AN4694

EMC design guides for motor control applications

Alessio Corsaro, Carmelo Parisi and Craig Rotay

Introduction

In recent years, continuous demand for efficient, compact and low cost applications in the motor control industry has led to a boom in inverter-based solutions driven by MCUs. These applications involve high switching frequencies and high power levels and must function compatibly with severe electromagnetic environments (EMC). The implementation of transient immunity protections (EMS) to prevent appliance malfunction and the design of countermeasures to limit application emissions (EMI) is therefore a becoming a growing concern for appliance designers.

Best practices regarding EMC control through PCB layout, circuit design and component selection can greatly improve EMC performance, especially when they are an integral part of the entire design cycle.

This application note discusses the effects of EMC on motor control applications and suggests some practical hardware guidelines to provide cost-effective protection against electrical fast transients (EFT), electrostatic discharge (ESD) and to limit the conducted and radiated emissions (EMI) in appliance applications.

Contents

List of figures

1 EMC definitions

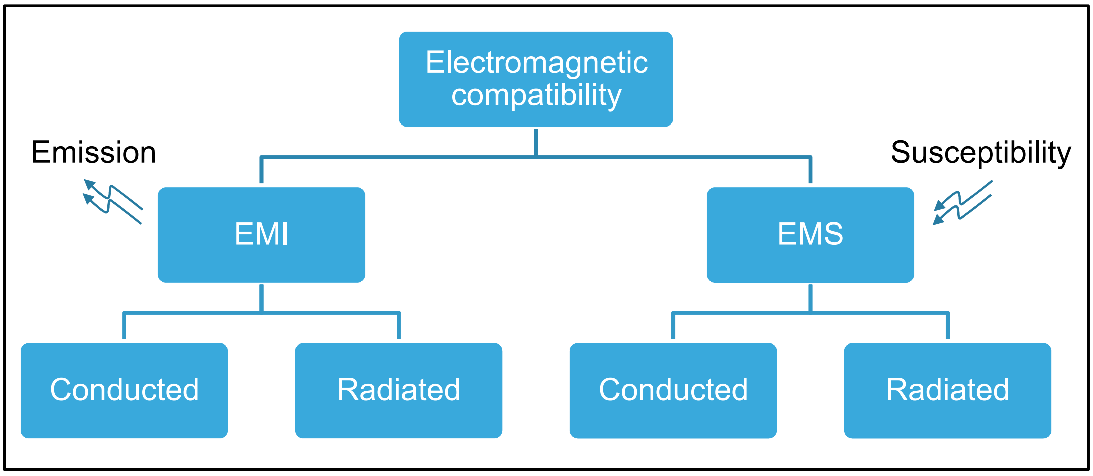
Electromagnetic Compatibility (EMC) is the ability of electrical and electronic systems, equipment and devices to operate in their intended electromagnetic environment within a defined safety margin, without suffering or causing unacceptable degradation as a result of electromagnetic interference (ANSI C64.14-1992). EMC is classified into electromagnetic interference (EMI) and electromagnetic susceptibility (EMS), as shown in *Figure 1: "Electromagnetic compatibility diagram"*

Electromagnetic Interference (EMI) refers to disruptive electromagnetic energy transmitted from one electronic device or equipment to another, it can be:

- conducted emission when it is propagated along a power line
- radiated emission when it transmitted through free space

Electromagnetic Susceptibility (EMS) represents performance immunity against disturbances like electrostatic discharge (ESD), electrical fast transient or burst (EFT), lightning surges and electromagnetic waves.

Figure 1: Electromagnetic compatibility diagram



1.1 EMC environments

OEM appliances are governed by different standards for both EMI and EMS, based on their intended application. These standards contain test methods to satisfy product specifications and regulatory requirements, and define transient sources, entry paths into a system and severity levels.

The principal international standards for immunity testing are:

- IEC61000-4-2: the ESD waveform simulates the discharge from a human operator and is injected at any location that the operator is likely to touch, including all user accessible controls and external connectors. The test levels for ESD vary widely depending on the application.
- IEC 61000-4-4: the EFT waveform simulates the transients created by the switching of relays or the interruption of inductive loads on power mains. It is applied as a specific burst waveform, usually introduced along the application’s AC power cord. The EFT waveform can also be injected in signal and control lines to only simulate the conducted coupling of the EFT in these lines.

EN55014-1 and EN55014-2 are, respectively, the principal European emissions and immunity standards for household appliances and power tools.

This standard establishes uniform requirements for radio disturbance levels applicable to the conduction and radiation of radio-frequency disturbances from appliances mainly governed by motors and switching. AC motors generate harmonic signals at the power input and output side, creating electromagnetic interference with surrounding electrical devices and mains power networks. AC drives can both cause and be affected by such disturbances.

2 EMC phenomena and testing

The following section introduces the most common EMC phenomena in electrical system designs and the tests used to emulate them.

2.1 ESD immunity test

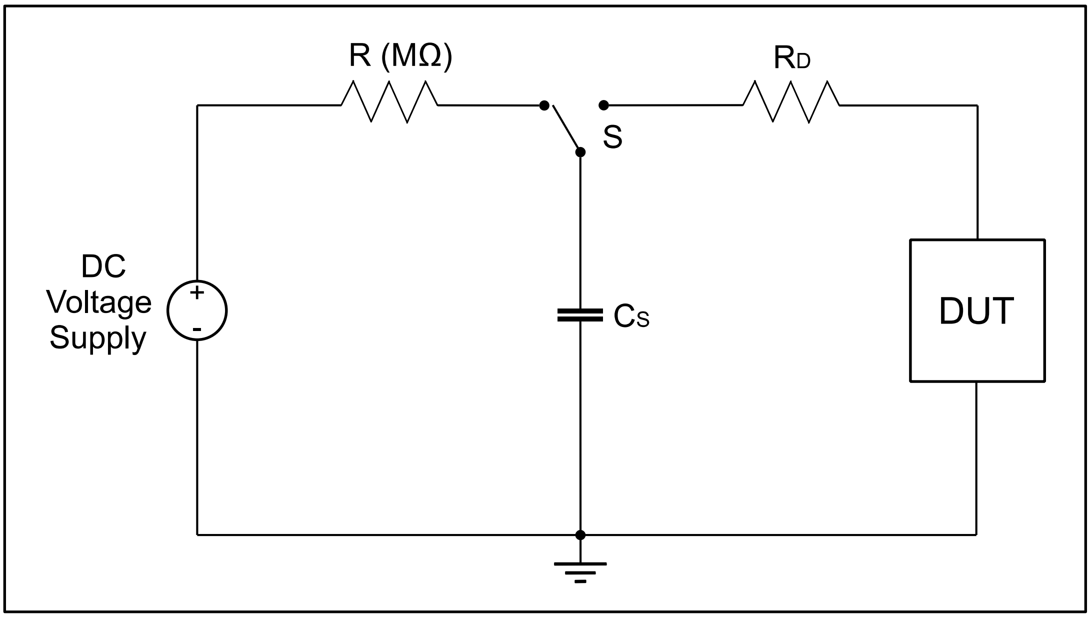
Electrostatic discharge (ESD) is the exchange of electrons resulting from the field accumulation between two objects with different charges and the damage to a device depends on its ESD sensitivity and its ability to dissipate the discharge energy

The test procedures are based on the principal ESD-event models below.

2.1.1 Human body model (HBM) testing

The HBM testing model (ref. ANSI/ESDA-JEDEC JS-001-2010: Electrostatic Discharge Sensitivity Testing - Human Body Model) represents the discharge that the fingertip of a standing individual delivers to the device. It is modeled by the circuit shown in *Figure 2: "Human body model (HBM) ESDS device sensitivity test circuit"*, featuring a 100 pF storage capacitor (C_S) discharging through a switching component and a 1.5 kΩ series resistor, R_D.

Figure 2: Human body model (HBM) ESDS device sensitivity test circuit

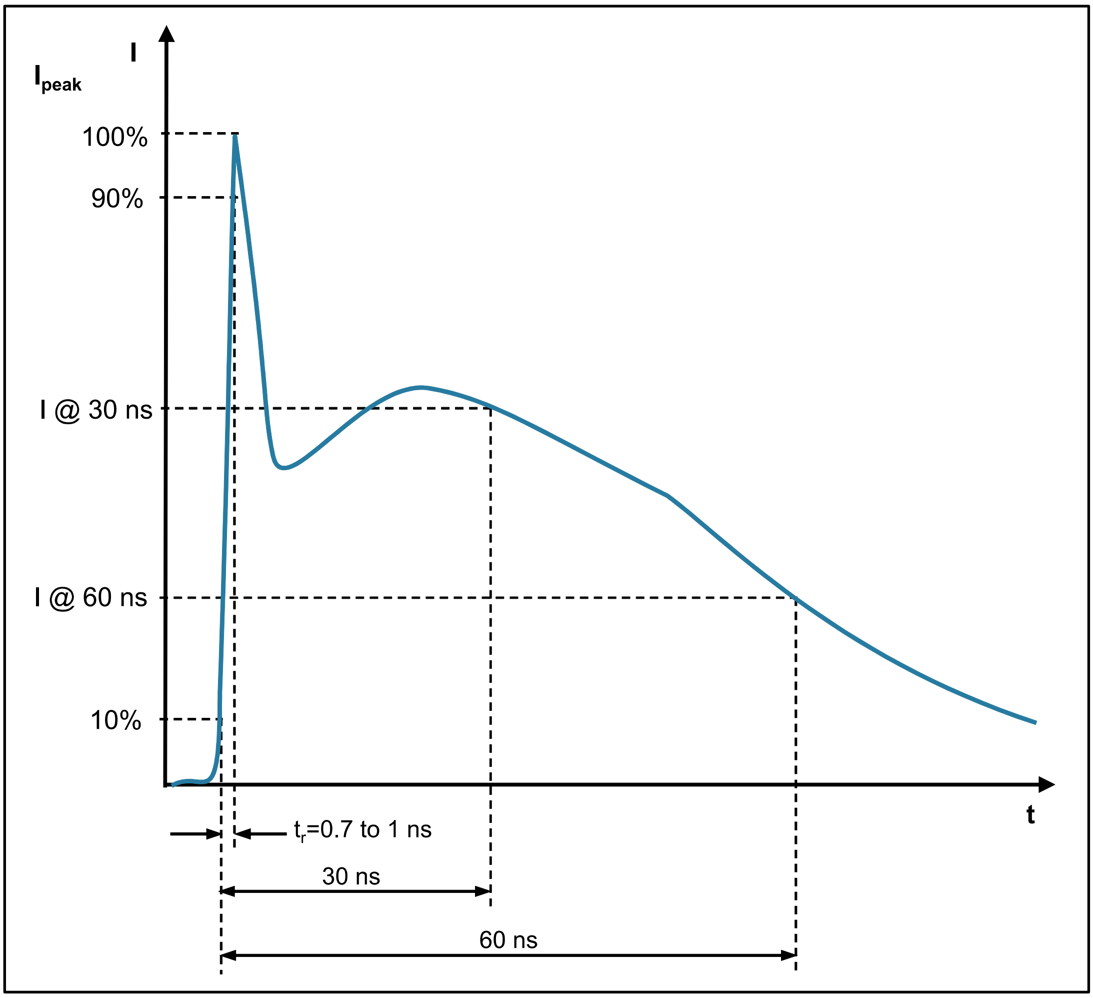


C_S and R_D represent the capacitance and the discharge resistance of the human body respectively. R is the series resistance of the DC high-voltage power supply. When switch S is connected to RC_S, the capacitor is charged.

The human body can store electrostatic energy to potentials in the order of several thousand Volts (8 kV to 10 kV is common) and peak currents in the order of a hundred Amperes.

The IEC 61000-4-2 standard specifies an ESD waveform with characteristics similar to a typical human body discharge pulse, but with far greater energy (*Figure 3: "ESD waveform"*)

Figure 3: ESD waveform

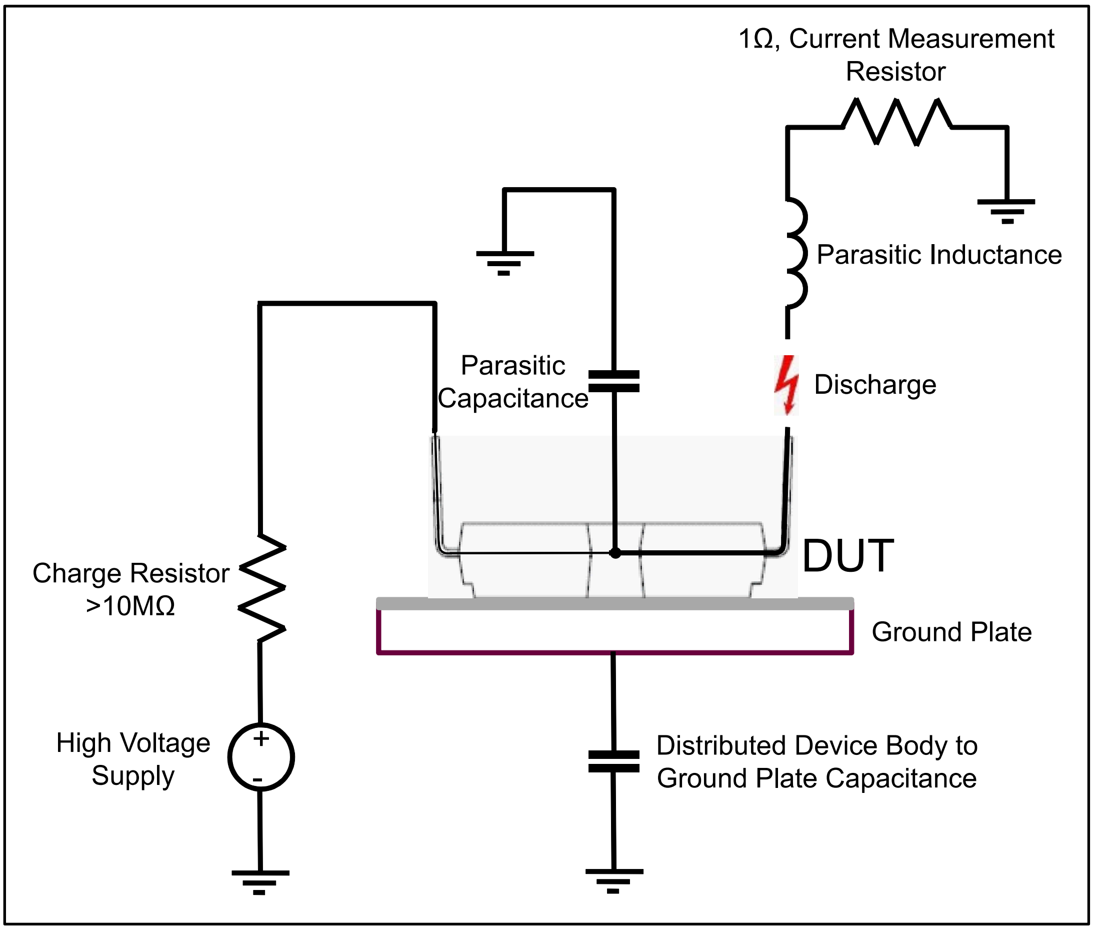


The HBM sensitivity test is usually an automatic system which delivers ESD pulses and signals device failure when datasheet parameters are not satisfied.

