to know this non ideal behaviour.

3 CHAPTER III. Control and switching parameters

3.1 PWM

Not only the components or layout are important to obtain an EMI friendly PCB. The control and switching strategy is equally important.

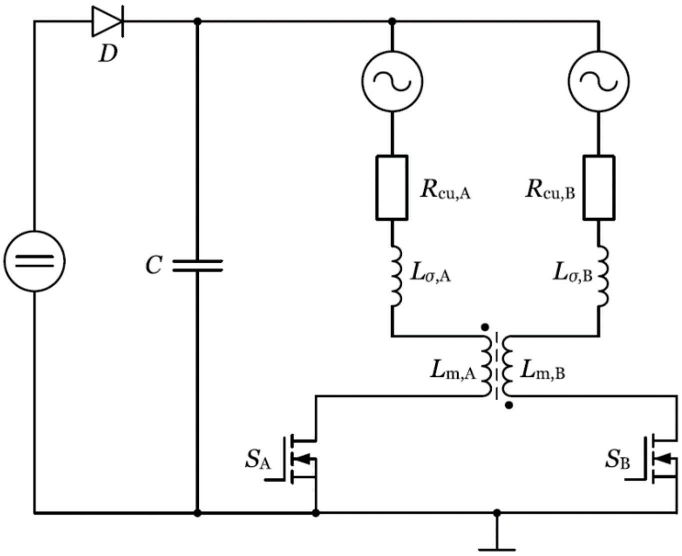


Figure 26: Wire diagram of the motor [23]

In this project, a brushless motor with bifilar winding is driven. A commonly used switching strategy is the **PWM** which stands for Pulse Width Modulation. It is a square wave with a constant frequency. The width of each pulse is the variable parameter. As explained in Section 2.4.1 the duty cycle is the percentage of time the pulse is 'high' compared to the period of the signal. This type of switching strategy is easy to implement but its main drawback regarding electromagnetic emissions is that when an electromagnetic test is performed, the switching frequency and its harmonics have a large impact on it.

Table 1: Motor parameters

Parameter	Value	Parameter	Value
$R_{cu,A}$	8,5 Ω	$R_{cu,B}$	8,5 Ω
$L_{\sigma,A}$	260 µH	$L_{\sigma,B}$	260 µH
$L_{m,A}$	2,8 mH	$L_{m,B}$	2,8 mH
C	16,6 µF	U_{DC}	12 V

3.2 PWM variations

There are some modifications in the PWM switching strategy, one of them is the change of the switching frequency randomly or every few changes in order to spread the frequency components. Two or more jumping frequencies can be selected in order to perform this strategy. Similarly, in order to smooth the frequency components even more, a complete band of switching frequencies can be used so there are no spikes in certain frequencies.

The last strategies are not entirely perfect because they only spread the frequency components and despite it can be enough to pass the electronic emissions test the device is still radiating.

3.3 Another alternative

In this case a new switching strategy can be used because of the high impedance of the motor winding and a relative low voltage. This strategy consists on turning on and off each MOSFET or winding every cycle. This can be used because the high resistance, 8,5 Ω of the winding and the input voltage around 12 V only allow a maximum current of 1,4 A through the winding which can be handled by the components and the winding. This way, each turn of the fan it is only switching each MOSFET on and off once. Which means that the total currant has a frequency of twice the spinning speed of the fan [24].

$$fHz = \frac{Fan\ speed\ [rpm] \cdot 2}{60} \tag{21}$$

Which with a speed of 5000 rpm equals to 166,66 Hz which is much smaller than the 22,89 and 34,33 kHz present in the original code. It is important to remind that this technique cannot be used in any motor because with higher input voltages, the current would be also higher and then probably it would be too much current to handle for the wiring. In those cases, other techniques have to be used.

4 CHAPTER IV. EMC Automotive standards

4.1 Norm

There are basically two norms that describe the measurement devices and the measurement settings and limits.

- EN 55016 or CISPR-16: Specification for radio disturbance and immunity measuring apparatus and methods.
- EN 55025 or CISPR-25: Vehicles, boats and internal combustion engines; Radio disturbance characteristics: Limits and methods of measurements for the protection of on-board receivers.

4.2 Detectors

When performing an EMI analysis, three different types of detectors are used. The peak detector, the quasi-peak detector and the average detector.

Peak detector: It is self-explanatory; it detects the maximum value seen during the time the scanner dwells a frequency. This is the fastest type of detector.

Quasi-peak detector: The EMI legislation usually uses quasi-peak detectors these detectors serves to detect the weighted peak value of the envelope of a signal. It weights the signals depending upon their duration and repetition rate. Its value is between the value of a peak detector and the average detector. [25]

Average detector: It provides the average amplitude of each signal component across its period. [26]

A preliminary test can be performed only with a peak detector in case of a lack of the quasi-peak detector or a lack of time. If the device tested is below the regulation’s limits using the peak detector it is almost sure that it will also comply when a quasi-peak detector is used because of its weighted measurement.

4.3 Conducted emissions measurements

All around the world products are being designed and manufactured, the way to test if these products comply with the regulations that are present in each region. For this matter, the noise coming from the power line must be attenuated otherwise it would add to the noise produced by the device and alter the measurements. The noise levels are also related to the impedance of the grid seen by the product, in order to make it impedance have the same value in any measurement site a device has been developed. This device is called LISN or *Line Impedance Stabilization Network*.

The emissions are evaluated over a frequency range from 150 kHz to 30 MHz. The impedance is only constant above 10 MHz where it is close to 50 Ω. In Figure 27 the impedance depending on the frequency can be seen.

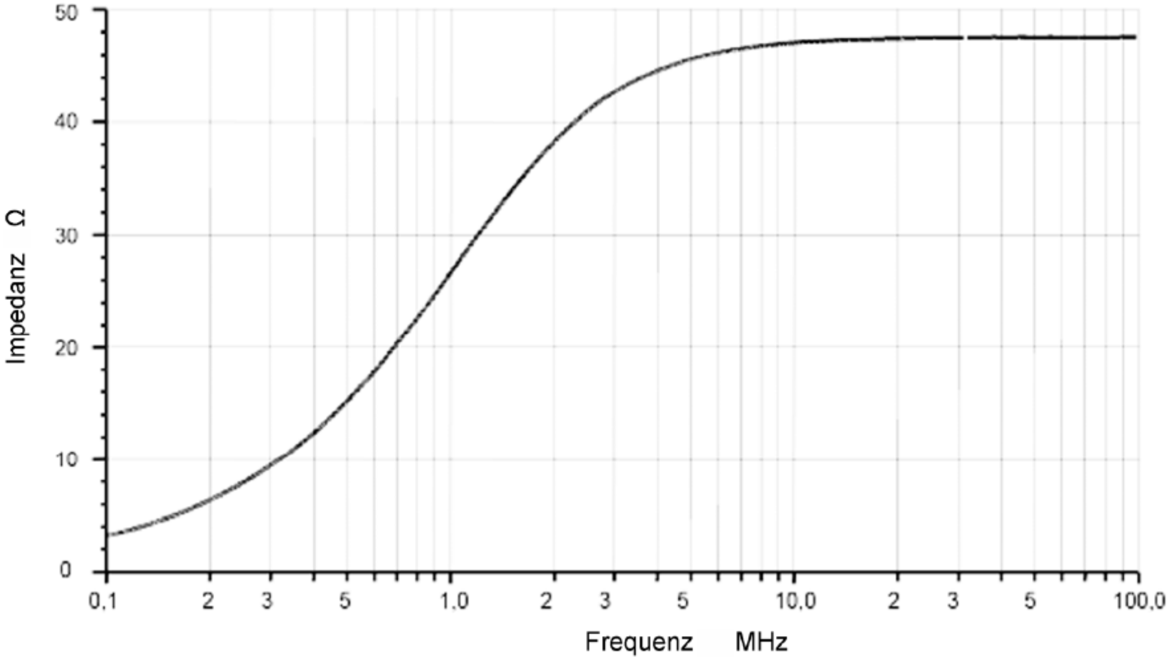


Figure 27: LISN impedance [27]

To sum up, the LISN has two main objectives:

1. Keep the impedance seen by the tested product the same in every measurement across the world made following the CISPR 25 standards.
2. Make sure that the noise from the grid does not propagate to the device so the measured conducted emissions are only the ones generated by the tested product.

How does the LISN work?

It is connected in series between the power outlet and the product.

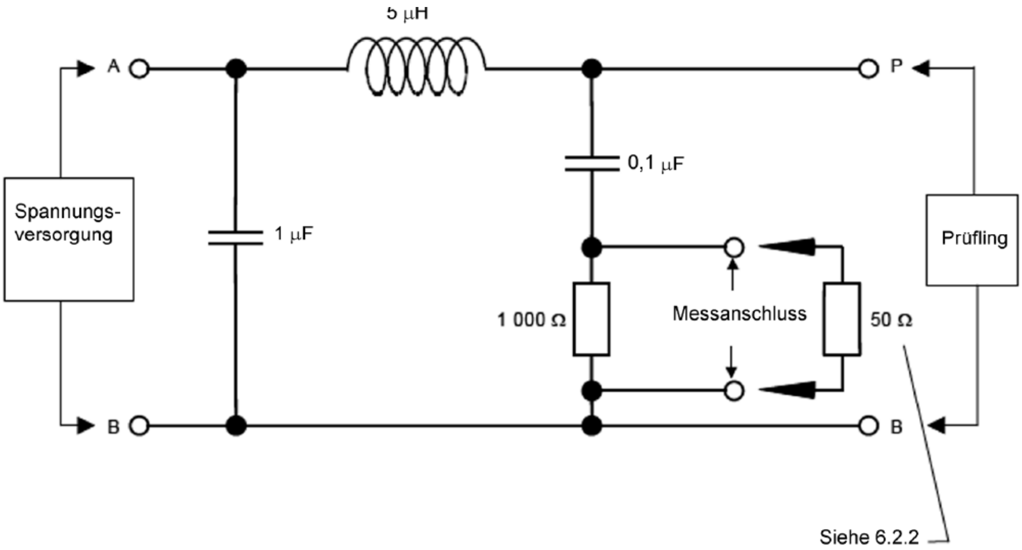


Figure 28: LISN [27]

The schematic of a LISN is shown in Figure 28, the task of the 1 µF capacitor and 5 µH inductor is to “filter” the noise coming from the power line so it does not contaminate the measurements. The 0.1µF capacitors is used to protect the test receiver from any overload. The 1 kΩ resistors are used to discharge the 0.1 µF capacitors if the 50 Ω resistors are removed. The 50 Ω resistors are placed in parallel with the 1 kΩ resistors.

If the range of frequencies where it analyses is taken into account and the impedance of the capacitors and inductors is calculated it results in:

Table 2: Impedance of passive components

	150 kHz	**30 MHz**
50 µH	47.1 Ω	9424.8 Ω
0.1 µF	10.61 Ω	0.053 Ω
1 µF	1.03 Ω	0.0053 Ω

In the lower frequencies, the order of magnitude is similar to 50 Ω but in higher frequencies it can be noted that inductors have a very high impedance and capacitors a very low impedance. Therefore, capacitors can be seen as short circuits and inductors as open circuits over all the frequency range of measurements.

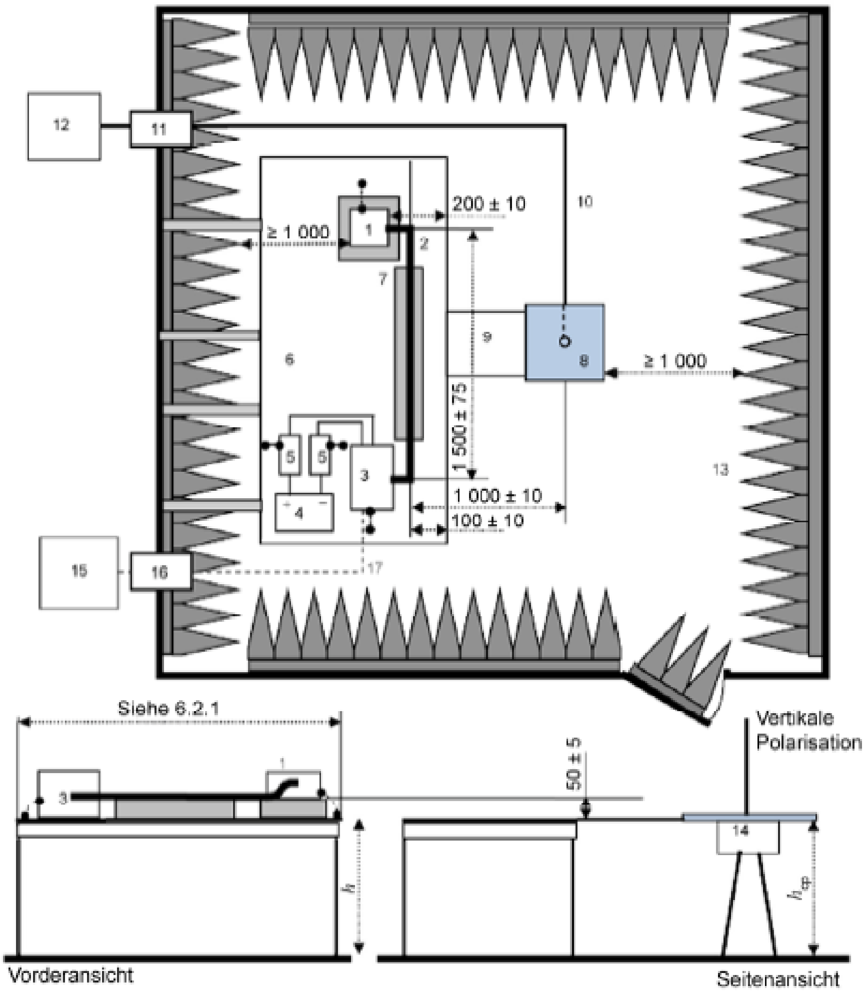
The measurement is made between the terminals of the 50 Ω resistor, therefore, the phase current can be written as:

$$\hat{I} = \frac{\hat{V}}{50} \tag{22}$$

This way, the voltages measured in the test are directly related to the conducted noise coming out the product.

4.4 Radiated emissions measurements

Measurements of radiated interference emissions must be carried out in the absorber room, in the case of the university a shielded chamber, in order to eliminate the high values of external disturbances from electrical devices and equipment as well as from radio transmitters. [27]



Legende

1 Prüfling (lokal mit Masse verbunden, falls entsprechend dem Prüfplan erforderlich)
2 Prüfkabelbaum
3 Lastnachbildung (Anordnung und Masseverbindung entsprechend 6.5.2.6)
4 Spannungsversorgung (Anordnung freigestellt)
5 Netznachbildung (AN)
6 Bezugsmassefläche (mit dem geschirmten Raum verbunden)
7 Unterlage mit niedriger relativer Permittivität ($\varepsilon_r \leq 1{,}4$)
8 Stabantenne mit Gegengewicht (typische Maße 600 mm × 600 mm)
9 Masseverbindung (Verbindung über die volle Breite zwischen Gegengewicht und Bezugsmassefläche
h = (900 ± 100) mm
h_{cp} = h + (+10/– 20) mm

10 hochwertiges Koaxialkabel, z. B. doppelt geschirmtes Kabel (50 Ω)
11 Durchführungsanschluss
12 Messgerät
13 HF-Absorbermaterial
14 Antennen-Anpassungseinheit (der bevorzugte Einbauort ist unterhalb des Gegengewichts; wenn sie oberhalb des Gegengewichts liegt, muss die Basis des Antennenstabs in der Höhe der Bezugsmassefläche liegen
15 Anregungs- und Überwachungssystem
16 Lichtwellenleiter-Durchführung
17 Lichtwellenleiter

Figure 29: Radiated emissions setup with vertical antenna [27]

According to the regulation, for a frequency range from 150 kHz to 30 MHz, a 1 m monopole antenna with a nominal output impedance of 50 Ω must be used.

The whole test rig is placed over a conductive surface which is the ground plane. The device tested should be placed on a base of non-conductive material with low relative permittivity ($\varepsilon_r \leq 1{,}4$) at a height of (50 ± 5) mm above the reference ground plane.

The side of the device tested closest to the edge of the reference plane shall be located at a distance of (200 ± 10) mm from the edge of the reference plane as can be seen in Figure 29.

The length of the part of the test harness parallel to the front face of the reference ground plane shall be (1 500 ± 75) mm.

The long section of the test harness shall be located parallel to the edge of the reference ground plane facing the antenna at a distance of (100 ± 10) mm from the edge. The arrangement of the test requires that the corner angle of the cables to be between 90^o and 135^o.

For measurements of radiated emissions, the screened space shall be sufficiently large to ensure that neither the test object nor the measuring antenna is closer than 1 m to the walls or ceiling or to the nearest surface of the absorber material used. No part of the radiating elements of the antenna shall be closer than 250 mm to the ground.

The distance between the longitudinal part (1 500 mm length) of the wiring harness and the reference point of the antenna shall be (1 000 ± 10) mm. In the case of biconical antennas, no part of the antenna shall be closer than 700 mm to the harness or to the test object.

The test object must be operated under typical load and other conditions in the vehicle, which must also be such that the highest emission of interference occurs. These operating conditions must be uniquely defined in the inspection plan in order to ensure that the supplier and customer are able to carry out identical checks. The orientation(s) of the test object when measuring radiated emissions shall be specified in the test plan.

5 CHAPTER V. EMC compliant layout design

Finally, after all the theoretical aspects of electromagnetic emissions this chapter's goal is to explain how to design a PCB layout in order to be EMC compliant.

5.1 EMC considerations

First of all, some considerations about which design techniques can be done to the given board in order to reduce the emissions.

1. **Separate areas for inputs and outputs and the high frequency components**
   The connector of the board is a way out for the conducted and radiated emissions, therefore, it is important to try to keep high frequency traces and components as far as possible from the traces that lead to the connector. This way, possible crosstalk between such traces will be avoided and high frequency components will not escape the board through the connector.

2. **Differentiate which traces are "electromagnetically relevant"**
   Before starting the board layout one has to be aware of which traces are the most critical to the board behaviour. For example, two traces of the same length but one being a 5 V DC 100 mA trace for the microcontroller and the other being a square waveform of 12 V of amplitude and 1 A with a frequency of 20 MHz are not going to radiate the same way. The second trace is much more critical to the system; that is why its length should be kept as short as possible.

3. **Minimize trace length as much as possible**
   As said in the last point, traces must be kept as short as possible so they cannot act as an effective antenna.

4. Cornering

When routing a corner, avoid a 90° corner, they will introduce small impedance discontinuities because the trace's width changes in the corner. In order to minimize this effect, there are other options than the 90°. Nevertheless, the effect of these discontinuities is insignificant unless, as a general rule, the rise times are smaller than 0,1 ns. Despite that, it has to be taken into account. [28]

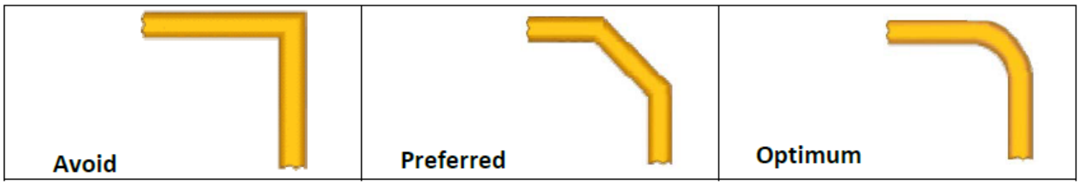


Figure 30: Cornering geometries [28]

5. Place decoupling capacitors as close as possible to the voltage pin

The use of decoupling capacitors has been explained in Section 2.7.2. It is really important that they are placed as close as possible to the pin. Otherwise, the trace from the last decoupling capacitor to the pin can be too long and pick up noise from nearby traces which will make the decoupling capacitors useless because the trace between the actual capacitor and the pin will pick up noise. Usually, more than one decoupling capacitors are used in parallel. If that is the case, the one with the less capacitance is the one that should be closer to the pin. This is done in order to have a better behaviour across multiple frequencies and not to be that much affected by the resonance frequency. [29]

For example, the impedance of two capacitors in parallel, one with a capacity of 47 µF and 100 nF would be the following