2.1.2 Charged device model (CDM) testing

Another ESD event is the transfer of energy from a charged ESDS (electrostatic discharge sensitive) device, perhaps due to the contact with a conductive surface. This event is known as the CDM model event (ref. ANSI/ESD STM5.3.1: Electrostatic Discharge Sensitivity Testing - Charged Device Model) and the associated high current peak can be even more destructive than the HBM. A typical CDM model test circuit is shown in *Figure 4: "Typical charge device model test"*.

Figure 4: Typical charge device model test

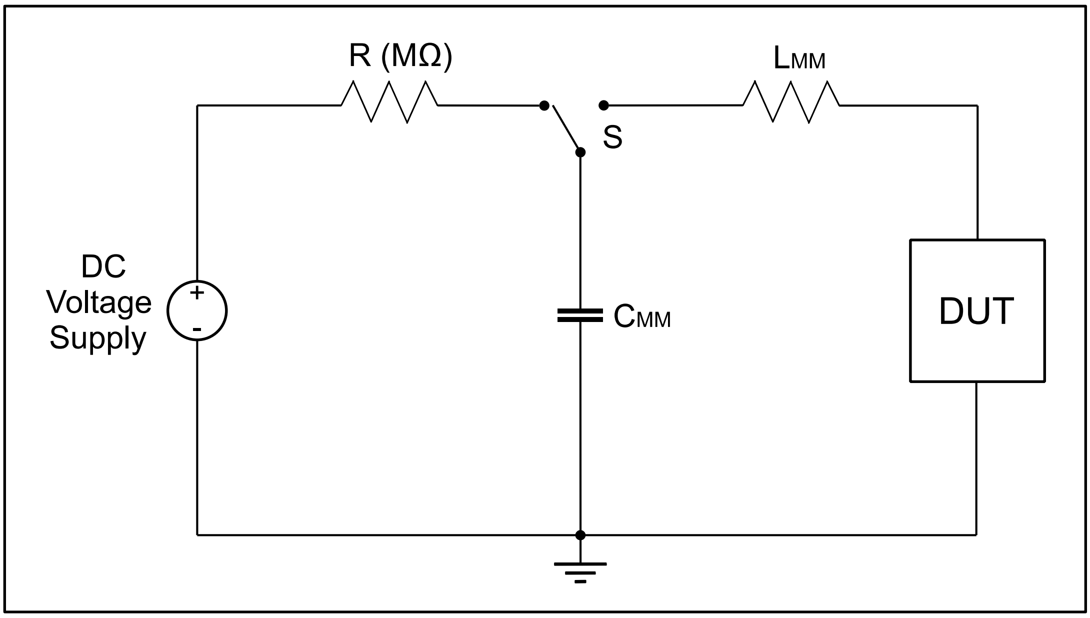


2.1.3 Machine model (MM) testing

Discharge events from conductive surfaces such as automatic equipment and ESDS devices are covered by the MM model (ref. ESD STM5.2: Electrostatic Discharge Sensitivity Testing - Machine Model).

The machine model consists of a 200 pF capacitor discharging into the DUT with no series resistor (*Figure 5: "Machine model (MM) ESDS device sensitivity test circuit"*). The series inductance L_{MM} creates the oscillating machine model waveform and is defined by the peak current, rise time and period of the waveform.

Figure 5: Machine model (MM) ESDS device sensitivity test circuit



2.1.4 ESD severity levels

The ESD threat in IEC 61000-4-2 is divided into four severity levels, depending on the operating environment of the device, with the corresponding test pulse peak voltage (*Table 1: "IEC 61000-4-2 severity levels"*):

- levels 1 and 2 are reserved for controlled environments with anti-static materials
- level 3 is for lightly handled equipment
- level 4 is for continuously handled equipment

Table 1: IEC 61000-4-2 severity levels

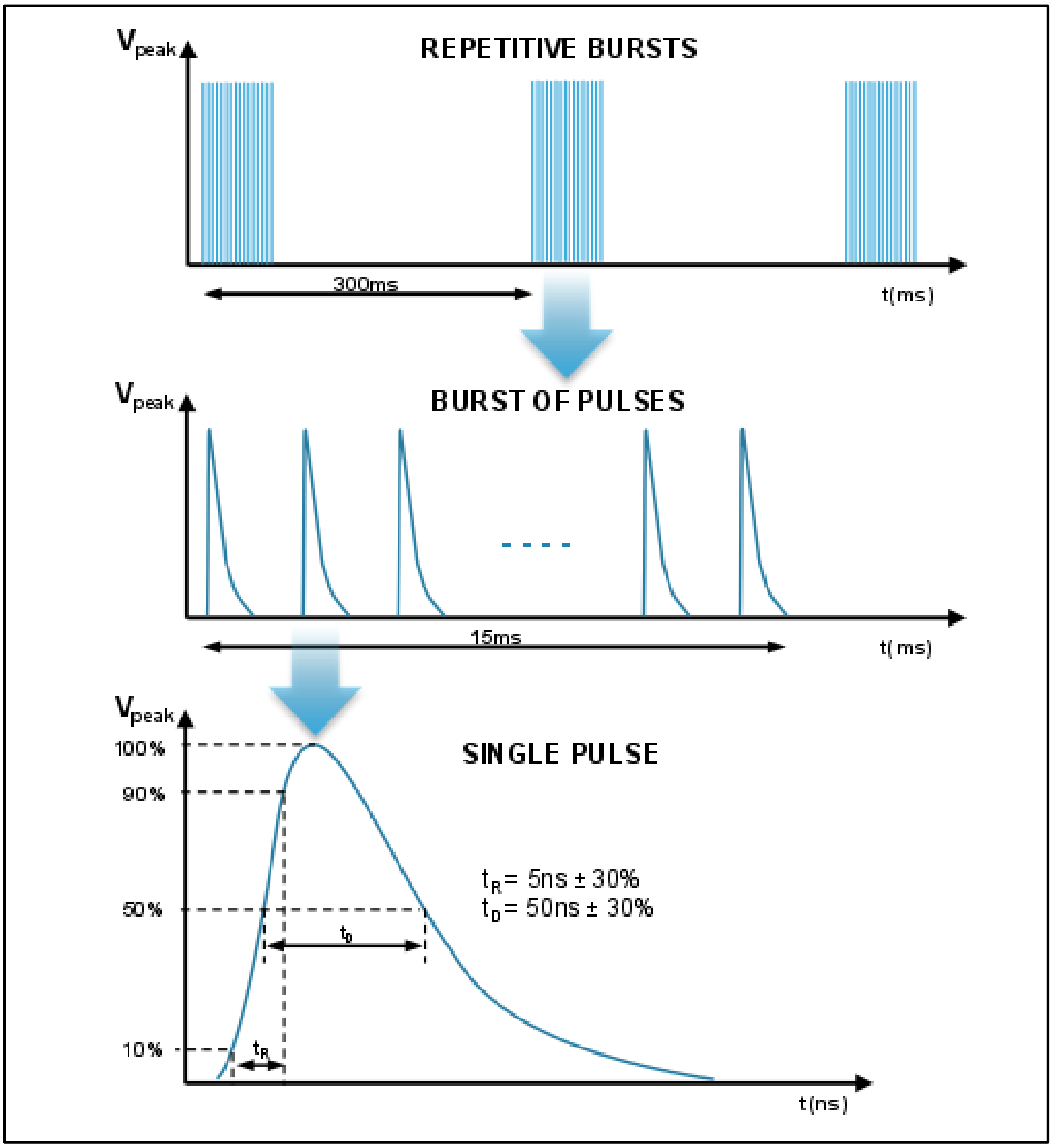
Severity level	Test voltage, kV contact discharge	Test voltage, kV air discharge
1	2	2
2	4	4
3	6	8
4	8	15

2.2 EFT immunity test

Electrical fast transient disturbances or bursts are common in all applications, including electrical switches and inductive loads. They are generally power line phenomena, but can also cause problems on signal lines due to inductive or capacitive coupling. They can occur during the commutation of inductive loads, when the current is disconnected and a series of small sparks delivers high-voltage spikes to power lines.

The IEC 61000-4-4 standard specifies the EFT-sensitivity test for electrical components . The disturbance is described in terms of a series of 2 to 5 kHz high voltage spikes, with 15 ms burst lengths at 300 ms intervals. The test circuit has a 50 Ω load driven by a voltage generator with a dynamic source impedance of 50 Ω. Each individual burst pulse is a 50 ns double exponential waveform with a 5 ns rise time (*Figure 6: "EFT waveform"*).

Figure 6: EFT waveform



Based on the susceptibility of the application, the following four severity levels are defined as a function of the installation environment:

1. well protected
2. protected
3. typical industrial
4. severe industrial

IEC 61000-4-4 stipulates the open-circuit test voltages for each threat level and the burst series frequency, as a function of the test level (*Table 2: "Table 2: IEC 61000-4-4 severity levels"*).

Table 2: Table 2: IEC 61000-4-4 severity levels

Severity level	EFT peak amplitude (kV)	Repetition frequency (kHz)
1	0.5	5
2	1	5
3	2	5
4	4	2.5

2.3 Immunity test (ESD, EFT) behavior classes

ESD and EFT results are classified in terms of the loss of function or degradation of the tested equipment:

- Class A: normal performance within limits specified by the manufacturer, requestor or purchaser
- Class B: temporary loss of function or degradation of performance which disappears when the disturbance ceases, and from which the equipment under test recovers its normal performance without operator intervention
- Class C: temporary loss of function or degradation of performance, the correction of which requires operator intervention
- Class D: loss of function or degradation of performance which is not recoverable, owing to damage to hardware or software, or loss of data

2.4 Emissions

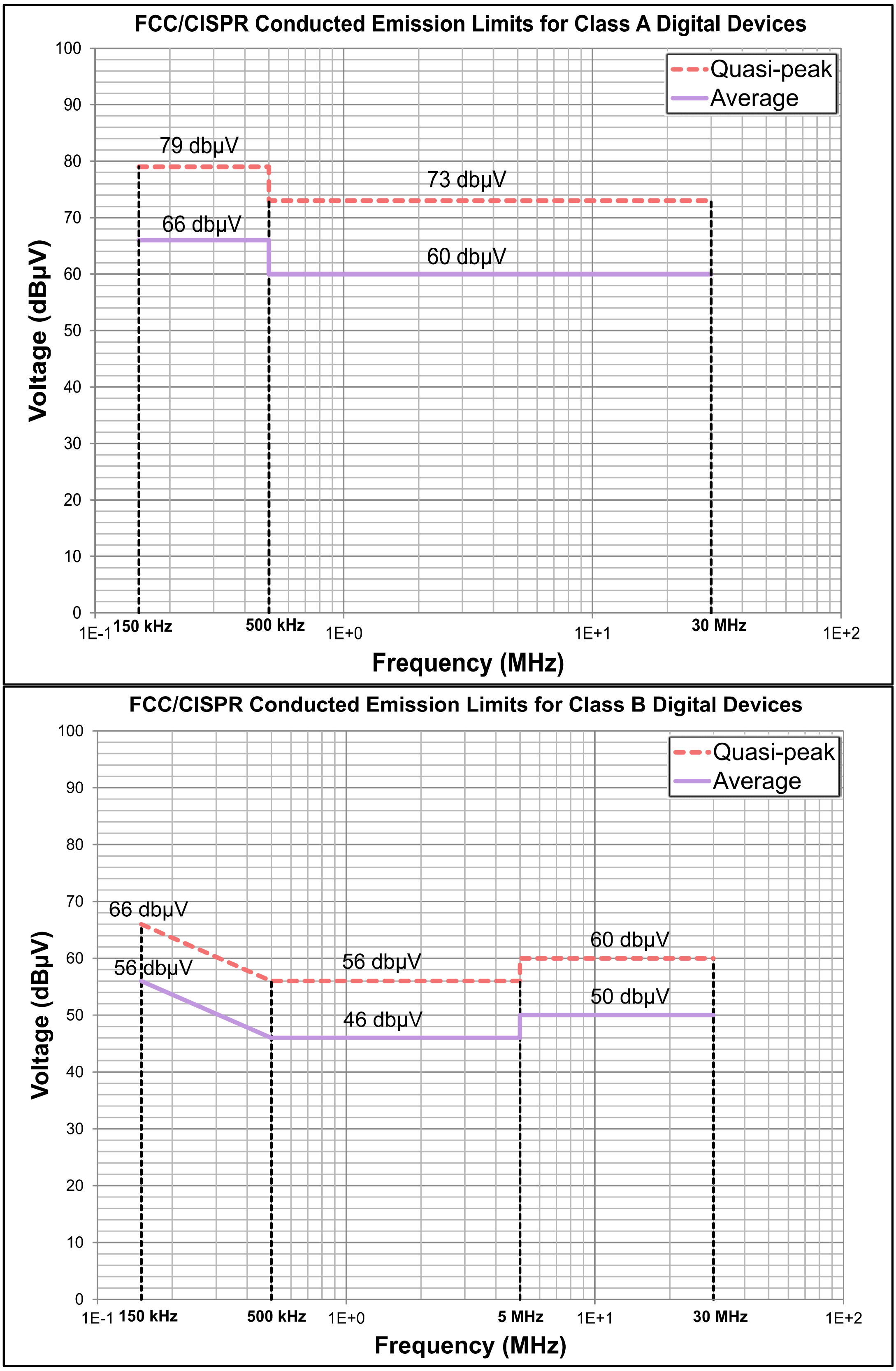
2.4.1 Conducted emissions: standards and testing

Any electronic device is a potential source of noise currents both on the power network of the installation and the overall power grid. This disturbance can affect other devices connected to the power grid through conductive coupling and the electrical length of the conductors may effectively allow this noise to radiate.

The CEI EN 55022 standard specifies the limits for both class A (products marketed for commercial or industrial use) and class B (product marketed for residential or domestic use) devices in the 150 kHz to 30 MHz frequency range. The conducted quasi-peak and average value emission limits are provided (*Figure 7: "Conducted emission limits"*).

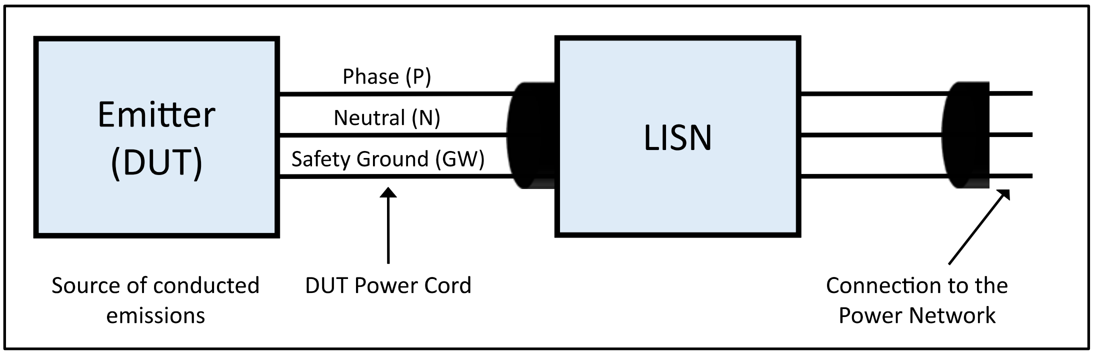
Although the conducted emissions are expressed as noise currents, they are measured in terms of proportional noise voltages and the standard limits are therefore expressed in dBμV.