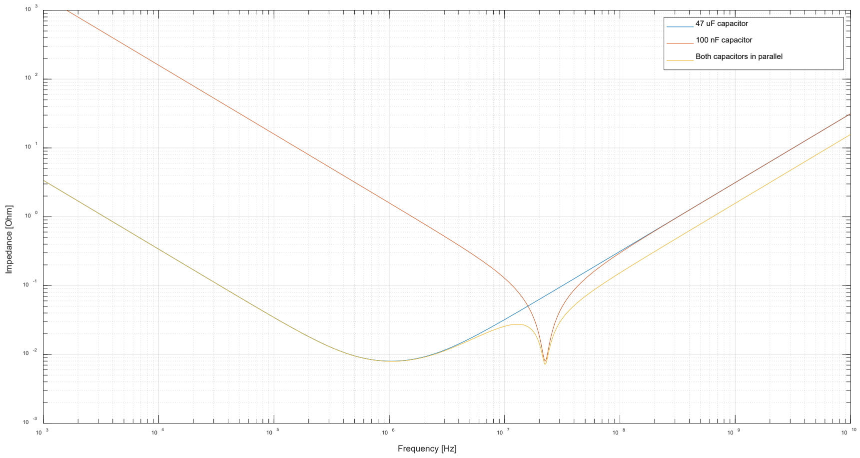


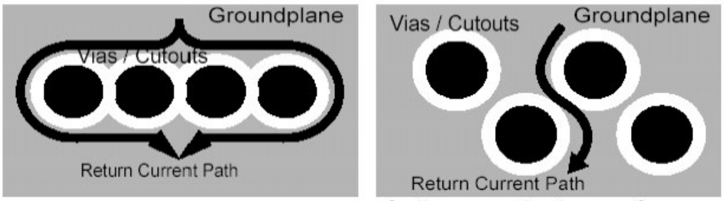
Figure 31: Impedance of different capacitors in parallel

In order to work properly, decoupling capacitors have to be placed in a way that the current has to flow through them. Which essentially means that they have to be physically placed between the voltage source and the voltage pin, not after or in a way that the current can find and alternative path.

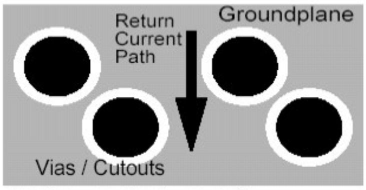
6. **Minimize loop areas**
   As explained in Section 2.5.2, large loop areas act as antennas for the radiated emissions that is why they have to be kept as small as possible. Note that there can be loop areas in the same plane of the board and also in its orthogonal planes. These loop areas are caused by the vias and the ground plane beneath the power traces.

7. **Try not to place vias in traces**
   Vias are necessary in multilayer boards. They allow the current to flow from one layer to another. In order to do that a hole is drilled in the board and then coated with a conductor material. But if they are placed in the middle of a trace, the hole reduces the width of the trace locally and it could produce some unwanted behaviours.

8. **Minimize slots in ground planes**
   When routing, sometimes the ground plane has a pad, a via or a trace which is not connected to ground. This leaves a gap in the ground plane which at the same time makes it more difficult for the current to find a return path. In essence, the return path impedance is increased. [30]



worst current return path **better current return path**



best current return path

Figure 32: Via placement [30]

5.1.1 Layers

Boards have two sides where components can be placed but between these two sides there are layers. The majority of simple boards have two layers. They are easy and cheap to manufacture and if the electronic device is simple, with not many traces it is the best cost friendly option. If high frequency components are placed in theses boards and they are keen to be worse than the same layout in a PCB with more layers. [31]

Higher number of layers provide PCB with more options about routing, power and ground planes creation and shielding.

Three important factors about layer stack-up:

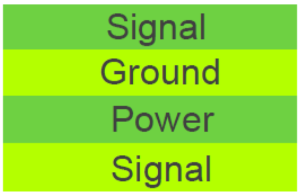
- **The count of layers**

  It is important to know how many signals have to be routed and the cost of increasing the number of layers. Equally important is to know the frequency and speed of the signals in order to be able to predict if more layers would be necessary for shielding purposes.

  In general, multi-layer PCBs provide significant reduction in radiated emission over two layer PCBs. As a rule of thumb:

  "A four-layer board will produce 15 dB less radiation than a two-layer PCB"

- **The spacing between the layers**
  In order to achieve a capacitance between planes that doesn't contribute to the emissions.
- **The sequence of the layers**
  The most common four-layer board configuration is:



Signal
Ground
Power
Signal

Figure 33: Four-layer stack-up [31]

This layer stack-up is similar to the one used in the original board and therefore, to the one used in the prototypes. This stack-up has a signal layer next to a reference plane which is good in order to keep loop areas as small as possible. A bad characteristic is that the outer planes are not ground planes and the signals are not shielded. In the next section more in depth information will be given about each prototype stack up.

5.2 Prototypes

The board provided by the manufacturer is a 4-layer PCB. The idea of the project is to manufacture different prototypes with a different layout in order to know if the components layout is really helpful when performing and EMI test.

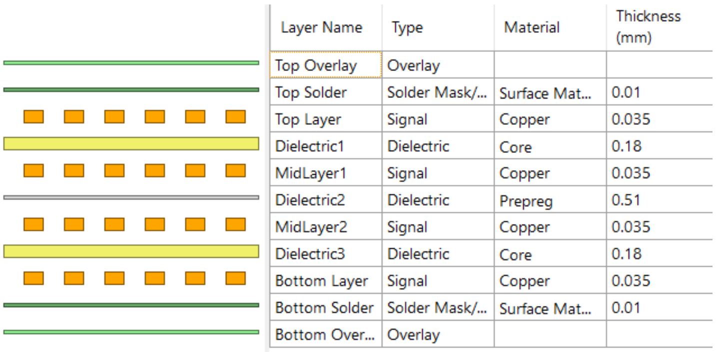
In order to understand some of points, it is important to notice that the motor is mounted on top of the PCB. As explained before, it is a BLDC and it needs a hall sensor to operate correctly. The position of this hall sensor is crucial because it detects the motor spinning via magnetic induction and has to be correctly placed. Its position will be a restriction in the layout.

5.2.1 Prototype 1

This prototype uses the exact same components as the original one and they have only been rearranged in a different way following the layout design rules previously exposed.

Layer stack-up

The layer stack-up of prototype 1 is the same as the original board.



Layer Name	Type	Material	Thickness (mm)
Top Overlay	Overlay		
Top Solder	Solder Mask/...	Surface Mat...	0.01
Top Layer	Signal	Copper	0.035
Dielectric1	Dielectric	Core	0.18
MidLayer1	Signal	Copper	0.035
Dielectric2	Dielectric	Prepreg	0.51
MidLayer2	Signal	Copper	0.035
Dielectric3	Dielectric	Core	0.18
Bottom Layer	Signal	Copper	0.035
Bottom Solder	Solder Mask/...	Surface Mat...	0.01
Bottom Over...	Overlay		

Figure 34: Layer stack-up

Note that the Dielectric 2 is thicker than the other dielectrics. This way, the overall width of the layer is enough to withstand mechanical tensions but more important, signal planes are closer to reference planes and they can couple together to ensure a small loop area for the returning current.

The Top Layer contains all the electronic components but the connector. It also contains some signal traces. Mid Layer 1 is a ground plane. Mid Layer 2 is an auxiliary plane for some signal traces and some power traces from the motor. And finally, the Bottom Layer contains the connector and a bunch of measurement and programming points.

Components arrangement

The first thing to notice about this prototype is that the motor stator has been rotated 45°. Which means that the location of the hall sensor has been changed in order to keep the same relative position to the motor.

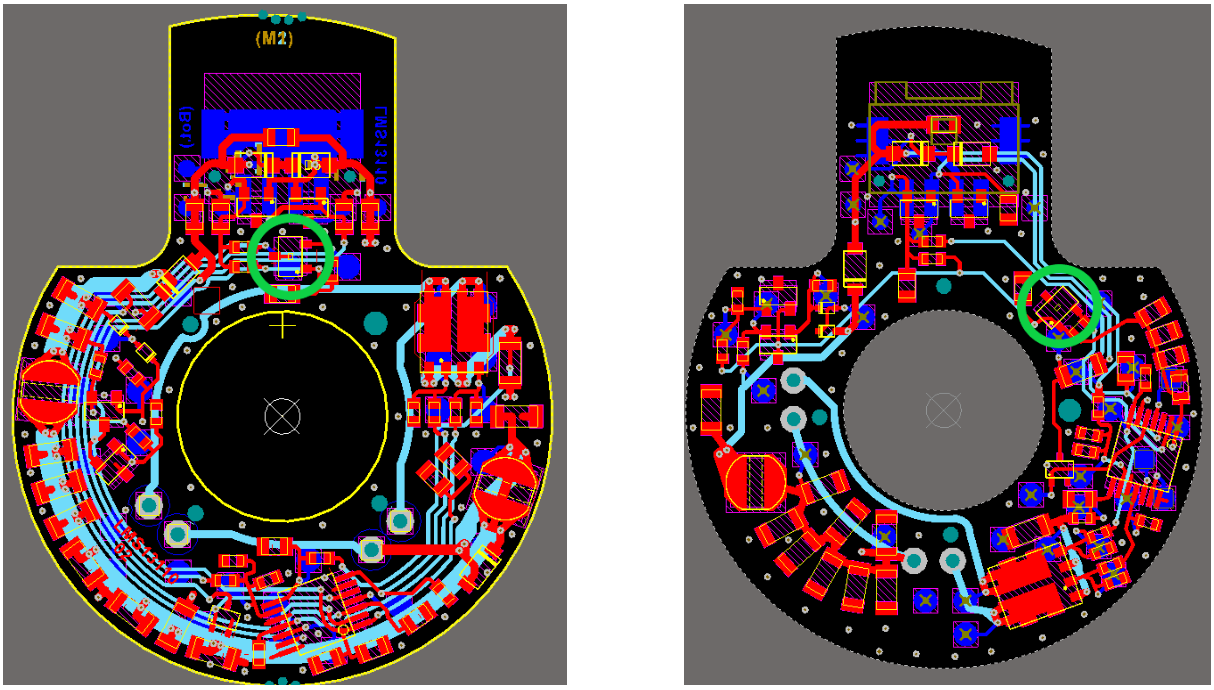


Figure 35: Original hall sensor position (Left). Prototype 1 hall sensor position (Right)

Another issue that has been improved was the long high current, high frequency traces from the MOSFETs to the motor. This traces are electrically critical and have to be as short as possible. The MOSFETs and microcontroller have switched positions in order for the MOSFETs to be closer to the motor pins.