Figure 7: Conducted emission limits



Conducted emissions are measured using a line impedance stabilization network (LISN), in series with a power cord (*Figure 8: "LISN layout"*). The LISN provides constant impedance for the DUT over the given frequency range, rendering the measurement independent of the position of the power network connection point, and filters power network noise currents which could affect the test.

Figure 8: LISN layout

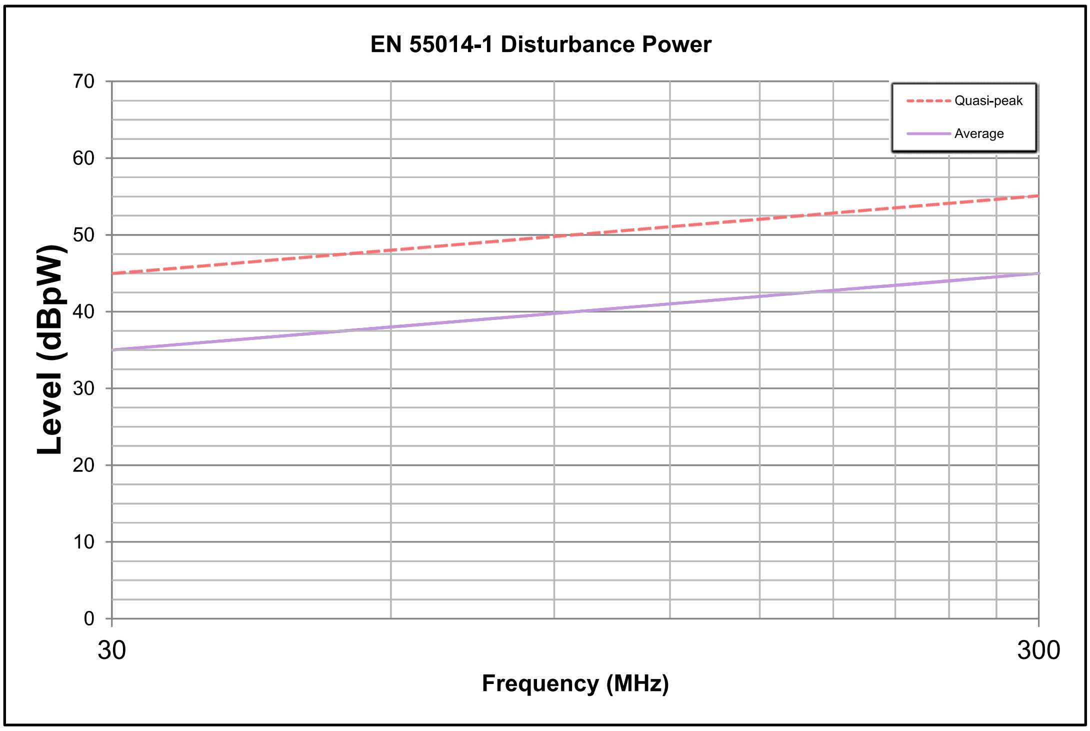


2.4.2 Radiated emissions: standards and testing

Any electronic device can generate and emit electromagnetic fields. The release of electromagnetic energy in the form of radiated emissions may interfere with the normal operation of the device itself or nearby devices.

The radiated emissions are measured in terms of disturbance power, defined as the power that the appliance could supply to its leads. The standard CEI EN 55014-1 standard specifies the limits (in dBpW) and the measurement methods (art.6) of the disturbance power in the 30 MHz to 300 MHz frequency range for both quasi-peak and average values (*Figure 9: "Disturbance power limits for household"*).

Figure 9: Disturbance power limits for household



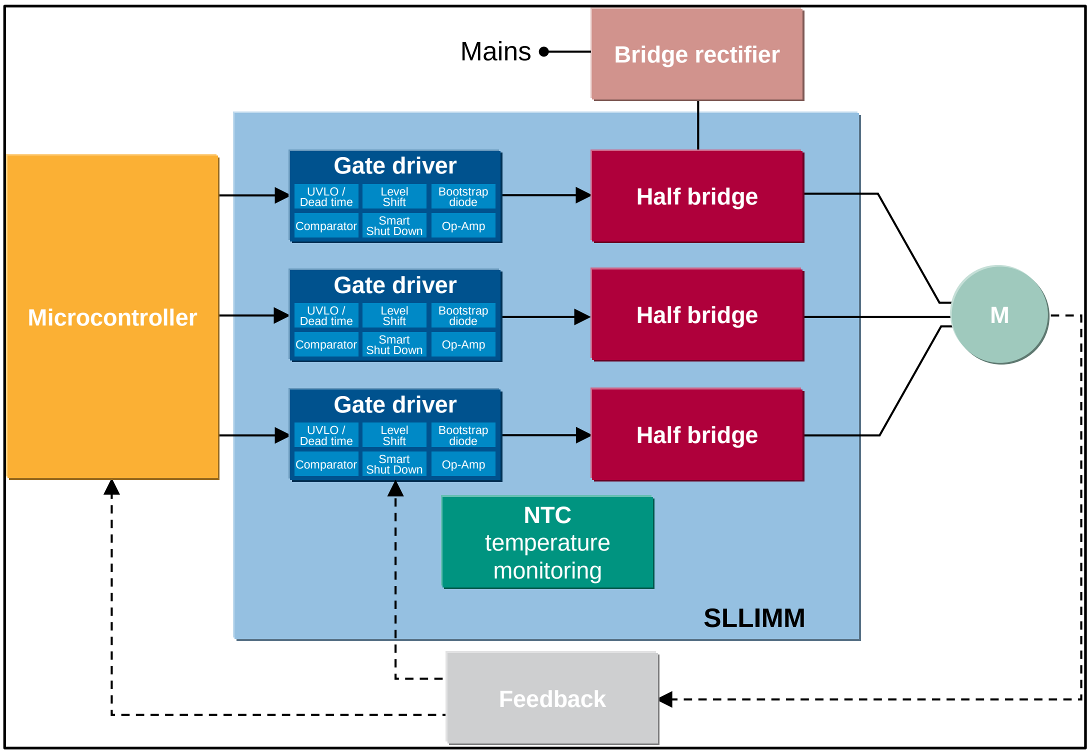
For frequencies above 30 MHz, the disturbance energy is mostly radiated by the mains leads, so the disturbance power can be measured via the power supplied by the appliance to a suitable absorbing device placed around these leads at the position of maximum absorption. Consequently, the standard specifies the measurement by an absorbing clamp placed around a power cord of length based on the wavelength of the lowest frequency.

3 Impact on motor control operation

EMC compliance must be a primary consideration when designing new applications in order to reduce cycle times and project costs and avoid wasting resources to retrospectively solve EMC issues. Furthermore, while good PCB layouts will involve similar production costs to bad ones, the costs associated with remedial activities can be high. Precautions should therefore be taken during the hardware system design implementation phase to control the impact of EFT, ESD and emissions.

An inverter-based motor control application (*Figure 10: "Block diagram of motor control inverter "*) generally consists of a digital part (microcontroller), a control part (IC gate driver, comparator for protection, op-amps for current sensing and other current and temperature sensors), a power stage (based on IGBT or MOSFET devices), a low voltage power supply and some voltage regulators.

Figure 10: Block diagram of motor control inverter



As the application manages high currents and voltages, the power stage configuration is critical and the board layout must include several aspects, such as track lengths and widths, circuit areas, as well as the proper routing of the traces and the optimized reciprocal arrangement of the various system elements and power sources in the PCB area.

Designers must first aim to reduce the EMI issues and over-voltage spikes due to parasitic inductances along the PCB traces.

The designer must also ensure that the EFT noise injected through the supply lines of the system is properly conducted, via external ground or via supply voltage, away from sensitive devices like microcontrollers or IC gate drivers, since it can cause bit errors in digital circuits and cause poor signal integrity in analog circuits.

Failure to do so can result in abnormal input PWM signals, undesired fault signals, insufficient protection, false current readings and overvoltage signals. All of these issues may lead to temporary loss of normal operation and even permanent device damage.

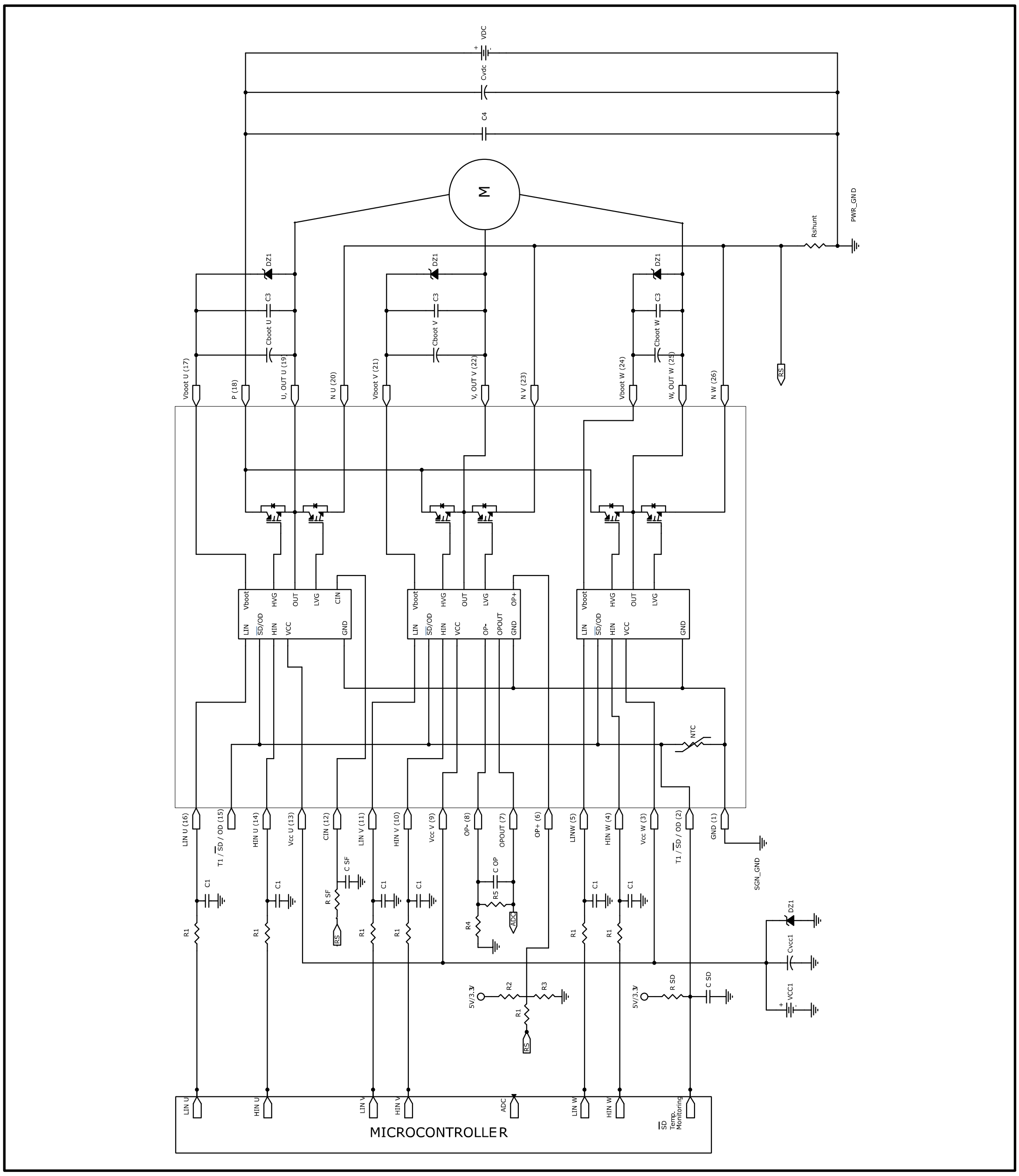
Finally, designers must prevent ESD-provoking conditions which can permanently damage components by implementing hardware solutions like low-pass filters, protection and clamp diodes as well as optimized PCB layouts.

4 PCB design and layout guidelines

In designing motor control circuits to meet EMC standards, the EMC requirements must be part of the product definition followed by target reductions for EMI emissions and susceptibility improvements during the circuit design, component selection and PCB layout phases.

A generic circuit topology of a highly integrated motor control design is illustrated in *Figure 11: "Typical motor control schematic"*. Here, we can visualize its various functional blocks and consider which functions might generate or be susceptible to EMI as well as the coupling paths that might exist between these sections.

Figure 11: Typical motor control schematic

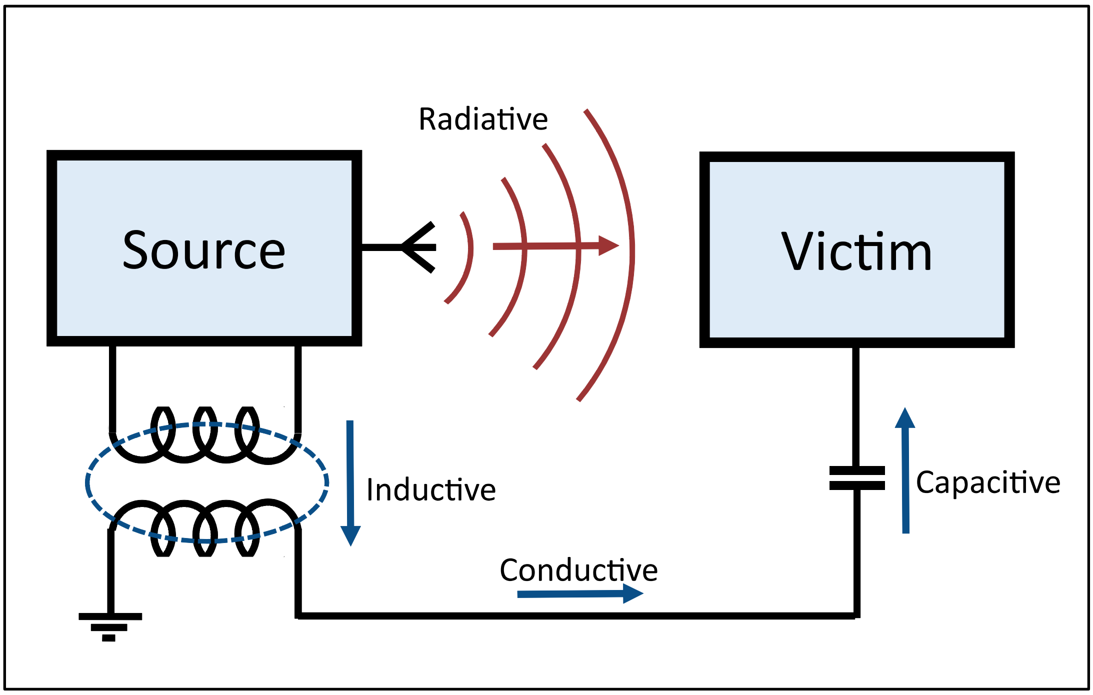


4.1 EMC Overview

A simple EMI model consists of the following elements (*Figure 12: "EMI model"*):

- EMI source
- Coupling path of EMI
- Victim or receptor of EMI

Figure 12: EMI model



EMI sources include microcontrollers, electronic discharges, transmitters, transient power components such as electromechanical relays, switching power supplies and lighting. In a microcontroller-based system, the clock circuitry is usually generates the most wide-band noise.