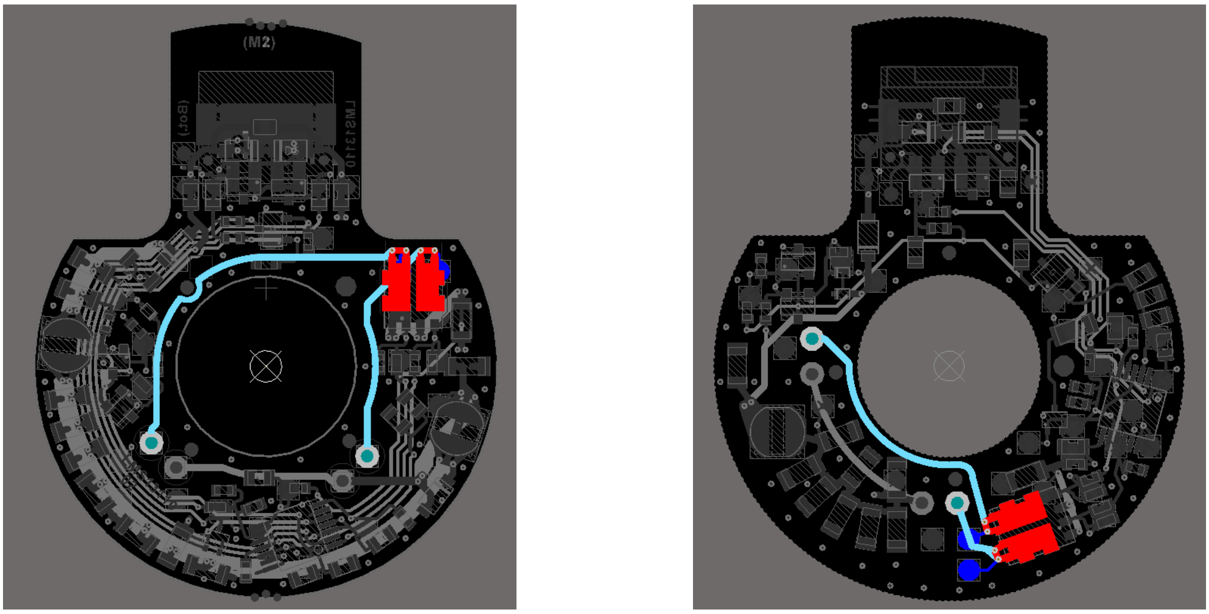


Figure 36: Original motor traces (Left). Prototype 1 motor traces (Right)

Another improved characteristic is the placement of decoupling capacitors. In the original board there are two out of the four capacitors that are connected to the voltage pin of the microcontroller that are "useless". As said before, the decoupling capacitors should be as close as possible to the voltage pin and also the current flow must be able to flow through them before entering the pin. Which means that its pad should be in the voltage trace.

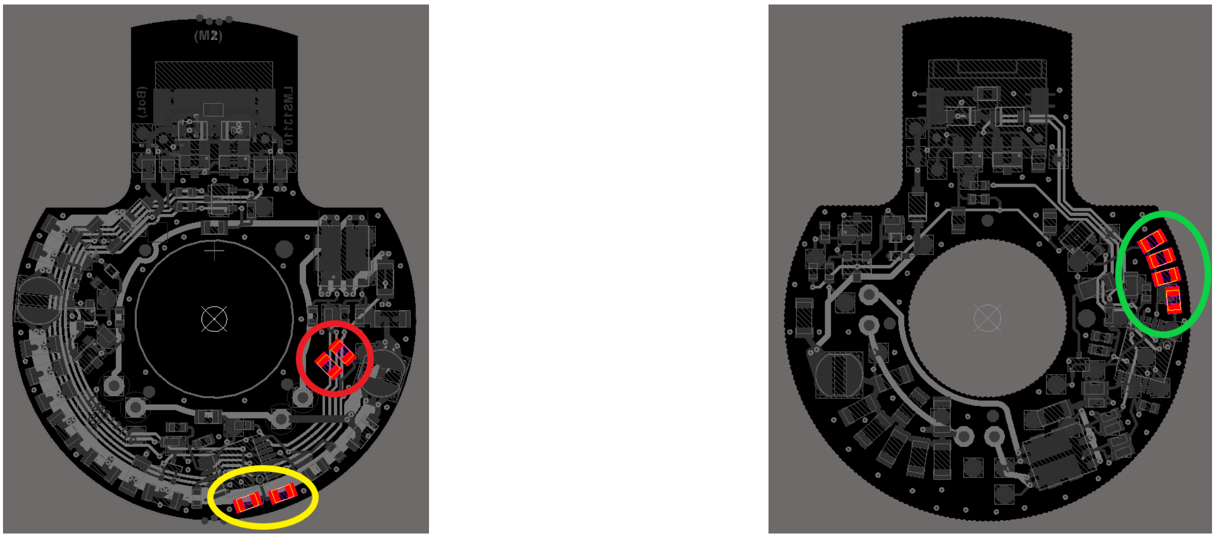


Figure 37: Original board decoupling capacitors (Left). Decoupling capacitors in prototype 1 (Right)

As it can be seen in the picture, the capacitors circled in red in the left picture are not even in the path between the voltage source and the voltage pin which mean that they are almost useless. The two other capacitors are better placed but far from the optimal way. The current must deviate from the main trace in order to flow through them.

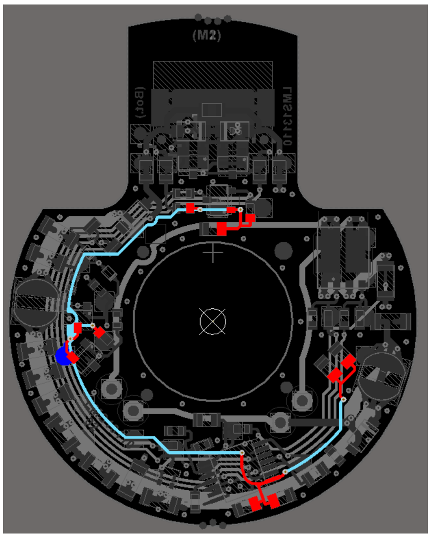


Figure 38: 5 V net in the original board

In the new prototype, the same four capacitors are placed as close to the voltage pin as possible and the current is forced to flow through each pad of the capacitors.

A similar problem was found in the LC filter; the main trace is in the Mid-Layer 2 but the components are in the Top Layer and connected to the trace using two vias. This method is not the best because vias generate an inductance and also it is more difficult for the current to flow through them and into the capacitors pads. The strategy use is that the trace is now on the Top Layer which can be worse in terms of radiation because it is more exposed to the environment but now the filter should perform better than before.

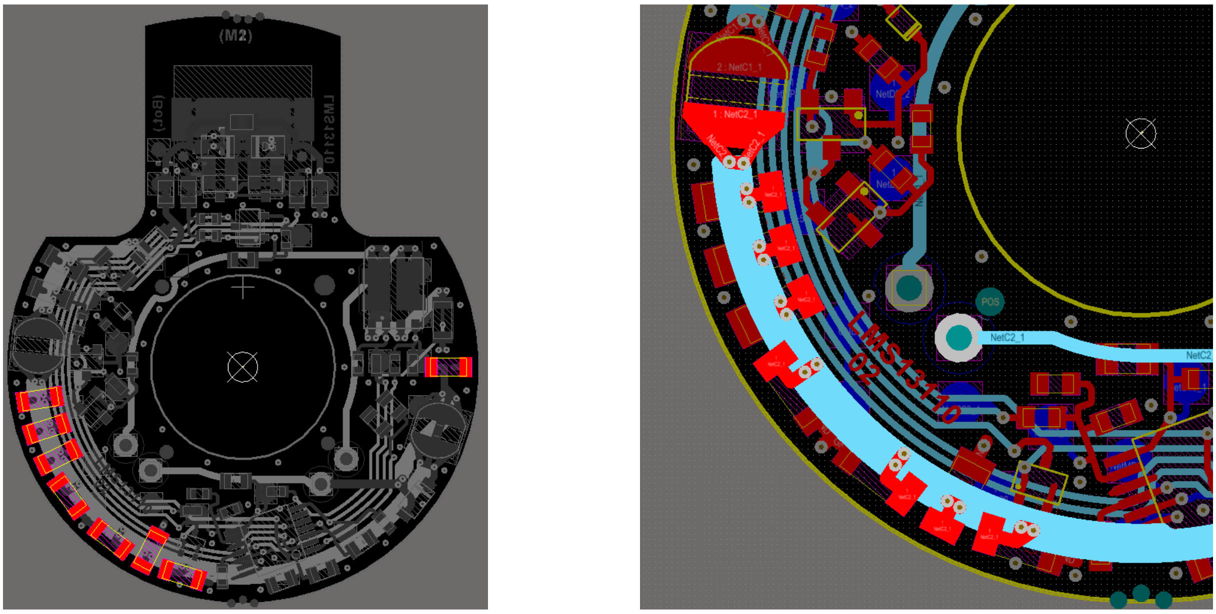


Figure 39: Original board filter's capacitors placement

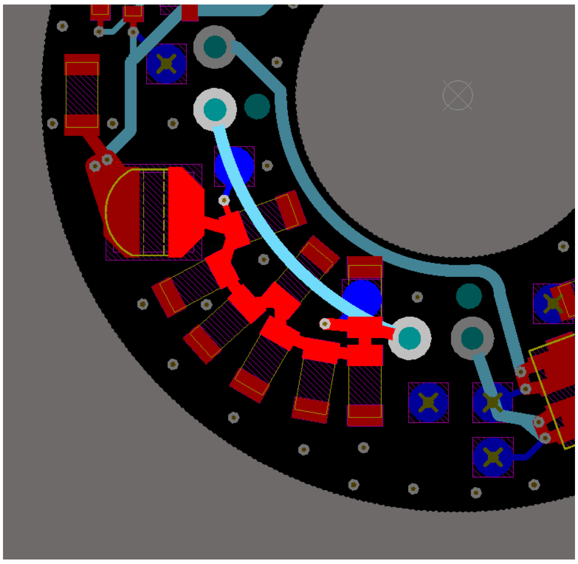


Figure 40: Prototype 1 filter's capacitors placement

As seen in Figure 40, the new trace goes from pad to pad in order to force the current to flow through the capacitors.

5.2.2 Prototype 2

This prototype also has four layers but now some diagnosis components have been removed from the PCB because they are not necessary. The removal of these components should not affect the overall electromagnetic behaviour because they do not perform any high frequency task.