Although all electronic circuits are receptive to EMI transmissions, the most critical signals are the reset, interrupt, fault, protection and control lines. Analog amplifiers, control circuits and power regulators also are susceptible noise interference.

The coupling path between the source and the receptor can be:

- conductive - where the coupling path between the source and the receptor is formed by direct contact like a wire, cable or track connection.
- capacitive - where a varying electrical field exists between two adjacent conductors or tracks typically less than a wavelength apart, inducing a change in voltage across the gap.
- inductive or magnetic - where a varying magnetic field exists between two parallel conductors or tracks typically less than a wavelength apart, inducing a change in voltage along the receiving conductor
- radiative - where the source and receptor are separated by a large distance, typically more than a wavelength. The source and receptor act as radio antennas with the source emitting or radiating electromagnetic waves which propagate through open air.

The switch-mode power supply is usually a major source of EMI in motor control applications. It manages transient high current and voltages in the form of square pulses with high rates of di/dt and dv/dt. The waveforms are highly nonlinear and thus have high harmonics content. With so many frequency components present, the signals contain what is often referred to as noise, which can easily be conducted or radiated into other motor control circuits, causing them to malfunction.

Designers often use snubbers and soft switching techniques to minimize the EMI from the SMPS.

Surprisingly, since today's power transistors are often capable of switching at frequencies far above what is required by the application, certain sections of the circuit can unwittingly amplify the noise and harmonic content and further compound the EMI problem. These unwanted frequency components can be high enough to be classified as Radio Frequency Interference, or RFI.

Inverter and driver circuits are also potential generators of EMI, and designers must focus on the turn-on and turn-off characteristics of the power transistor components to minimize EMI in these circuits. When the design is based on discrete IGBT or MOSFET components, the designers have more flexibility in tuning the turn-on and turn-off behavior using appropriate gate resistors to set the best trade-off between EMI and power loss.

When an IPM (Intelligent Power Module) is used, the driving network is internally set and already optimized for both EMI and power loss behavior.

Motor control designs also have sections that provide control or sense functions. These circuits are often susceptible to EMI, so design strategies such as bypassing, filtering and buffering are essential to avoid their malfunction.

Once the EMI sources and susceptible components have been identified, the best circuit topology can be chosen within the performance and cost constraints.

Finally, with the initial circuit design frozen and its schematic captured, attention can be focused on what often lies at the heart of the EMC source and control problem: the physical realization of the PCB. This phase of the design may be considered a "segmentation strategy", because it takes into account how the layout and routing of various three dimensional structures and components may impact EMI on the final product. Many EMC obstacles are often found and dealt with during the segmentation and layout phases of the design.

The main phases in which EMC requirements should be addressed are:

1. circuit definition phase: identification of the appropriate EMC standard on which the design is based
2. circuit design phase: during the schematic capture the engineers should:
   a. identify circuits and components that are potential EMI Sources
   b. identify circuits and components that are sensitive to EMI (susceptors)
   c. identify potential conductive and radiated paths between EMI sources and susceptors
3. develop a circuit segmentation strategy for efficient layout and routing planning

4.2 Segmentation strategy

The critical layout structure/topology aspects which have a significant impact on EMI are:

1. PCB: determine the type of PCB, its size and the number of layers (often cost driven)
2. grounding: determine the grounding topology which is directly related to the PCB selection
3. signals: decide what types control, power and ground signaling will be present for the desired motor control functionality
4. coupling paths (crosstalk): determine the preferred method for exchanging signals between functional blocks (routing) and whether the majority of lumped components will be SMT or through-hole

5. component orientation and placement: identify large components or ones that require heat sinks as these may have placement restrictions and require special treatment
6. shielding: if other methods for controlling EMI do not satisfy your EMC goals or limits, consider how shielding may be applied to the PCB

4.3 Segmentation

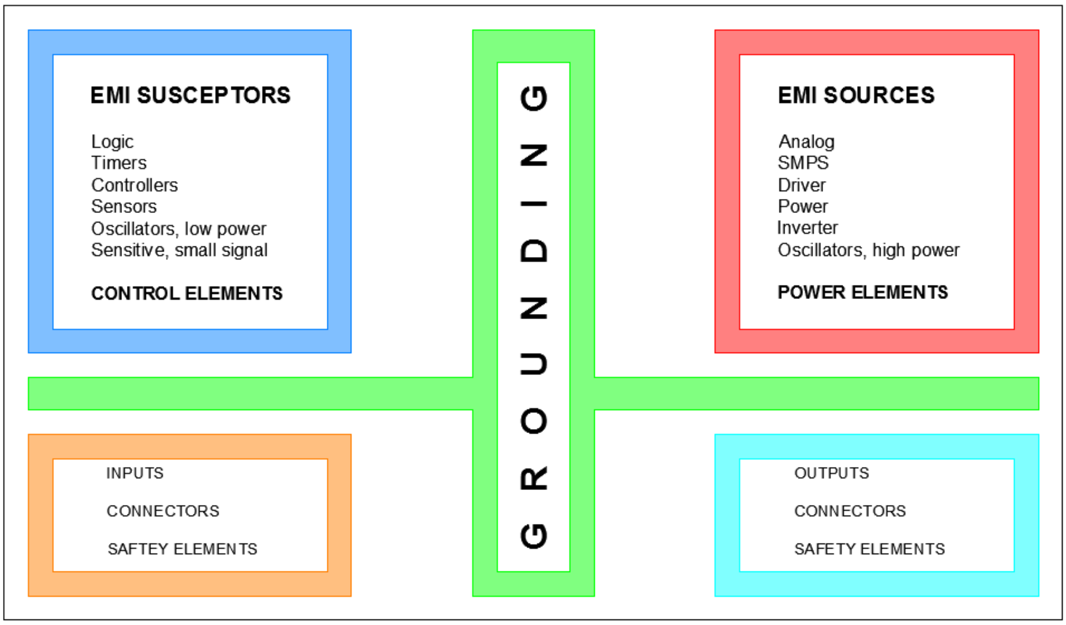
After thorough planning, the actual segmentation process should be logical and straightforward. The block diagram in *Figure 13: "Segmentation model"* shows the result of a segmentation plan which that takes into account all the major contributors to EMI. It shows an overview of:

- how circuit functions should be compartmentalized into blocks
- how the blocks will be arranged
- how the blocks are to be separated by a grounding scheme

In addition, it is a graphical tool for the efficient planning and routing of the PCB.

Within each functional block, each EMI contributor is identified directly from the schematic. Since the grounding scheme is so important to the success of EMC, it is shown clearly separating the blocks. Of course, this is merely an ideal arrangement for the blocks and the grounding scheme, serving as a constant reminder to keep the design as close to the ideal as possible.

Figure 13: Segmentation model



So far, we have described a top-down approach to EMC considerations in which the global contributors to EMI are identified, a segmentation strategy is applied and the foundation for an EMI friendly layout is established.

4.4 Physical layout and EMI: PCB selection and layout guidelines

In this section, we switch to a bottom-up approach for managing EMC objectives via an intelligent layout, involving the efficient placement of EMI contributors and corresponding interconnections and interactions.

4.4.1 The PCB

Since the design is based on the PCB, we must consider the numerous factors regarding PCB selection for an EMI-friendly product.

Conductors that are electrically long with respect to the physical wavelength of the associated signal (λ), have a greater potential to be affected by EMI sources or to become EMI sources, so designers should choose PCB materials with the lowest possible εr (dielectric constant of the substrate material).

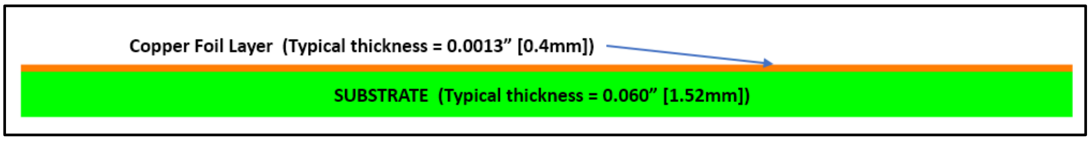
FR4 is commonly used for low frequency designs, with its εr of approximately four owing to the insulating layers in glass-filled epoxy resin.

The thickness of the substrate layer is significant because it influences the degree of coupling between different conductive layers and between adjacent conductors. The width of a conductor compared to the thickness (height) of the isolating layer (w/h) governs the amount of coupling between conductors, and is therefore important for effective EMI control.

The most significant factor in the final EMC performance of a particular design is probably the specification of the number of available layers for the PCB. This decision is so important because it relates to the intended grounding topology, which determines the overall EMI behavior. Access to ground and the shielding offered by the grounding structures is the key to EMI management.

Figure 14: "Single-layer PCB cross section" shows a low-cost single-layer PCB, where particular attention must be paid to the grounding scheme as power, control and ground circuitry are all confined to a single plane. This adds considerable complexity to the layout and increases the risk of EMC problems as there are more opportunities for circuitry to interact and access to grounding structures is limited.

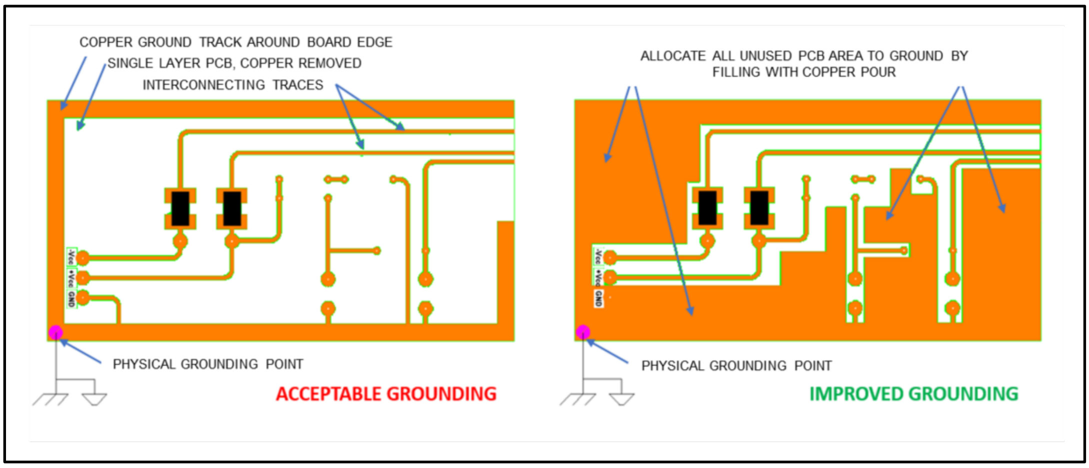
Figure 14: Single-layer PCB cross section



On this type of PCB, as much of the board perimeter should be dedicated to ground as possible, with frequent short and wide connections to this feature. All areas not requiring interconnecting traces should utilize a copper pour (*Figure 15: "Single-layer PCB with copper pour"*) and connect these areas to ground. The copper pour should, however, be removed from areas isolated from ground.

Single-layer PCBs do not therefore provide sufficient flexibility for a robust EMC solution.

Figure 15: Single-layer PCB with copper pour



A two-layer board (*Figure 16: "Two-layer PCB cross section"*) allows for a dedicated grounding layer, lower layout complexity and shielding at a slightly higher cost. However, the possibility that control and power blocks influence each other persists, so it remains critical to separate EMI sources and susceptors.

Figure 16: Two-layer PCB cross section

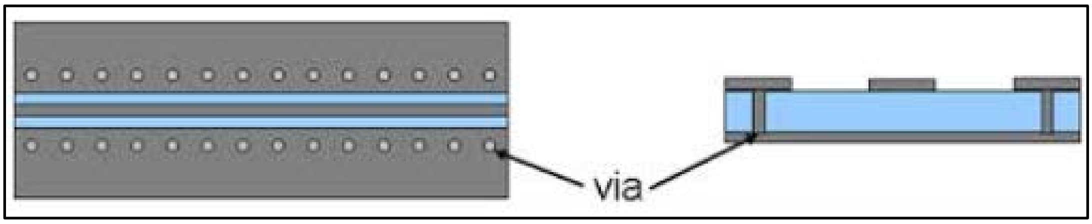
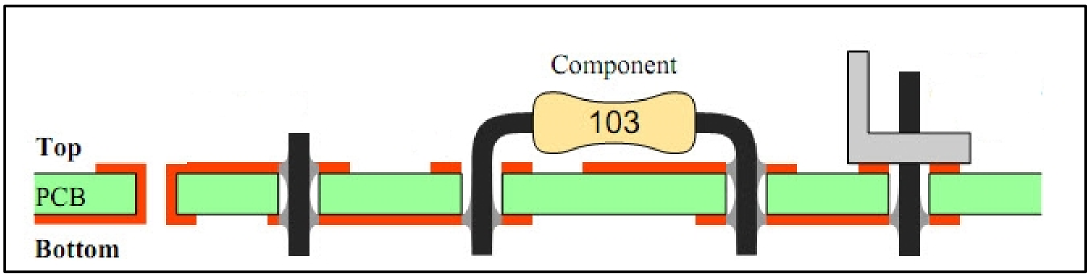


Figure 17: "Through-hole component mounting" shows how components mounted through the board can be placed on the side which is mostly ground, which can be an effective shield between the component side and the side with the traces.

Figure 17: Through-hole component mounting

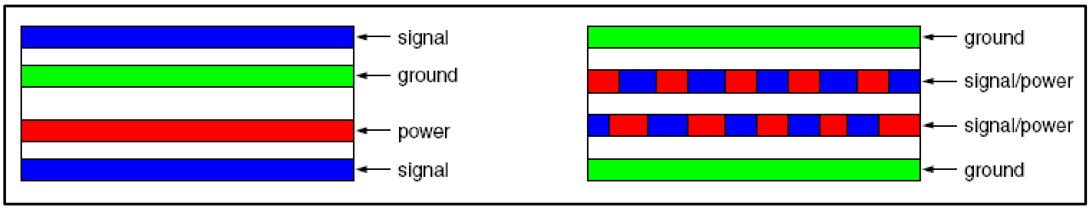


The four-layer PCBs in *Figure 18: "Four layer PCB cross section"* show two very different, more costly solutions. The board on the left has better self-shielding thanks to a dedicated ground layer sandwiched between a control signal layer on top and a power signal layer below. It is also possible to place the components on the top or bottom of the PCB and go through the board where needed. The power layer, however, cannot benefit from air cooling and may cause interference to the bottom signal layer; both signal layers can also be affected by external EMI sources.

The PCB on the right has external ground layers providing self-shielding from off-board or board-generated EMI. Unfortunately, it mixes signals on the same layers and has no provision for placing components.

In any case, the challenges associated with EMC control are greatly reduced with four conductive layers thanks to the increased availability and access to ground.

Figure 18: Four layer PCB cross section



A minimum of one dedicated grounding layer strongly benefits EMC control and any EMC-friendly design should therefore consist of at least two conductive layers.

4.4.2 Grounding

Many sources of and susceptors to EMI can be readily managed by a good grounding strategy.

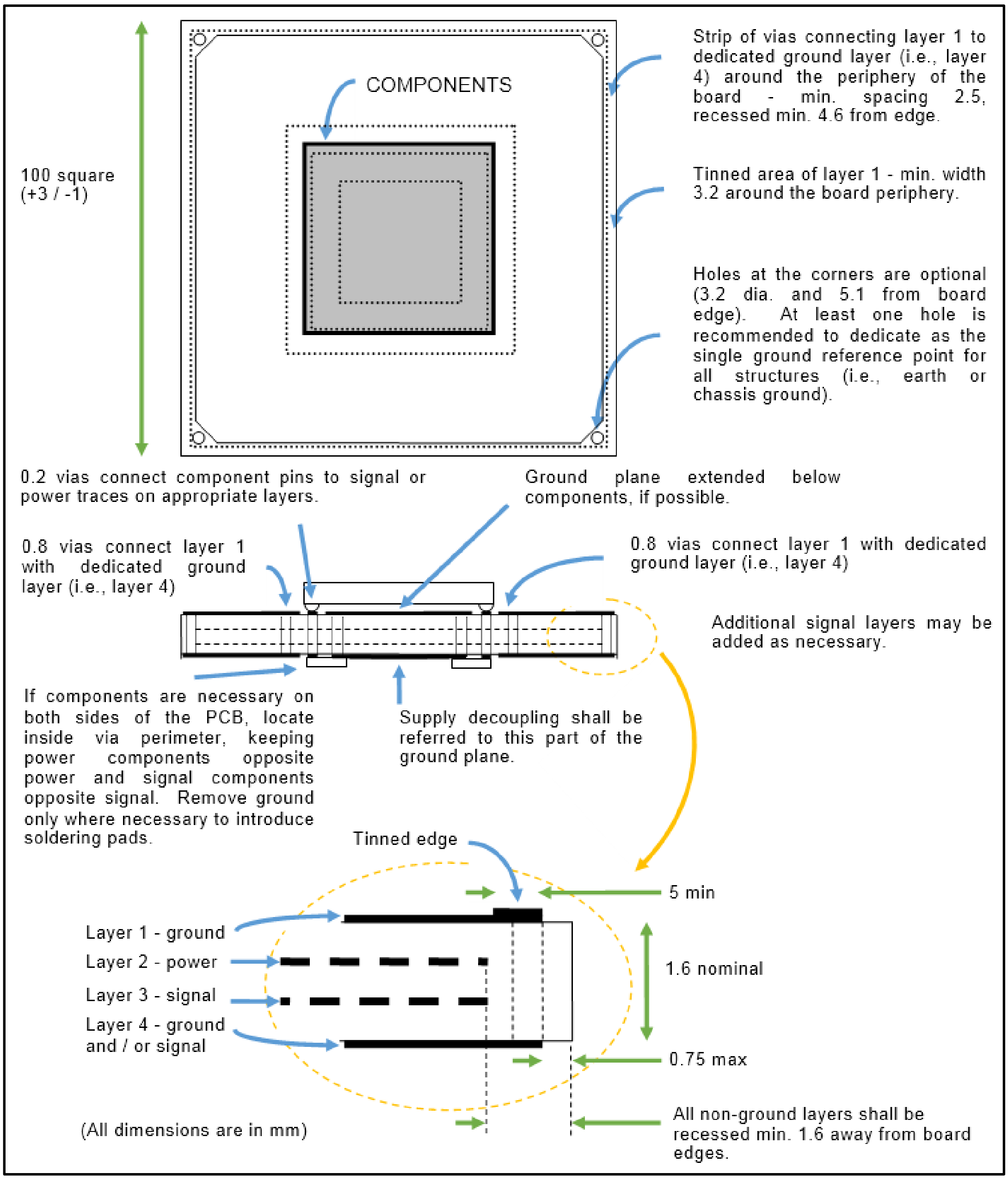
The design team must first decide where on the PCB to locate the definitive ground reference for all signals. This is often a single physical point on the board; possibly where the PCB is attached to a chassis or metallic housing. The connection of this point to a dedicated layer on the PCB helps maintain this reference.

It is then important to ensure that ground paths to this point take the shortest route possible, even if tradeoffs are often inevitable, including limited available areas for ground:

- a single-sided board has no dedicated ground layer, so grounding topologies are very limited.
- a two-sided board has only one layer that can be dedicated to ground, with the power and signals sharing the second layer.
- boards with more than two layers have more flexibility for ground placement and therefore a greater potential for EMI control.

Figure 19: "A multilayer PCB layout (taken from IEC 61967-1)" provides an overview of a multilayer (in this case, four) PCB. It features a grounding shield around the perimeter of the top layer, with many periodic via connections to a dedicated bottom ground layer. Layers two and three are designated for power and signal traces, respectively. This configuration allows quick access to ground for the power and signal traces and a high degree of shielding between critical functional areas.

Figure 19: A multilayer PCB layout (taken from IEC 61967-1)



Actual layout planning can begin once segmentation and PCB selection processes based on grounding have been completed. To recap, the basic grounding objectives are:

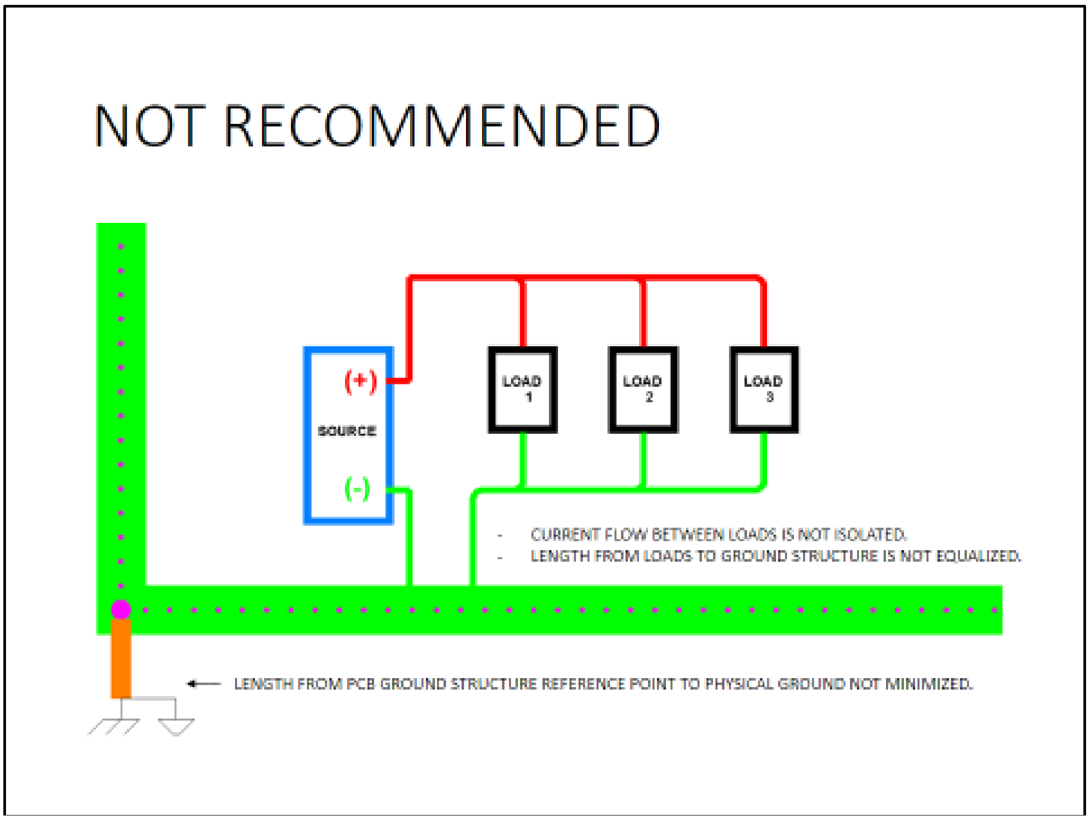
1. to allow electric charge and current to flow from source to load and back to the source via a return path
2. to provide a stable reference potential of 0 Volts
3. to control electromagnetic interference due to electric and magnetic field coupling, i.e., provide adequate isolation

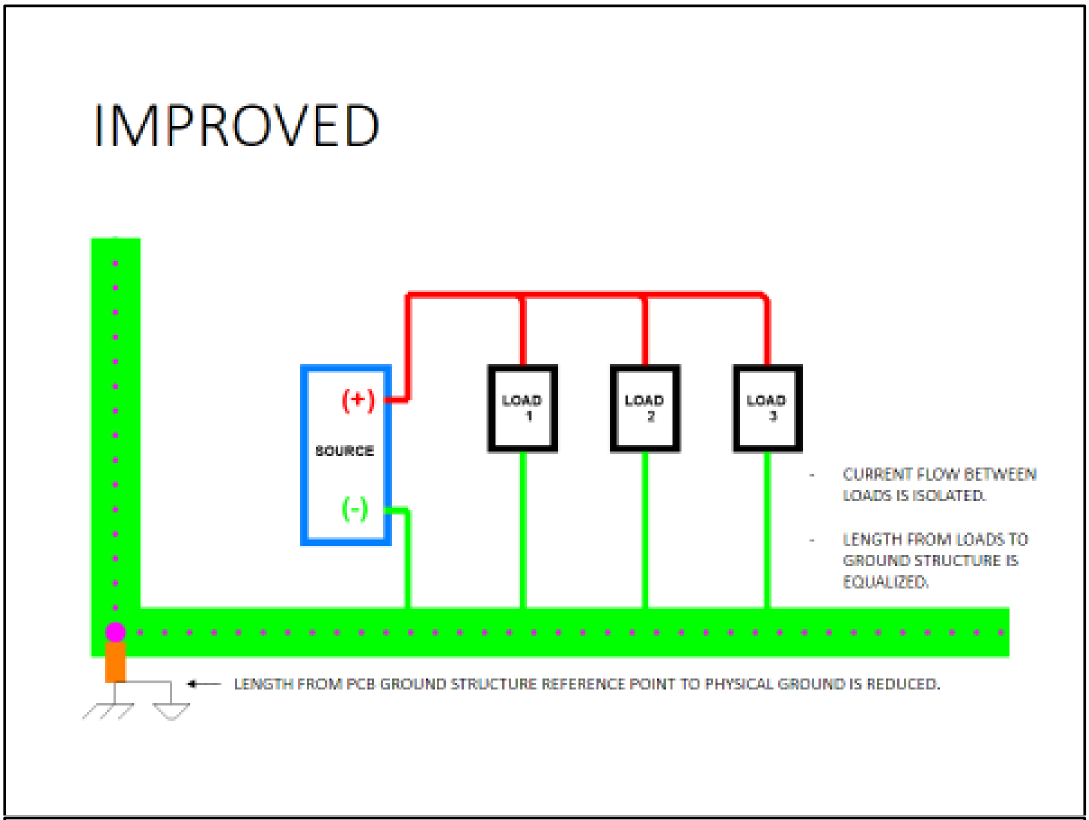
The following EMC-friendly layout guidelines will help achieve the basic goals of grounding and take full advantage of grounding related to EMC.

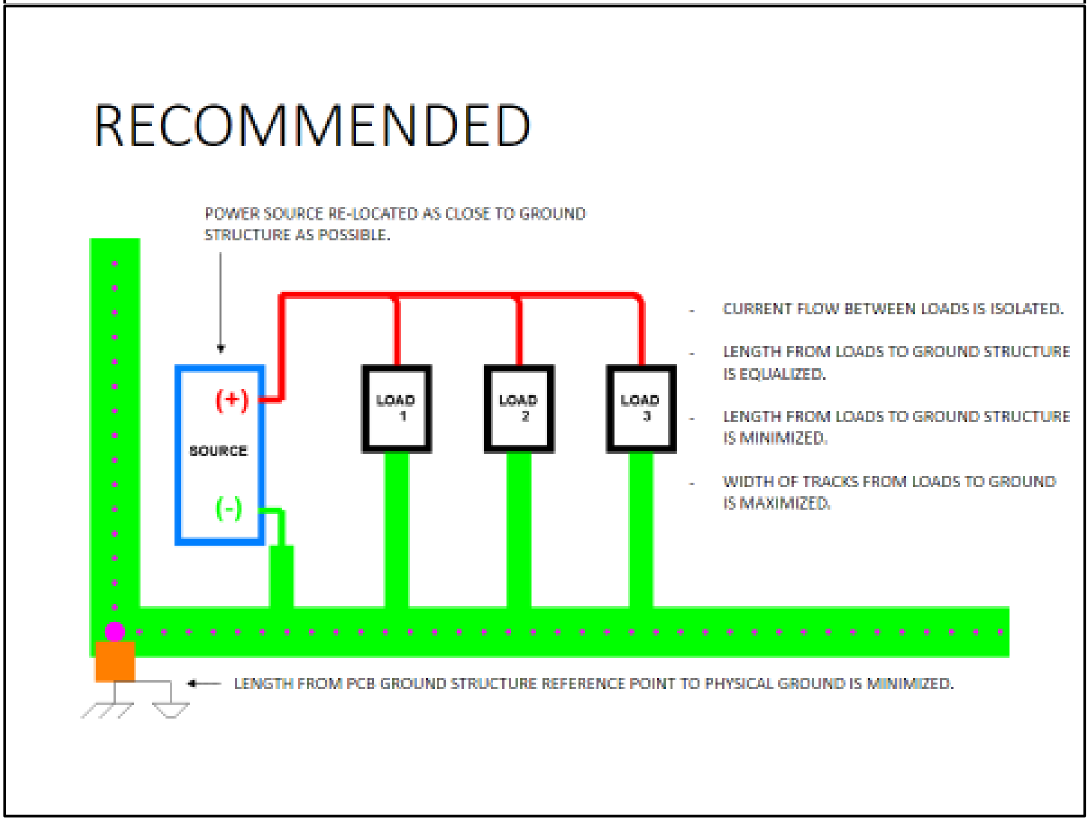
4.4.2.1 Minimizing ground impedance

Dedicating large areas of the PCB to ground and connecting components to these areas along the shortest routes possible minimizes impedance to current flow, thus minimizing ground impedance. Inductance and resistance are minimized by using wide, short traces when immediate connection to the ground plane is not possible. *Figure 20: "Ground loop minimization"* illustrates the principle for reducing ground impedance.

Figure 20: Ground loop minimization




IMPROVED
(+)
SOURCE
(-)
LOAD 1
LOAD 2
LOAD 3
- CURRENT FLOW BETWEEN LOADS IS ISOLATED.
- LENGTH FROM LOADS TO GROUND STRUCTURE IS EQUALIZED.
LENGTH FROM PCB GROUND STRUCTURE REFERENCE POINT TO PHYSICAL GROUND IS REDUCED.


RECOMMENDED
POWER SOURCE RE-LOCATED AS CLOSE TO GROUND STRUCTURE AS POSSIBLE.
(+)
SOURCE
(-)
LOAD 1
LOAD 2
LOAD 3
- CURRENT FLOW BETWEEN LOADS IS ISOLATED.
- LENGTH FROM LOADS TO GROUND STRUCTURE IS EQUALIZED.
- LENGTH FROM LOADS TO GROUND STRUCTURE IS MINIMIZED.
- WIDTH OF TRACKS FROM LOADS TO GROUND IS MAXIMIZED.
LENGTH FROM PCB GROUND STRUCTURE REFERENCE POINT TO PHYSICAL GROUND IS MINIMIZED.