Some of the other components are also different. In the original board there are a group of components that step down the voltage form the battery to 5 V in order to feed the microcontroller. One of the components used is a regulated diode that is quite expensive. Also, the use of several components to perform this task makes the layout quite difficult because a lot of traces have to be routed and there is not much room for all the components. That is why in this prototype, all these components have been substituted by a linear voltage regulator. This option is cheaper and in theory shouldn't affect the EMC.

Some of the most critical traces are the control signals from the microcontroller to the MOSFET's gate and the wide power traces that go through the MOSFETs and to the motor winding. Decreasing the length of all these traces might help on having better results on the test. That is why instead of using a double MOSFET in the same IC, two independent MOSFETs have been selected so the routing can be easier and shorter.

Layer stack-up

The layer stack-up is exactly the same as the prototype 1.

Components arrangement

As before, the motor and hall sensor positions will be commented first. The layout plan for this prototype led to the decision of putting back the motor in its original location as well as the hall sensor.

The removal of the diagnosis components and the substitution of the various elements that stepped the voltage down for a linear voltage regulator means that now there is more space and less traces to route.

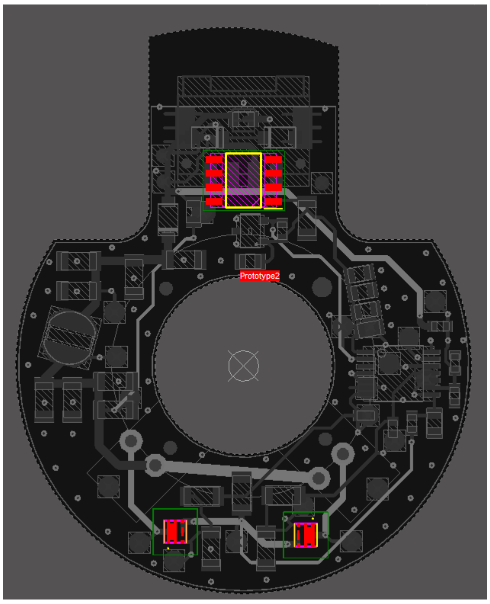


Figure 41: Voltage regulator and MOSFETs position

Filter placement theory

As commented in Section 2.3 about Filters, the selected filter for this prototype is a π-filter which should perform better than the previously existing LC filter. There are general rules of filter components placement in order to minimize coupling between the components and leading to a bad EMC performance. [32]

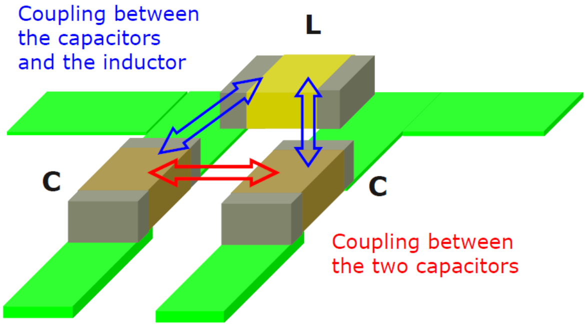


Figure 42: Coupling between components of CLC filter [32]

The example is from a study performed by Dr. Winkler from TU-Graz in collaboration with Infineon. They studied the impact of the layout in a CLC filter.

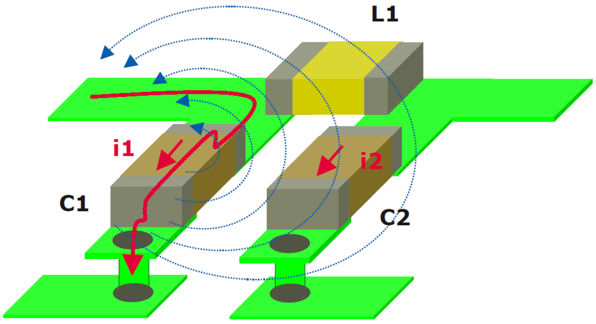


Figure 43: Coupling between capacitors [32]

If both sides of the filter are parallel to each other, they can couple and then, noise from one side can jump to the other side without filtering.

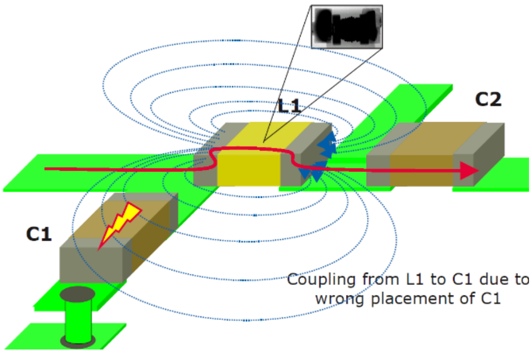


Figure 44: Coupling between C1 and L1 [32]

For this reason, placing the capacitors 90° to each other is a good solution to minimize the coupling. Thera can also appear coupling between L1 and C1 because C1 is not parallel to the flux lines of L1.

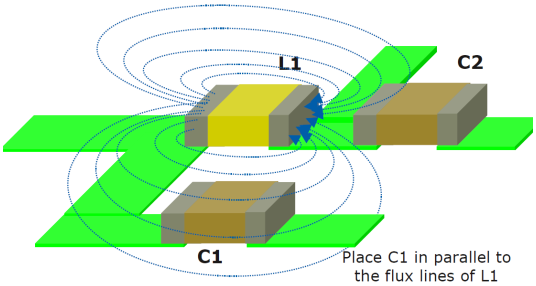


Figure 45: Optimal layout [32]

With this configuration, all the problems should be fixed. Note that C1 and C2 are again parallel to each other but with an offset so the coupling should not be high as before.

The actual filter of this prototype has more than one capacitor in each side of the inductor and the inductor has the coil oriented in vertical. This means that not all the previous rules can be applied.

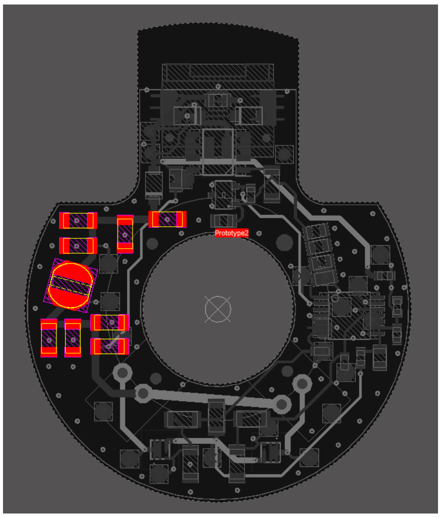


Figure 46: CLC filter layout

Regarding the coupling between capacitors, two capacitors of each side are placed parallel to each other but they are quite far away and with a little bit of offset so there should not be a significant coupling. Regarding placing the capacitors parallel to the flux lines of the inductor, in this case it is impossible because the flux lines of the inductor are perpendicular to the board and the capacitors have to be surface mounted. Hopefully, they are far enough and will not exist coupling.

We should not forget that a motor stator is mounted in this PCB, the motor windings have a much higher impedance and the flux generated by the windings will be higher than the one generated in the inductor of the filter.

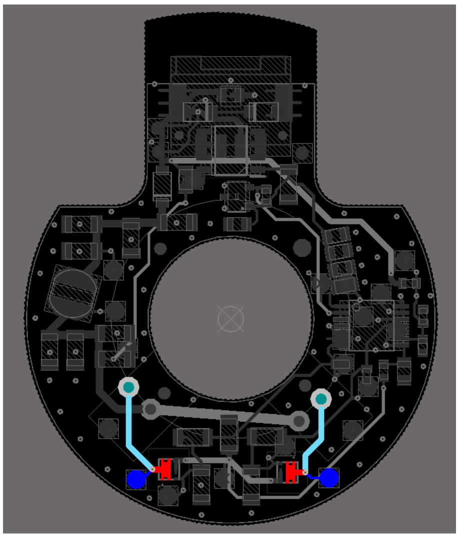


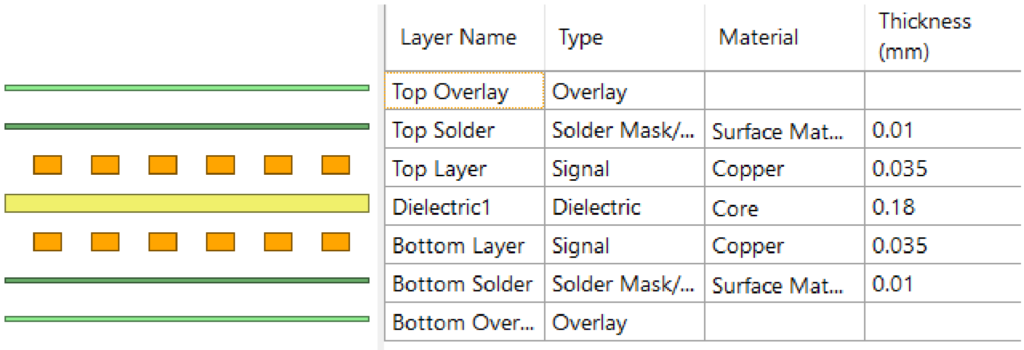
Figure 47: Motor traces of prototype 2

As it can be seen in prototype 2, the motor traces are even shorter. This should improve the emission performance. In order to test different values of resistors and capacitors for the MOSFET's gates, a 1206 package has been selected so it is easier to solder by hand. This will allow us to control the rise/fall times of the MOSFETs and thus reduce the emissions.

5.2.3 Prototype 3

This prototype has the exact same layout as Prototype 2 but it is a 2-layer PCB. That way we will be able to test the impact of having 2 or 4 layers. The routing had to be slightly modified because there where less layers to use but it is essentially the same.

Layer stack-up



Layer Name	Type	Material	Thickness (mm)
Top Overlay	Overlay		
Top Solder	Solder Mask/...	Surface Mat...	0.01
Top Layer	Signal	Copper	0.035
Dielectric1	Dielectric	Core	0.18
Bottom Layer	Signal	Copper	0.035
Bottom Solder	Solder Mask/...	Surface Mat...	0.01
Bottom Over...	Overlay		

Figure 48: Layer stack-up for the third prototype

Both Mid layers have been deleted. There are only two signal layers and there is not any solid ground plane.

In prototypes 1 and 2, the grounding is performed using Mid Layer 1 is a solid ground plane. It has no traces in it although it has some holes due to the vias. In the other planes, a ground section has been created, following the board shape. All the grounds of each plane are connected together using via stitching and also placing some manual vias where necessary to ensure that there is always a good connection minimizing the loop area and the impedance.

6 CHAPTER VI. Prototypes results

Prototypes changes

There is an important aspect to address before starting the results comparison. The original inductor was a 47 µH but when designing the prototypes a mistake was made. The inductor was placed too close to the centre and now the rotor of the motor is touching the inductor and it doesn't rotate freely. To fix it, the inductor has been changed for a 4,7 µH, in all the prototype's boards and original boards, which is shorter. In order to have a reliable comparison between prototypes and original board, the inductor has been replaced in all the boards.

Manufacturing mistakes

Due to a manufacturing error in Prototype 3, the trace which carries the control signal for one of the MOSFETs has been cut. Therefore, it never switches on. For these reason, comparing the results obtained from Prototype 3 cannot be entirely done.

The motor has a bifilar winding and with this mistake, one winding is not working. This is not a problem if the load of the motor is low because the voltage would not drop a certain threshold. Remember that the nominal voltage is 13,5 V and the operating voltage range is from 8 to 16 V.

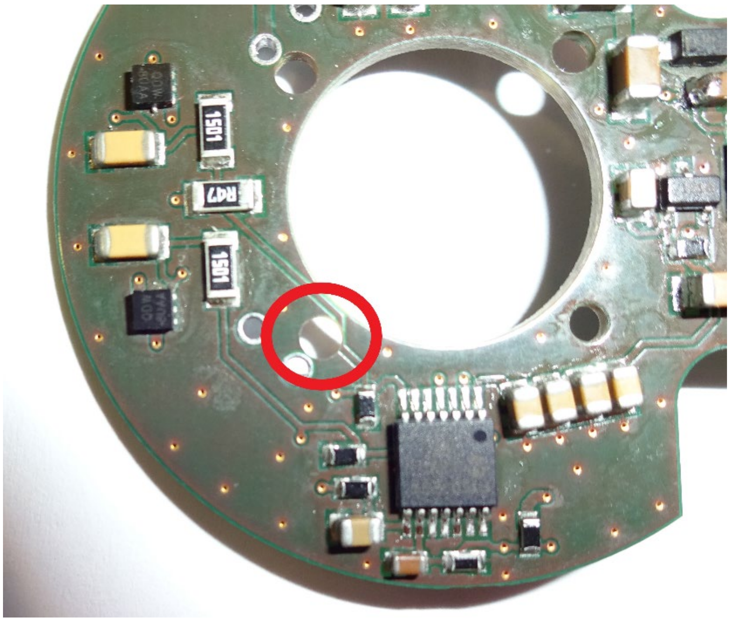


Figure 49: Prototype 3 manufacturing mistake

6.1 Comparing results

6.1.1 Prototype 1 vs reference

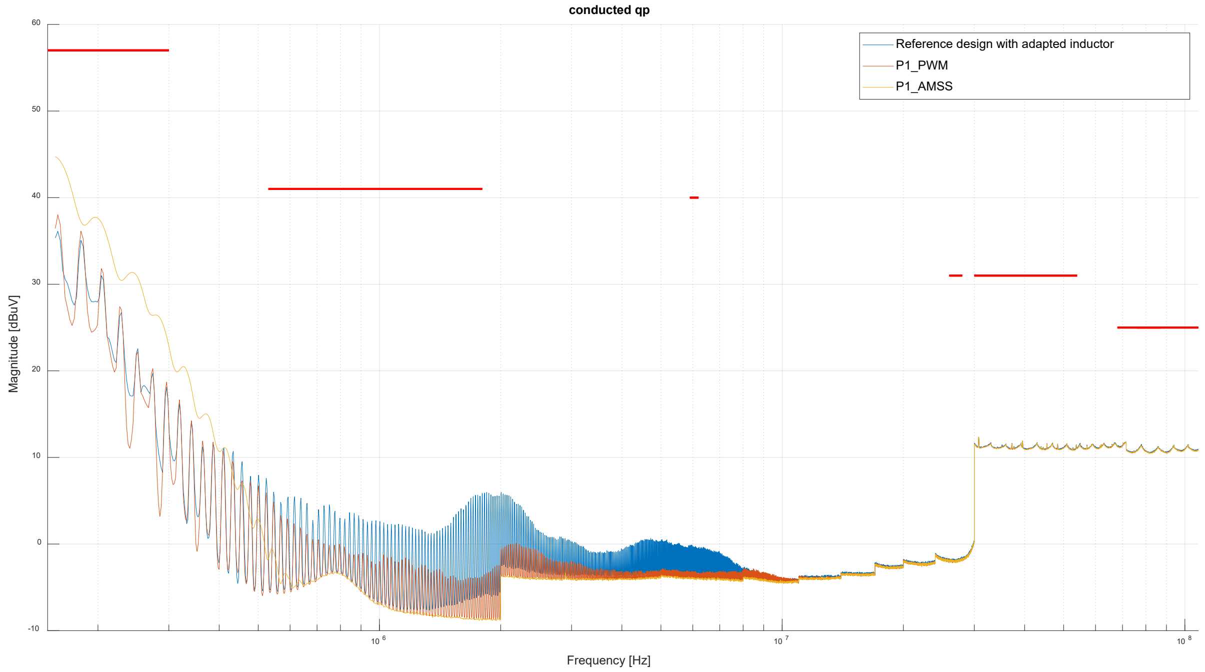


Figure 50: Conducted emissions of prototype 1 versus original board

The blue line in Figure 50 is the reference value from the original board with a switching strategy of FHPWM. Lines in orange and yellow are the ones from prototype 1. The orange one is obtained using the same switching strategy and the yellow one using the strategy explained in Section 3.3. This strategy is called AMSS which stands for Angle Modulated Switching Strategy [24].

As it can be seen, the new layout with the same switching strategy decreases the overall conducted emissions between 500 kHz and 7 MHz.

The AMSS switching strategy increases the conducted emissions in the lower frequencies of the tests but decreases them in frequencies above 500 kHz. This strategy is meant to be really effective in radiated emissions as it will be seen next.

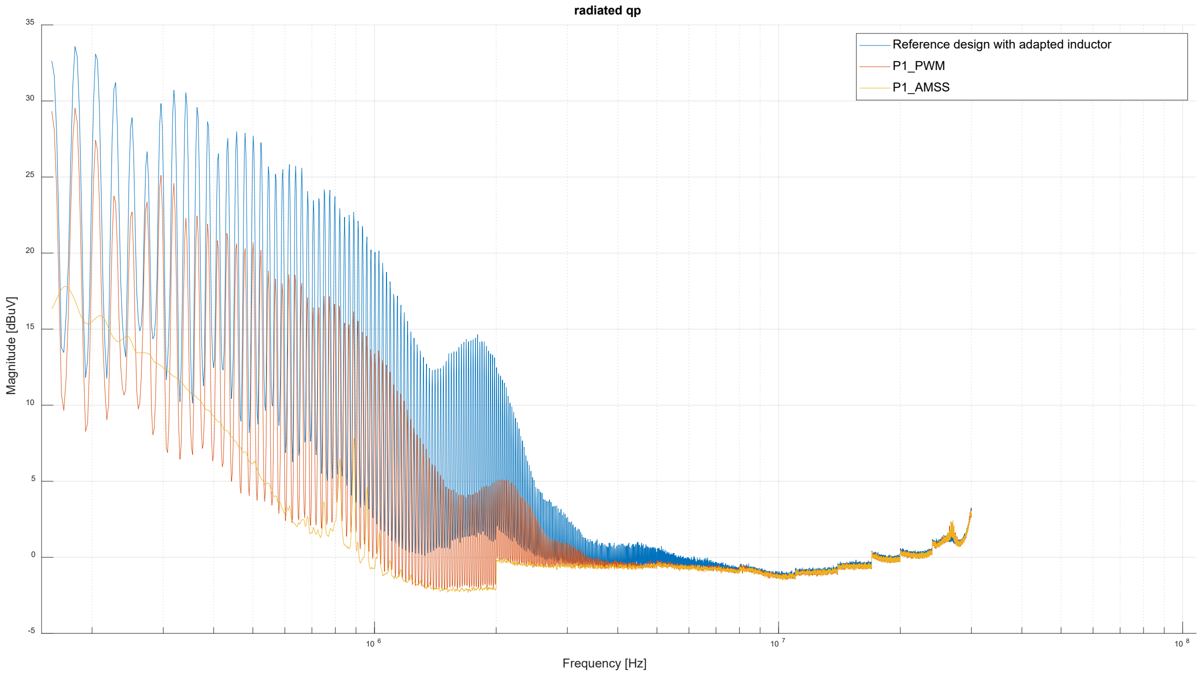


Figure 51: Radiated emissions of prototype 1 versus original board

Radiated emissions of the original board were above the limits of the regulations EN55025 (in red). The new layout of prototype 1 has a better performance in radiated emissions too.

The AMSS switching strategy is much better than the PWM switching strategy as it can be appreciated in the figure. As said before, this switching strategy performs really well in the radiated emissions test but it cannot be used with every device.

It seems that the motor winding and the magnets rotating at high speed close to the connector traces are inducing a voltage thus incrementing the conducted emissions. In order to prove that a capacitor was soldered between the pins of the connector in order to filter out the possible noise. This was the result:

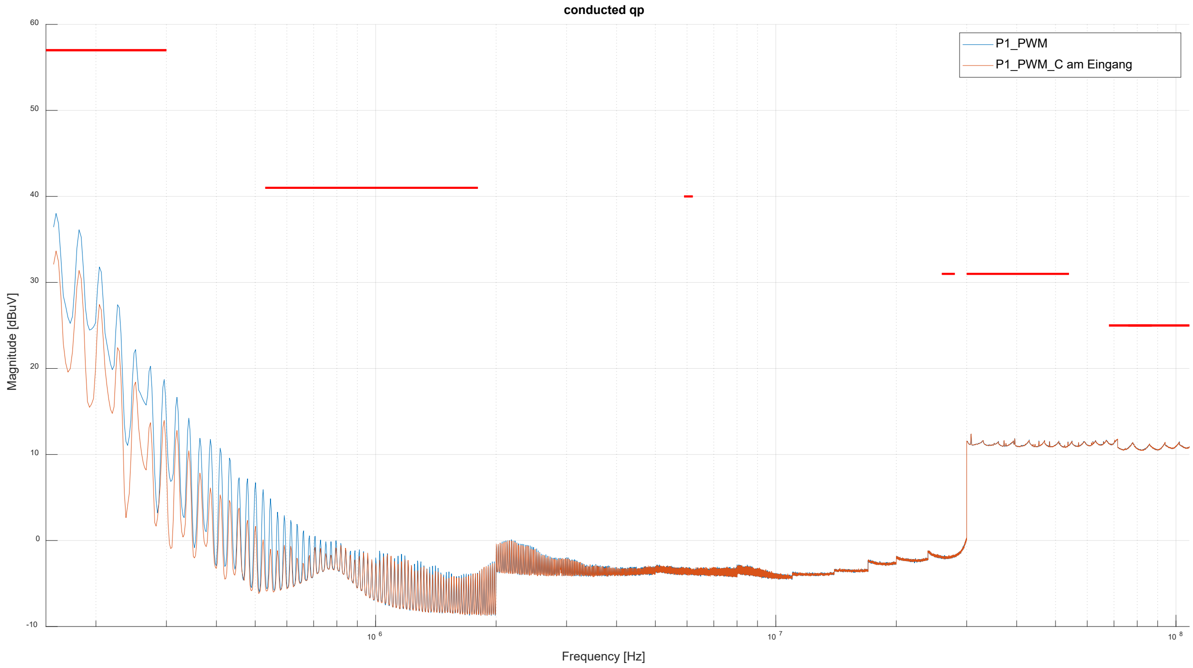


Figure 52: Conducted emissions of prototype 1 with capacitor soldered between connector pins

This results confirm what was thought it was happening. In the schematics there was already a decoupling capacitor connected to the connector pin but it seems that it was too far away.