4.4.2.2 Signal and power ground connections

Special care must be taken on the connection between the signal and power grounds.

It is preferable to connect signal circuit grounds to power grounds at a single point because the transient voltage drops along power grounds can be substantial due to high values of di/dt flowing through finite inductance.

If signal processing circuit returns are connected to power ground a multiple points, these transients appear as return voltage differences at different points in the signal processing circuitry.

As signal processing circuitry seldom has the noise immunity to handle power ground transients, it is generally necessary to tie the signal ground to the power ground at only one point.

This rule can also be extended to use of ground planes. For power circuits, it is important to have either separate ground planes or to ensure that the high current path on the ground plane does not traverse sensitive signal ground areas on the same plane.

4.4.2.3 Identify high current and voltage (power) blocks

To decrease the likelihood of unwanted emissions, locate power elements as close as possible to ground. This includes blocks which don't only have the highest currents/voltages, but also the highest rates-of-change (di/dt, dv/dt). Such blocks are:

- clocks
- bus buffers/drivers
- power oscillators

Remember that the same high currents and voltages must also flow through the ground path to the actual ground and are thus ground signals. It is important to physically separate the ground signals of these high-power blocks from the ground currents that exist in the lower power or sensitive circuitry blocks.

4.4.2.4 Identify sensitive circuits

Sensitive circuits are the next priority for placement in proximity to a ground plane (but away from the power elements). These blocks are therefore situated further from the actual ground than the power blocks.

It is good practice not to locate sensitive blocks near the edge of the board to decreases the likelihood that these are affected by EMI (susceptibility) from sources of the board. One solution is to arrange the ground between the edge of the board and the sensitive blocks.

Sensitive blocks can be:

- low-level analog or DC signals like current sensing, fault signals and protections
- high-speed digital data

4.4.2.5 Prioritize ground over all routes

When possible, separate all functional blocks from one another with a path to ground. The ground paths should form periodic connections with the actual ground layer as frequently as possible.

The layout designer might accomplish this prioritization by imagining the PCB as starting with a complete conductive copper layer into which all interconnecting traces must be pushed. This perspective ensures that paths to ground are always as short as possible, and blocks are as separate as possible.

4.4.2.6 Types of grounding structures

A ground plane is preferable to ground traces because:

- it reduces common impedance coupling and promotes return current to flow as near to the source current as possible
- it has much lower partial self-inductance and resistance with respect to ground traces, vastly reducing the common impedance effect

By dedicating an entire layer or plane of the PCB to the grounding function, any node of the circuit requiring grounding is simply connected through the board to the dedicated layer, which is the shortest possible path.

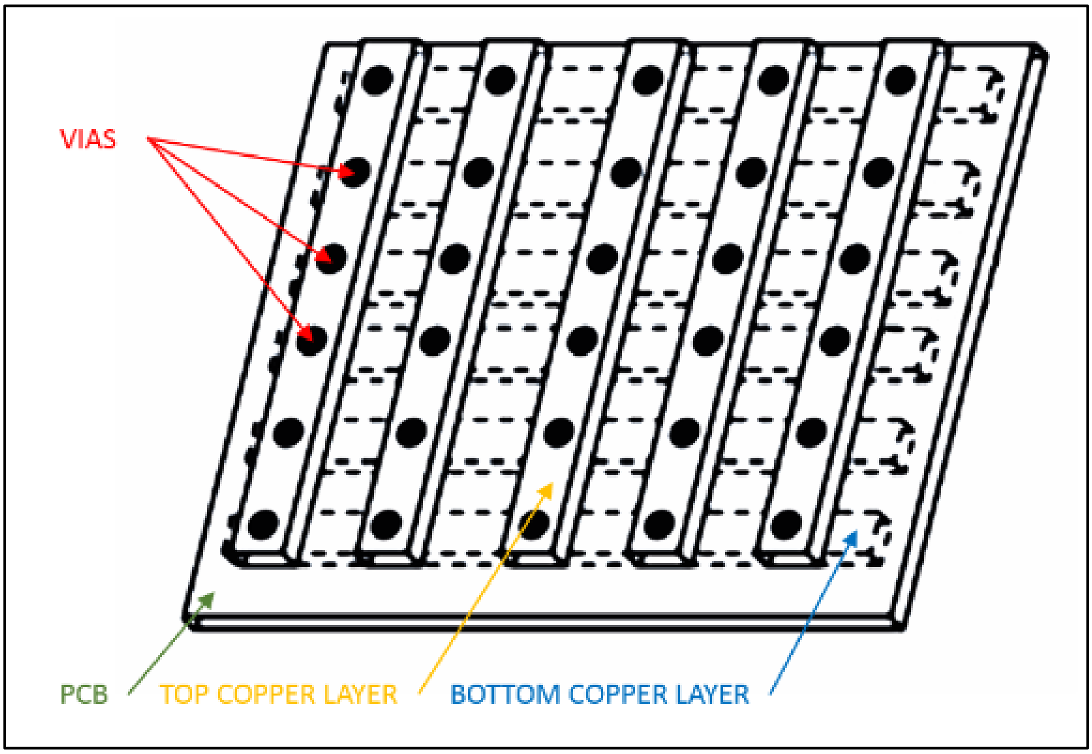
Ground grids can approximate the underlying properties and benifits of PCB ground layers when the latter are too costly or impractical for a particular application, and thus represent the next best solution.

Multiple paths for the signal current are formed across the PCB layers and joined between layers using vias. This allows the return current to follow multiple paths to the common grounding reference, thus providing the path of lowest impedance.

The higher the grid density, the more closely a continuous ground plane is approximated, but the density may be adjusted to make room for components or completely removed from areas where higher component concentrations are required. By maintaining a surrounding grid, there is no drastic change to the low impedance ground path.

Note that schematic groups surrounded by a ground grid are inherently shielded between segments as well as between board layers.

Figure 21: Example ground grid layout



When neither a continuous ground plane nor a ground grid approach is feasible, the path length for return current to reach the ground structure is increased. If the path length is excessive or the trace is too narrow, a ground loop may be formed.

Ground loops are not examples of minimum ground impedance and must be avoided. Current flowing through ground loops can radiate energy from the PCB to sensitive components. External magnetic fields inducted into the loop can cause fluctuation of the true ground reference potential and stray current can flow into the circuit (common-mode current).

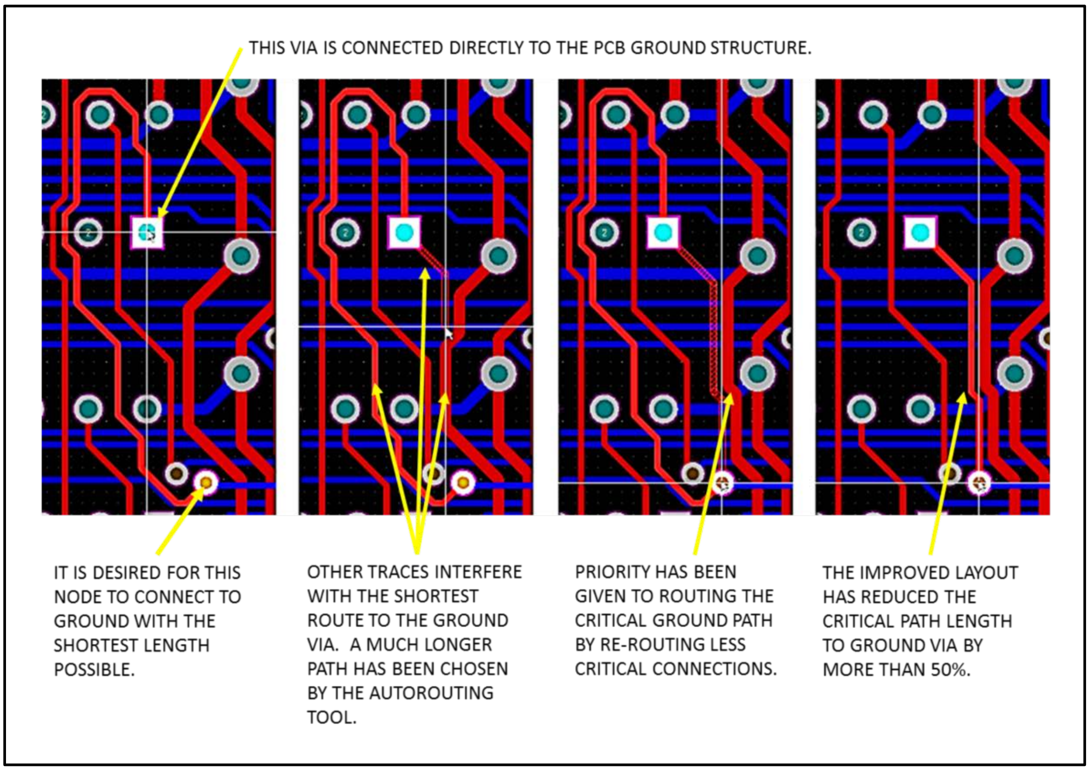
As the layout effort progresses, it is best EMC design practice to continuously monitor for ground loops and take actions to eliminate them. For example, ground loop hazards frequently develop near power components since the current flowing through them is high. Referring back to *Figure 20: "Ground loop minimization"*, the recommended practice illustrates that the return current path, and hence the ground loop impedance, is minimized by re-locating the power source closer to the physical ground reference and connecting the loads using the shortest and widest route.

Examine each power source to ensure its supply and return traces are laid out near each other and in the smallest possible area. This ensures the minimum inductance between different current loops. If a ground loop is unavoidable, it is best to use capacitive bypassing between the offending source and return traces.

Figure 22: "Improved ground loop" shows a small section of a high density PCB with a potential ground loop problem. As circuit density increases, it becomes more difficult to identify ground loops and layout tools may sometimes choose trace routings that are not ideal.

Here, the layout designer identified a large loop while reviewing the layout for critical ground nodes and re-routed non-critical traces to give priority to the ground trace, thus significantly reducing the ground loop impedance.

Figure 22: Improved ground loop



4.4.3 Signals

The types of signals to be conducted by the traces on the application board are control, power and ground. Each type of signal has a unique influence on EMI behavior, so it is best to apply the appropriate set of design guidelines to the traces of each signal type.

For example, control signals may have low amplitude voltages or current, so the conductors dedicated to routing these signals can be narrower. On the other hand, these control signals may be high-speed complex waveforms rich in harmonics, so longer traces may emit higher EMI, or their traces may be susceptible to sources of EMI.

The ground currents associated with these control signals behave similarly and should be treated together.

Power signals have high voltages or currents, and might require very wide isolations between traces for regulatory constraints, or to avoid excessive resistive voltage drops.

The ground currents associated with these power signals behave similarly and should be treated together.

As both control and power signals have corresponding ground signals, these should also be isolated along with the control and power signals themselves.

Finally, place sensitive and high frequency tracks away from high-noise power tracks (current sensing, fault signals and protections), avoiding the use of wire jumpers and minimizing layer transitions. Where necessary, keep the same number of vias on each signal track.

4.4.4 Coupling paths (crosstalk)

One of the more effective means of achieving EMC requirements is to focus on coupling paths from EMI sources to EMI sensitive components.

A coupling is a connection between two or more elements and, in electronics, it refers to the influence of one or more circuit elements on other elements. It is even possible for a circuit element to be coupled with itself via a parasitic or undesired path.

When dealing with so many coupling paths, it is helpful to first categorize them as described below.

4.4.4.1 Conductive path: direct connection of energy between points

A conductive path can be:

- intentional - such as to conduct a required signal from one function to another; an example of this would be any ordinary transmission line/trace
- unintentional - resulting in one type of signal interfering with another or even the same signal

Normally, conductive paths are those formed by copper traces on the board.

4.4.4.2 Radiated path: indirect connection of energy between points

A radiated path can be:

- intentional - such as a trace which is designed specifically to resonate in such a way that the conducted signal transmits (directionally or omni-directionally) into the air; an example of this is an ordinary antenna
- unintentional - such as a conductor being an inopportune fraction of a signal’s wavelength resulting in an undesired transfer of energy to the air; an example of this might occur when “jumping” from one location on a PCB to another with a wire, unintentionally forming a dipole antenna

Remember that radiated paths are often efficient sources of EMI as well as efficient susceptors of EMI.

It is of utmost importance to consider not only the fundamental frequency of the signals, but also their harmonic components. Often overlooked, a source of EMI at the 2nd, 3rd or 10th harmonic can be nearly as high in amplitude as the fundamental.

While PCB traces often form efficient radiators of and susceptors to electromagnetic signals, so too do many lumped component elements such as long leads, transformer windings, etc.

A few of the rather well-known coupling paths, most often associated with lumped component elements, are discussed below. It is however important to note that the layout of traces on a PCB, depending on the frequency of operation, can intentionally or unintentionally behave similarly to lumped components.

4.4.4.3 Capacitive coupling path

This is the coupling of an alternating signal through a dielectric medium to route it from one function to another, while blocking the flow of direct current.

A capacitive path can be:

- intentional - such as the intentional de-coupling or bypassing of a transient signal from a conductor to ground using a capacitor, thereby preventing the signal from traversing into other circuitry and possible malfunction
- unintentional - such as choosing a capacitor with good conductive properties regarding the fundamental (design) frequency, but poor properties with respect to passing harmonic frequencies, thereby allowing the harmonic energy to traverse into other circuitry, possibly resulting in malfunction

When filtering an alternating signal, consider that the modification of a signal depends on the circuit topology, such as:

- whether or not the capacitance is in series with or parallel to the circuit, and
- the reactance to the signal from the capacitance is 1/2 pifC, and
- the impedance to the harmonic components of the signal

4.4.4.4 Inductive coupling path

The blocking of an alternating signal caused by the presence of an opposing magnetic field in an electrical conductor, while passing the flow of direct current.

An inductive path can be:

- intentional - such as when an inductor is positioned between circuit functions and designed to present an open-circuit to a transient signal, thereby preventing signal passage
- unintentional - such as a condition where the chosen inductor does not exhibit open-circuit performance at harmonic frequencies of a transient signal, thus allowing the harmonics to pass as noise

As many inductors are physically large and/or above the board, unintended behavior could include antenna-like phenomena at certain unanticipated frequencies.

When filtering an alternating signal, consider that the modification of a signal depends on the circuit topology, such as:

- whether or not the inductor is in series with or parallel to the circuit, and
- the reactance to the signal from the inductance is 2 pifL, and
- the impedance to the harmonic components of the signal

4.4.4.5 Resistive coupling path

Resistances can be introduced to modify the amplitude and routing of any signal.

A resistive path can be:

- intentional - such as introducing a required 50 Ω termination to minimize reflections on a trace
- unintentional - such as using a narrow trace to conduct a signal over a long distance, thereby resulting in a weak signal or high voltage drop

Any practical or physical connection between nodes or components involves a resistive coupling path, even the copper traces on the PCB and the terminations (leads) of individual lumped component elements can be resistive coupling paths. The design team must determine the impact these resistance coupling paths have on EMC objectives.