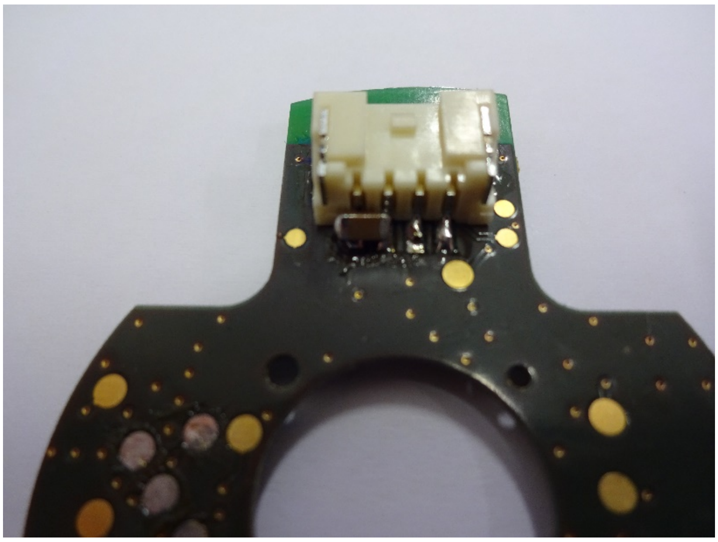


Figure 53: Location where the capacitor was soldered in order to perform the tests

6.1.2 Prototype 2 vs reference

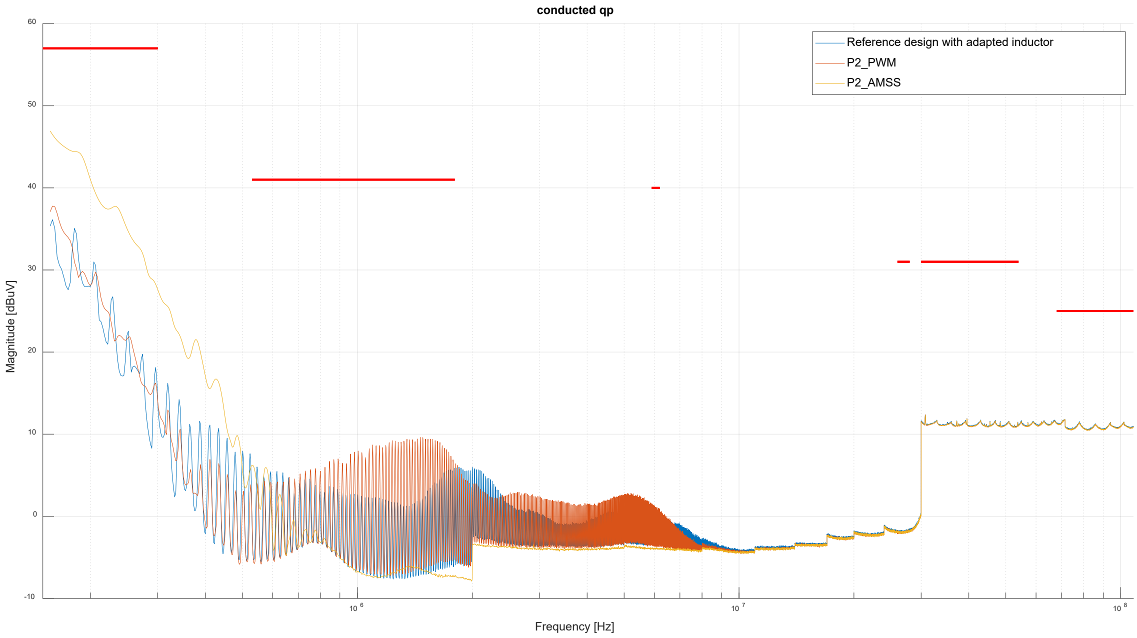


Figure 54: Conducted emissions of prototype 2 versus original board

The performance of prototype 2 in the conducted emissions is similar to the original board but definitely not better. The AMSS switching strategy has the same effect as previously, it is higher in low frequencies but better at higher frequencies. Due to the change in the inductor inductance, the filter doesn’t perform as planned.

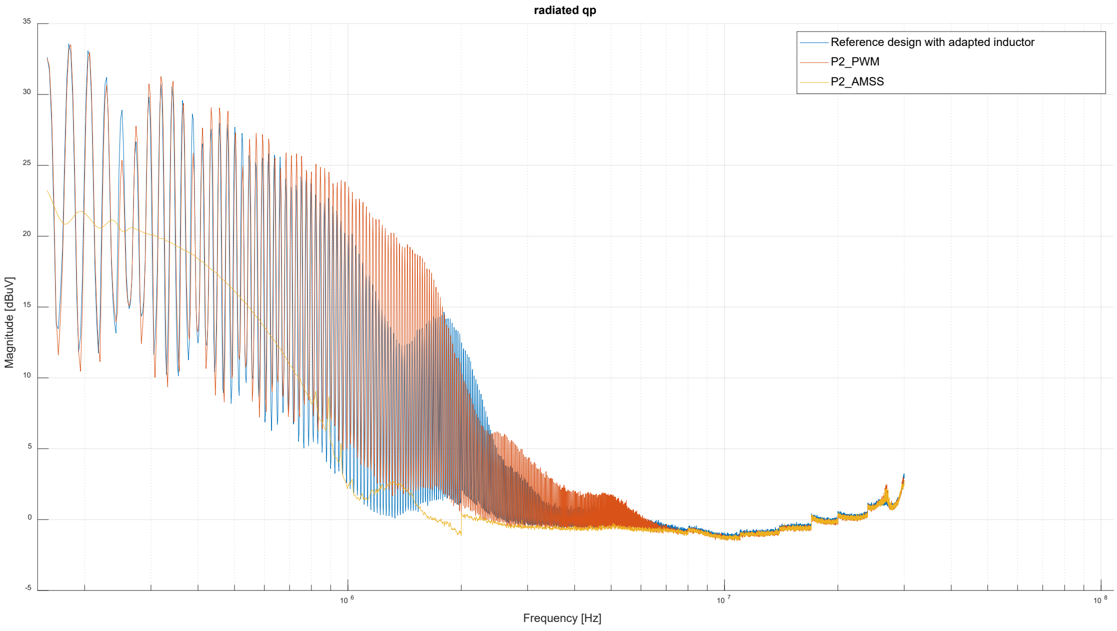


Figure 55: Radiated emissions of prototype 2 versus original board

Prototype 2 is not performing as expected in the radiated emissions tests. The level of emissions is not lower than the original board but higher. As a reminder, the differences between both prototypes were that prototype 2 has a different layout, a different type of filter and different components.

The most likely reason why it happens is because of the different components, precisely the MOSFETs. The layout has been designed following EMC friendly rules and the π-filter should perform better than the previously used LC filter. The importance of having slow rise/fall times has been pointed out previously too. The MOSFETs selected for prototype 2 have similar electrical capabilities but have faster rise/fall times. For this reason, it is possible that these faster rise/fall times are the ones increasing the emissions.

When deciding if this prototype could be used in the real world the answer is: it depends. If the AMSS switching strategy is used, then yes. If it is programmed with a PWM switching strategy, then the answer is no because it would exceed the limits of the regulations.

6.1.3 Prototype 2 vs Prototype 3

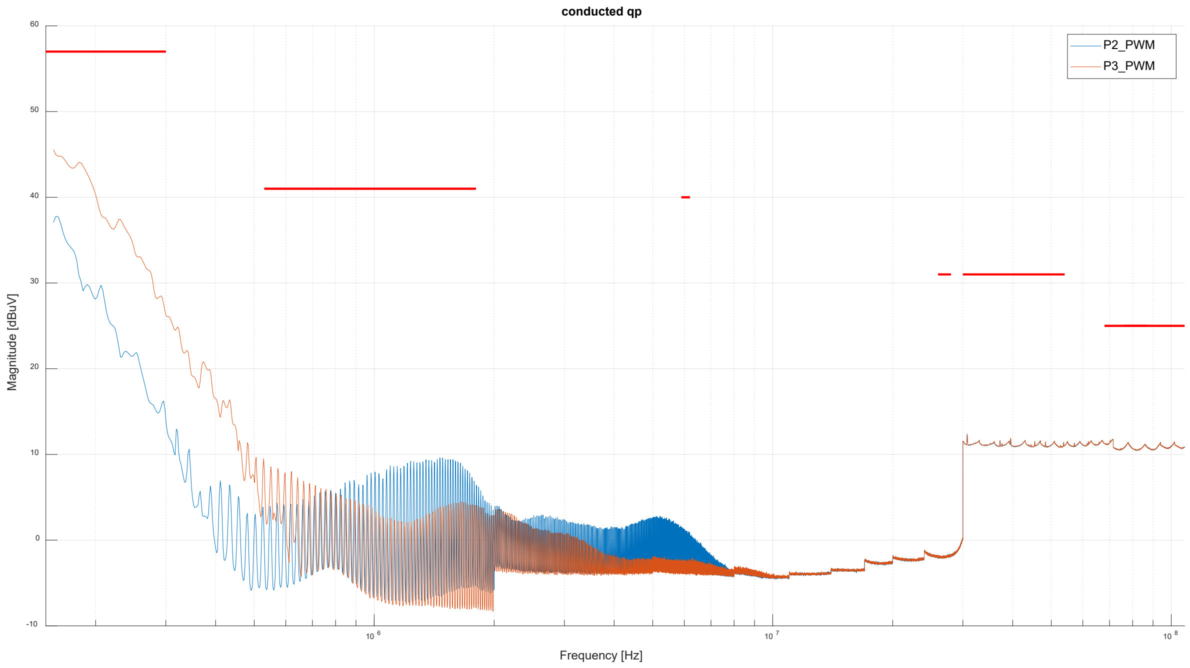


Figure 56: Conducted emissions of prototype 2 versus prototype 3

It is worth remembering that in prototype 3 only one winding is working due to an error in the manufacturing process.