Resistors in bias/supply lines from a common power source form potential coupling paths for EMI. In such cases, the supply lines may need to be decoupled.

4.4.4.6 Traces, the primary conductive paths

It is important to make a distinction between the above mentioned coupling paths and a primary conductive path. We define conductive paths as the primary means by which signals are conducted and all the coupling paths above conduct signals by one means or another.

As traces are direct conductive paths used to route signals from one function to another on a PCB, understanding the behavior of traces and how they are routed into paths is key to the successful control of EMI.

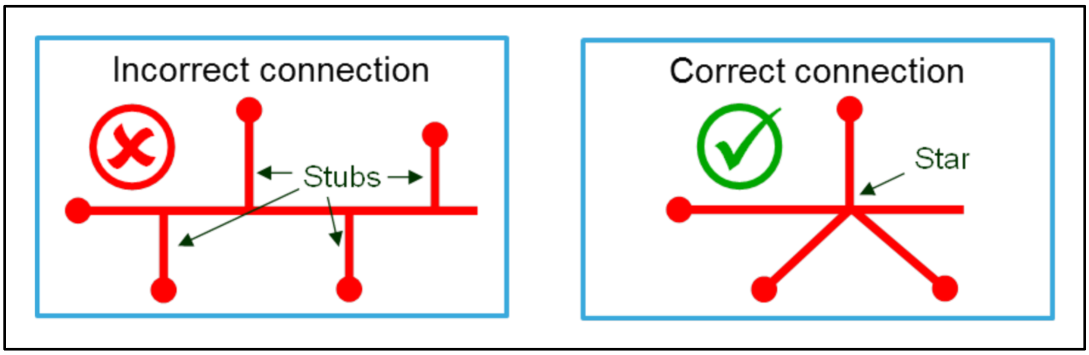
4.4.4.7 Discontinuities

Consider how the components in *Figure 23: "Stub connections"* are joined to a main trace by a short trace segment, or stub. Stubs should be kept as short as possible and frequency-based rules are often applied to limit their length.

Software layout tools often allow the layout designer to limit the maximum length of stubs to fractions of wavelengths, thereby minimizing the potential for EMI radiation from one section of the PCB to another.

Consider how multiple components are connected to a conductive path. Each of these connections represents a discontinuity which can cause signals to be reflected into unknown paths and cause interference. It is preferable to join all the components to the conductor at a single location (star connection).

Figure 23: Stub connections



4.4.5 Component orientation and placement

4.4.5.1 Lumped elements

Resistors, capacitors, inductors and transformers are physical entities often referred to as lumped elements. Lumped elements possess parasitic features which can unknowingly couple EMI signals. While it is generally known that positions and orientations of lumped elements influence specific coupling phenomena, it may be less evident to consider the impact on EMC of side-effects due to parasitic elements of any component.

It is often beneficial to research and possibly characterize lumped elements in advance if this information is not already included in the designers' component database. Such characterization includes the component behavior at the design frequency, temperature ranges, operational voltage and current, as well as other component behavior and harmonic frequencies.

4.4.5.2 Orientation and placement

The strategic placement and orientation of SMT and leaded capacitors, inductors and resistors can significantly reduce EMI generation. Some components have higher magnetic fields in one orientation versus another.

In general, SMT and leaded components should be mounted to achieve the lowest possible profile on the PCB to minimize magnetic fields. Therefore rectangular-shaped SMT components should be orientated with longest side parallel to PCB and leaded components should be positioned as close to the PCB as possible with minimum lead length. When connecting one terminal of an SMT or leaded component to ground, the component should be placed on the PCB so the ground connection is made to the lowest impedance ground point in that area of the PCB.

4.4.6 Shielding

When access to preferable grounding techniques is limited, discrete shielding solutions such as Faraday cages may be required. Specific components or even regions of the board may be shielded by a metallic case.

5 Layouts

The main objectives regarding the layout of the power stage are:

- to control the noise to the IC gate drivers and, for example, ensure the proper turn-on and -off state for the IGBTs
- to minimize the radiated noise and spike voltages

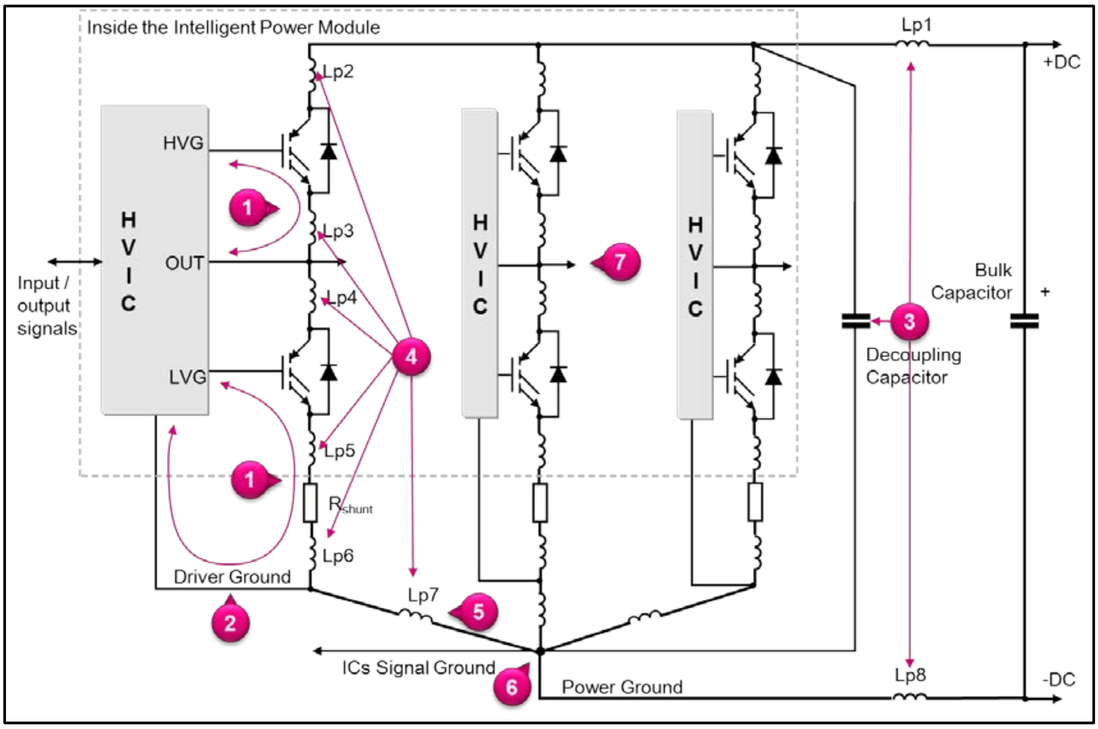
Using a discrete approach, the motor control designer has a higher degree of freedom in achieving the above objectives even if this implies greater complexity in routing the PCB layout.

Conversely, using an IPM approach, several of the high power and low power three-phase power system connections are inside the module (dashed line in *Figure 24: "Suggested layout for a three-phase power system"*) and this aids application design as the internal stray inductances are already optimized by the device manufacturer.

Figure 24: "Suggested layout for a three-phase power system" shows a set of parasitic inductances related to the different circuit tracks of a three-phase power system. The various groups of inductances may have undesired effects which should be minimized.

Whatever the device approach (discrete or IPM), layout optimization is paramount and the following guidelines will help minimize stray inductances and noise.

Figure 24: Suggested layout for a three-phase power system



1. The gate driving PCB traces should be as short as possible and the area of the circuits should be minimized to avoid the sensitivity of such structures to the surrounding noise. This also helps reduce gate driver impedance and prevent dv/dt-induced turn-ons. A possible solution is to route the gate drive signal either directly above or beneath its return.

Typically, a good power system layout keeps the power IGBTs (or MOSFETs) of each half-bridge as close as possible to the corresponding gate driver.

2. We suggest using a separate gate return from the common power ground. If the return track is relatively wide (2.5 mm or greater) it forms a miniature ground plane.
3. Stray inductances in the DC side of the loop (Lp1 and Lp8) should be decreased to limit the voltage transients on the bus. The effects of these inductances can be reduced with a low-ESR decoupling capacitor connected as close as possible to the IGBT terminals.
4. Residual inductances in the DC loop (Lp2, Lp5, Lp6 and Lp7) and stray inductances in the AC loop (Lp3 and Lp4) are the main cause of voltage spikes at turn-off. Furthermore, the group of Lp4, Lp5 and Lp6 stray inductances located between the OUT pin and the ground of the respective driver provides an undesired contribution to the issue of below-ground voltage spikes on the IC gate driver.
   These spikes can be mitigated by good PCB layout based on:
   - short tracks
     – Lp5 may be reduced by placing the Rshunt resistor as close as possible to the emitter of the low side IGBT
     – Lp6 may be minimized by connecting the ground line (also called driver ground) of the related gate driver directly to the shunt resistor
   - paralleled tracks with current flowing in opposite directions - positive and negative planes of the DC bus on top and bottom layers of the PCB and overlapped as much as possible
   - ground planes
   - the use of shunt resistor Rshunt with a low intrinsic inductance
5. Lp7 represents the parasitic inductance located between the ground connections of each gate driver (driver ground) and the ground connection of the application controller (signal ground). This parasitic inductance introduces noise in the input logic signals and the op-amp output analog signals. You can reduce this noise by minimizing the distance between the signal ground and the driver ground (for each gate driver in the system).
6. Connecting the signal ground to the three driver grounds through a star connection improves the balance and symmetry of the three-phase driving topology.
7. It is also useful to ensure some distance between the lines switching with high voltage transitions, and the signal lines sensitive to electrical noise. Specifically, the tracks of each OUT phase carrying significant currents and high voltages should be separated from the logic lines and analog sensing circuits of op-amps and comparators.

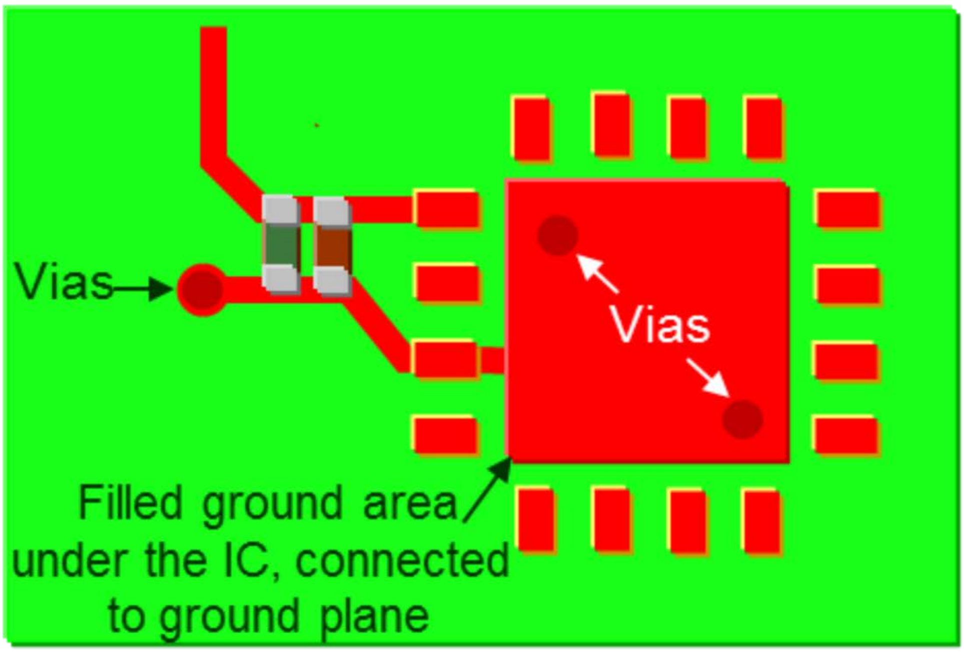
6 Practice case studies

The main aspects in EMC-oriented PCB design are as summarized below.

6.1 Ground

- separate signal ground tracks from power ground tracks and connect with at a single star connection directly on the shunt resistors
- wide ground traces reduce parasitic inductance
- fill unused board spaces with copper areas connected to the ground plane, especially underneath all the high frequency IC (*Figure 25: "Filled copper areas connected to ground plane"*).
- when more than one power supply is required, separate the power and ground tracks
- when a multilayer PCB is used, implement a complete ground layer or place ground traces in parallel with power traces to keep the supply clean

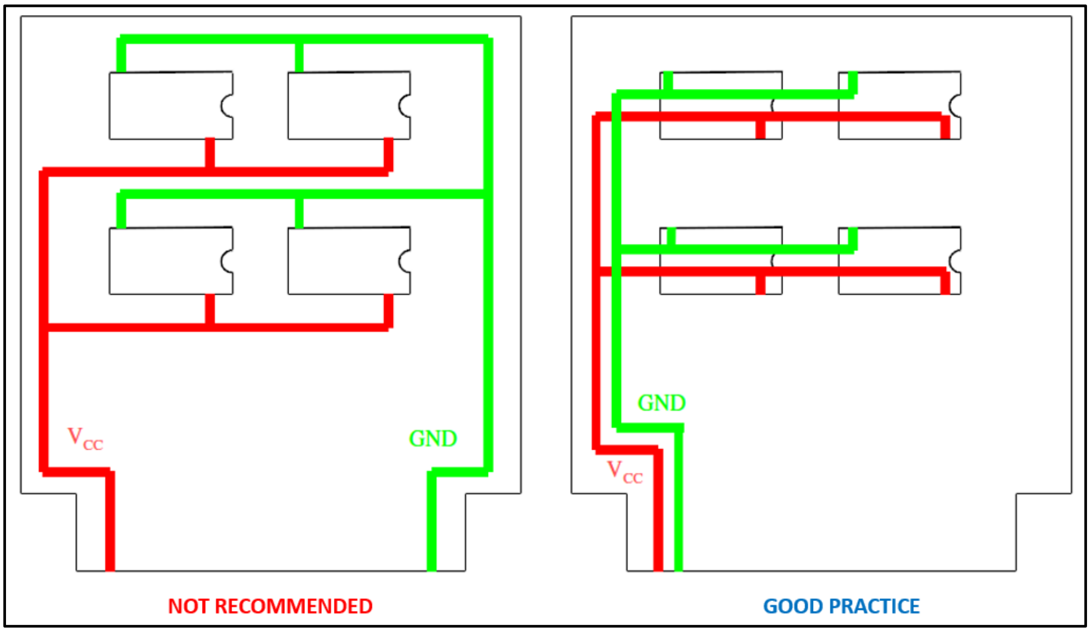
Figure 25: Filled copper areas connected to ground plane