The main original objective of comparing prototype 2 and prototype 3 was to compare the effect of a 2-layer PCB to a 4-layer PCB. Now it cannot be done but it can be seen the effect of only using one phase of the motor with a 2-layer PCB and see if it is different than a fully functional 4-layer prototype.

Regarding conducted emissions, both boards are below the limits and prototype 3 has higher components at lower frequencies but the levels of emissions at medium frequencies is a bit lower.

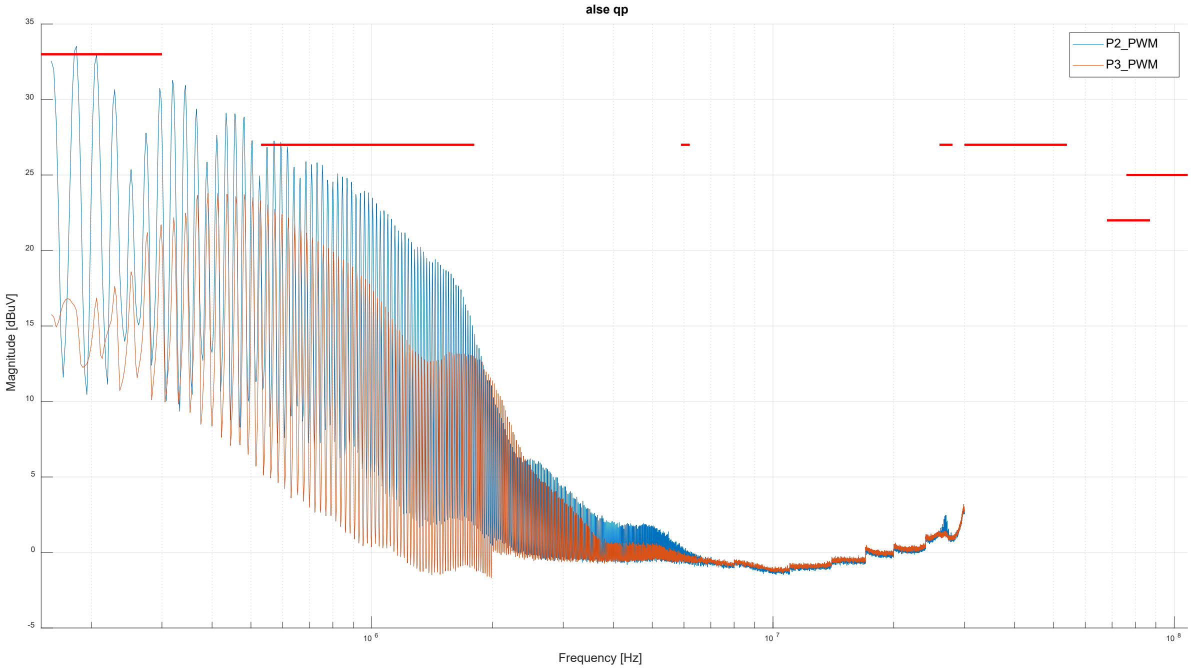


Figure 57: Radiated emissions of prototype 2 versus prototype 3

When comparing radiated emissions, it can be seen that the emissions levels are lower when driving the motor with a PWM switching strategy. As said before, it is not really relevant because the conditions of the experiment are not the same. But from this mistake it can be seen that in order to reduce even more the electromagnetic emissions, only one phase of the motor could be used when there is no need of using both of them.

7 CONCLUSIONS

The main objective of this project was to determine if the layout of a PCB for small drives was relevant when trying to comply with the EMC regulations. This has proved to be true with prototype 1 (See section 6.1.1) where the same components were rearranged and emission levels dropped. This means that even though the frequencies used in this type of PCB are not really high it is still important to keep in mind the EMC design rules and design the PCB properly.

In order to see if the production cost could be reduced, prototype 2 was designed. From this experiment it can be concluded that the production cost could be reduced but only if a different switching strategy which lowers the radiated emission levels is used. Further modifications could be performed in order to achieve lower rise/fall times in the MOSFETs in order to optimise the radiated behaviour. Also, as said before, the change of the inductor size makes the filter work differently than simulated previously. Maybe with the original inductor size the filter would perform better and filter out more frequency components.

The difference between a 2-layer PCB and a 4-layer PCB cannot be shown in this project but despite that it has been proved that using only one phase of the motor is feasible in certain conditions and that it reduces the electromagnetic emissions.

During the results section it has been seen that the electromagnetic emissions can be reduced by hardware or software. The scope of this project was to reduce them by hardware (only rearranging or changing the components). The reduction by software is part of another project [24].

8 REFERENCES

[1] Shieldingshop, “History of Electromagnetic Compatibility,” [Online]. Available: https://medium.com/@Shieldingshop/history-of-electromagnetic-compatibility-c1d3eab16a38. [Accessed March 2019].

[2] D. Etiemble, “45-year CPU evolution: one law and two equations,” University Paris Sud, 2018.

[3] “The singularity is near,” [Online]. Available: http://www.singularity.com/charts/page61.html. [Accessed March 2019].

[4] “Computer Chips,” [Online]. Available: http://www.chipsetc.com/computer-chips-inside-the-car.html. [Accessed March 2019].

[5] F. Auzanneau, “Wire troubleshooting and diagnosis,” CEA Saclay Nano-INNOV, 2013.

[6] S. Alexandersson, *Automotive Electromagnetic Compatibility*, Lund University, Sweden, 2008.

[7] P. Clayton, *Introduction to Electromagnetic Compatibility*, Wiley-Interscience, 2006.

[8] “EMI filter insertion loss,” [Online]. Available: https://medium.com/@emianalyst/emi-filter-insertion-loss-f4837d3ea051. [Accessed March 2019].

[9] C. Basso, “Introduction to Fast Analytical Techniques,” ON Semiconductors, 2016.

[10] D. Knight, “Introduction to Linear Voltage Regulators,” Digikey, 2016. [Online]. Available: https://www.digikey.com/en/maker/blogs/introduction-to-linear-voltage-regulators. [Accessed May 2019].

[11] B. Klein, “Exploring and understanding linear voltage regulators,” EETimes, [Online]. Available: https://www.eetimes.com/document.asp?doc_id=1272466.

[12] “Radiated Emission,” [Online]. Available: https://en.wikiversity.org/wiki/Radiated_emission. [Accessed June 2019].

[13] “Electromagnetic Radiation and How It Affects Your Instruments,” [Online]. Available: https://www.osha.gov/SLTC/radiofrequencyradiation/electromagnetic_fieldmemo/electromagnetic.html. [Accessed June 2019].

[14] S. Farahani, “Dipole antenna,” [Online]. Available: https://www.sciencedirect.com/topics/engineering/dipole-antenna. [Accessed March 2019].

[15] P. Albersman, “Reduce EMI in Switching Power Supplies,” [Online]. Available: https://aerospacepal.com/emi-current-loop-area/. [Accessed May 2019].

[16] Texas Instruments, “AN-988 High Speed Clock Sources and Their Effect on Electromagnetic Compatibility”.

[17] ROHM Semiconductor, “Calculation of Power Loss,” no. AEK59-D1-0065-2, 2016.

[18] D. Adamczyk and J. Teune, “InCompliance,” [Online]. Available: https://incompliancemag.com/article/crosstalk-reduction-between-pcb-traces/. [Accessed June 2019].

[19] S. Sattel, “What are decoupling capacitors?,” [Online]. Available: https://www.autodesk.com/products/eagle/blog/what-are-decoupling-capacitors/. [Accessed June 2019].

[20] “Series Resonance Circuit,” [Online]. Available: https://www.electronics-tutorials.ws/accircuits/series-resonance.html. [Accessed June 2019].

[21] L. Green, “RF-inductor modeling for the 21st century,” *EDN Magazine,* pp. 67 - 70, 2001.

[22] Würth Electronik, “Inductor datasheet”.

[23] H. Gruebler, S. Leitner, A. Muetze and G. Schoerner, “Improved Switching Strategy for a Single-Phase Brushless Direct Current Fan Drive and its Impact on Efficiency,” *IEEE Transactions on Industry Applications,* vol. 54, no. 6, pp. 6050-6059, november/december 2018.

[24] F. Krall, H. Gruebler and A. Muetze, “Angle modulated switching strategy for fractional horsepower BLDC,” in *EPE 2019*, Graz, Austria, 2019.

[25] Tektronix, “Troubleshooting EMI: To Peak or NOT to Peak,” no. 48W-28871-0, 2013.

[26] D. Gurney, “EMI emissions testing: peak, quasi-peak, and average measurements,” EDN, [Online]. Available: https://www.edn.com/design/test-and-measurement/4460716/EMI-emissions-testing--peak--quasi-peak--and-average-measurements. [Accessed June 2019].

[27] “OVE EN 55025:2017 / CISPR25:2016,” 2018.

[28] M. Rangu, Getting EMC design right first time, EMC Fastpass, 2014.

[29] M. I. Montrose, “Analysis on Loop Area Trace Radiated Emissions from Decoupling Capacitor Placement on Printed Circuit Boards,” 1999.

[30] Infineon, “EMC and System-ESD Design Guidelines for,” no. AP24026, 2016.

[31] A. Schweitzer, “PCB Layer Stack-up,” in *Fine Line*, Hilden, Germany, 2015.

[32] D. Winkler and D. Deutschmann, “EMV-gerechtes Filterdesign,” TU-Graz, Infineon, 2009.

[33] “Full wave rectifier,” [Online]. Available: https://www.electronics-tutorials.ws/diode/diode_6.html. [Accessed April 2019].