6.2 Power

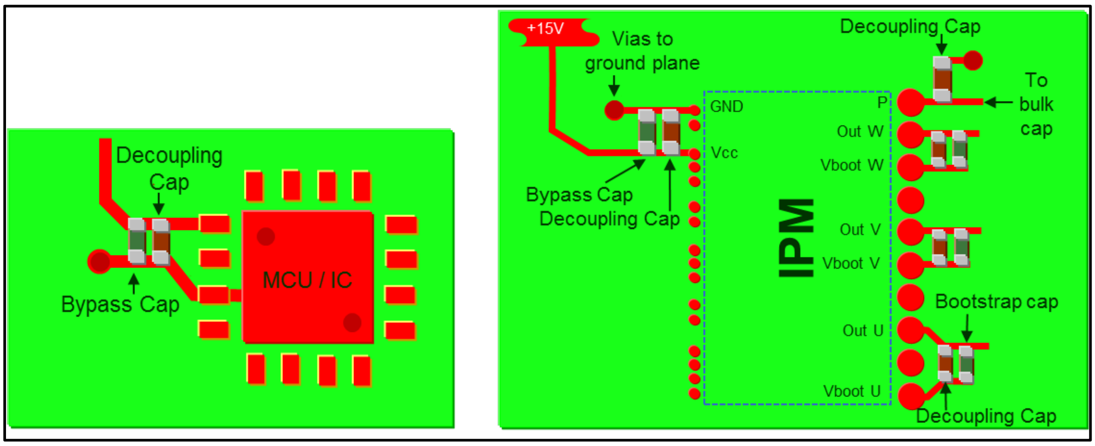
- route parallel to ground on the same or adjacent layers to minimize loop area (*Figure 26: "Routing supply and return traces"*)

Figure 26: Routing supply and return traces



- PCB planes or wide traces reduce parasitic inductance
- bypass capacitors (aluminum or tantalum) placed as close as possible to each IC and IPM reduce the transient circuit demand on the power supply
- decoupling capacitors (with low ESR) placed as close as possible in parallel with the bypass capacitor (*Figure 27: "Example decoupling capacitor placement"*) reduce high frequency switching noise on the power supply lines
- a 21 V Zener diode connected to each power supply pin prevents surge destruction
- a decoupling capacitor (with low ESR) in parallel with each bootstrap capacitor filters high frequency disturbances
- a 21V Zener diode in parallel with each bootstrap capacitor prevents surge destruction
- a decoupling capacitor (with low ESR) in parallel with the electrolytic bulk capacitor filters surge voltage; both capacitors should be placed as close as possible to the power device (the decoupling capacitor has priority over the bulk capacitor)
- use low inductance shunt resistors for phase leg current sensing
- minimize the wiring length between the shunt resistor and power ground to avoid malfunctions
- connect signal ground and power ground at only one point (near the terminal of the shunt resistor) to avoid any malfunction due to power ground fluctuation

Figure 27: Example decoupling capacitor placement



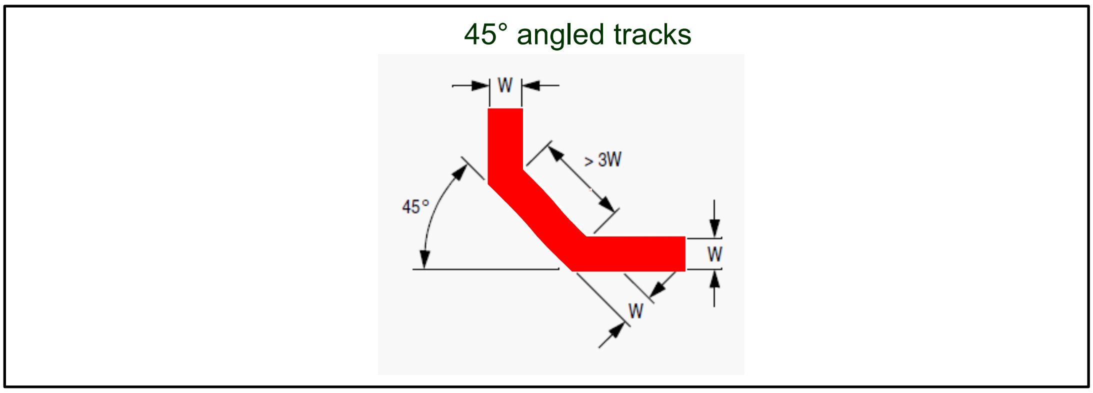
6.3 Signal

- increase the distance between adjacent tracks and separate them to minimize capacitance coupling interference
- place sensitive and high frequency away from high noise power tracks
- on double-layers boards, place signal and power tracks on the same side and ground on the other
- do not use wire jumpers, minimize layer transitions for critical signal traces and keep the same number of vias on each signal track where necessary

6.4 PCB

- group components according to their functionality (analog, digital, power, low-speed and high-speed sections)
- place a filter at subsystem boundaries to promote signal flow between different sections
- minimize the number of vias, especially in the high frequency signal tracks, as they introduce parasitic impedance; distribute them around the PCB, avoiding concentrations in small areas
- avoid right-angled track turns as these produce fields at the inner edge; 45° angled tracks are preferable (*Figure 28: "Example angled track"*)
- prefer star connections to stub connections, especially on critical signal tracks, as the latter produces reflections
- keep a constant signal track width during the entire routing as variations change its impedance and produce reflections
- unused pins cannot be unconnected and must be pulled-up or pulled-down
- respect current flow in layout design in EMI input filters

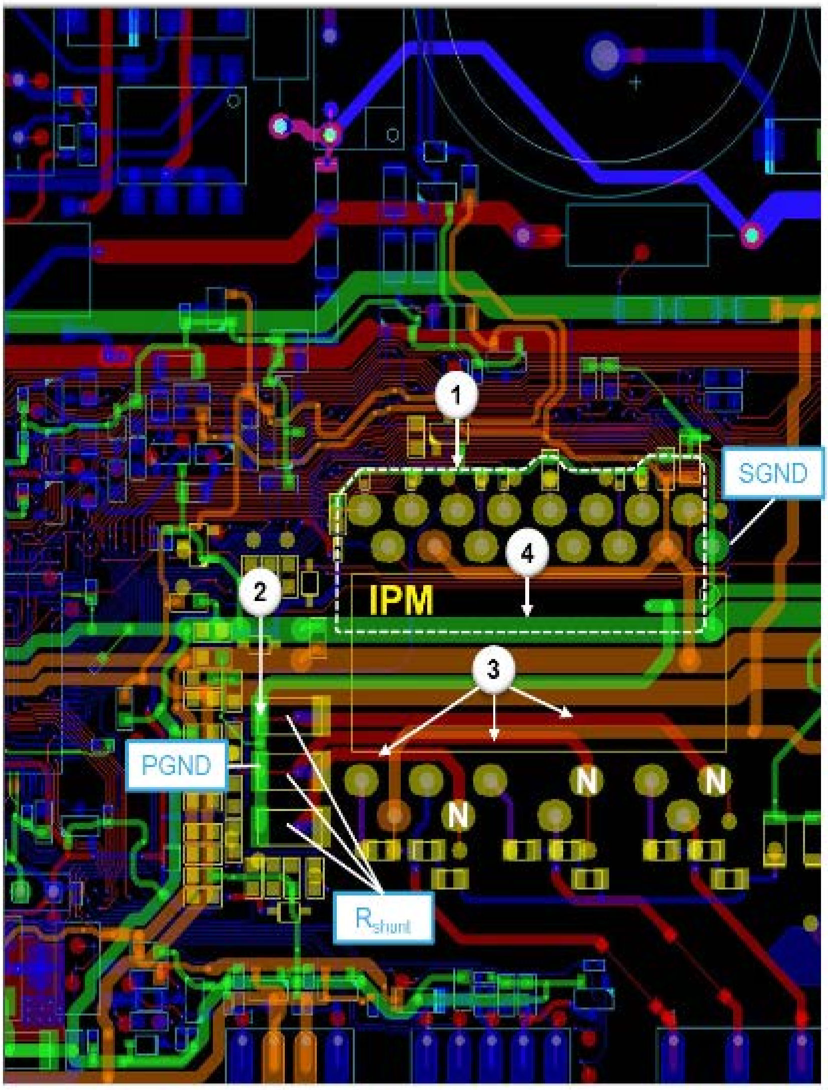
Figure 28: Example angled track



The following case studies reveal some examples of PCB layouts requiring improvement and some enhanced design practices regarding EMC.

6.5 Case study 1

Figure 29: Case 1

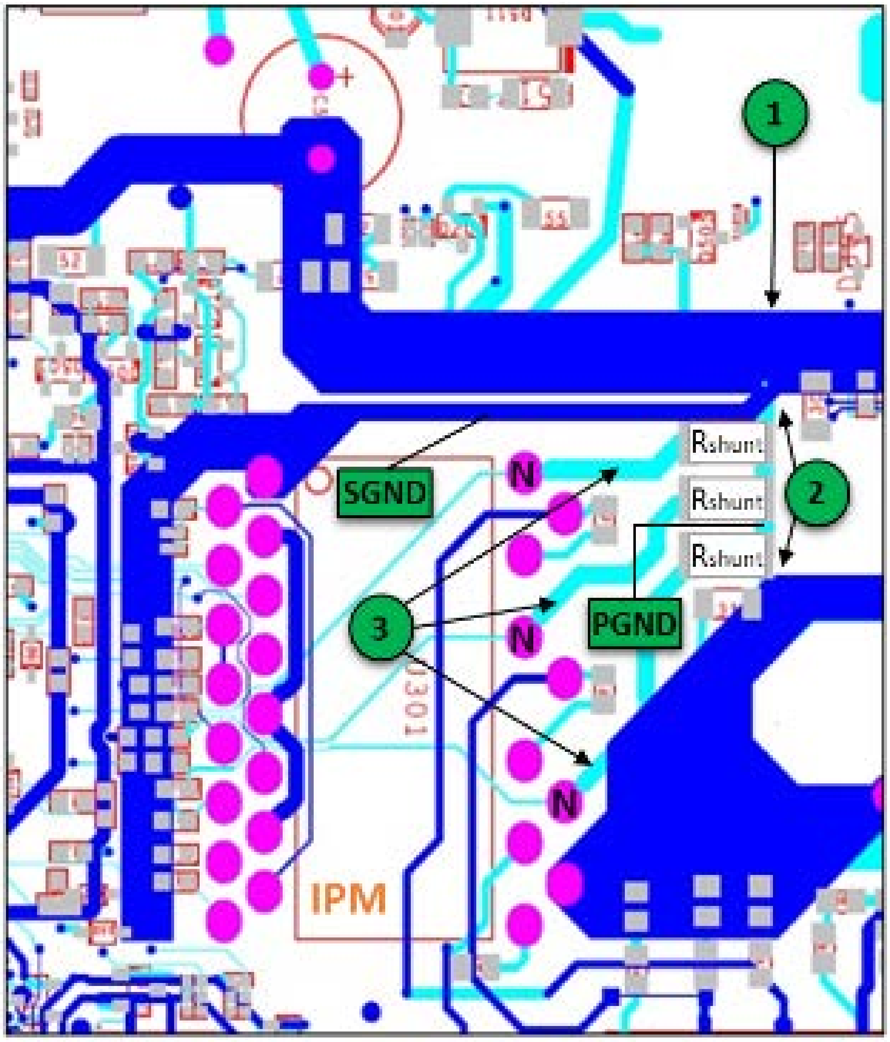


1. ground tracks form a closed loop (white dashed line in *Figure 29: "Case 1"*) which may increase EMC problems by introducing noise into the ground (due to high voltage switching tracks) and affect driver or application performance
2. the signal ground (SGND) of the IPM is connected away from the power ground (PGND); all signal grounds must be connected in a star configuration to the power ground
3. the shunt resistors are too far from the N-pins of the IPM and connected asymmetrically (the net lengths are too dissimilar)
4. power ground tracks are too narrow; this may increase parasitic inductance

By implementing solutions such as rotating the IPM to improve the symmetry of the PCB, the layout becomes the one shown in *Figure 30: "Case 1 improved layout"*, where:

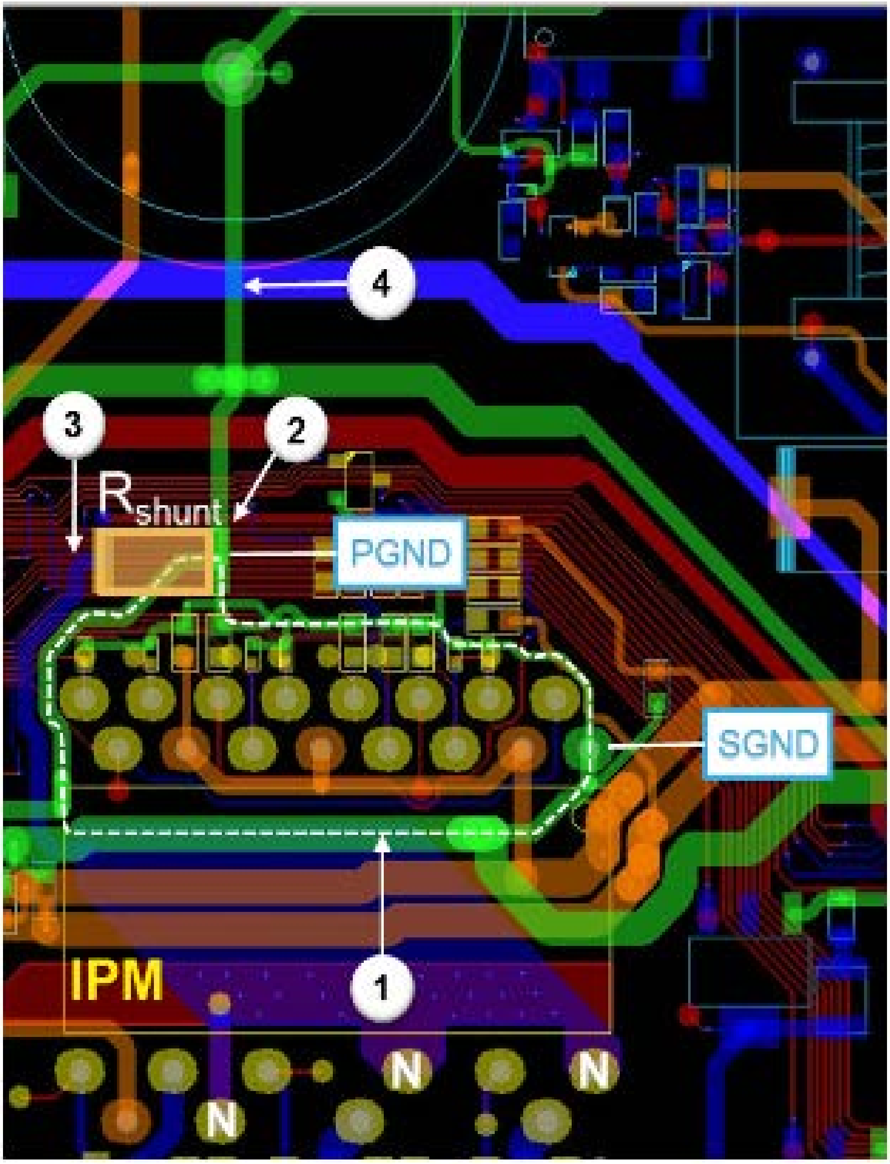
1. the ground path has been reshaped to remove any loops and increase the width/length ratio of the tracks
2. the shunt resistor ground has been changed to a star point configuration for both signal and power grounds
3. the shunt resistors have been relocated to guarantee shorter and symmetric connections with the N pins of the IPM

Figure 30: Case 1 improved layout



6.6 Case study 2

Figure 31: Case 2



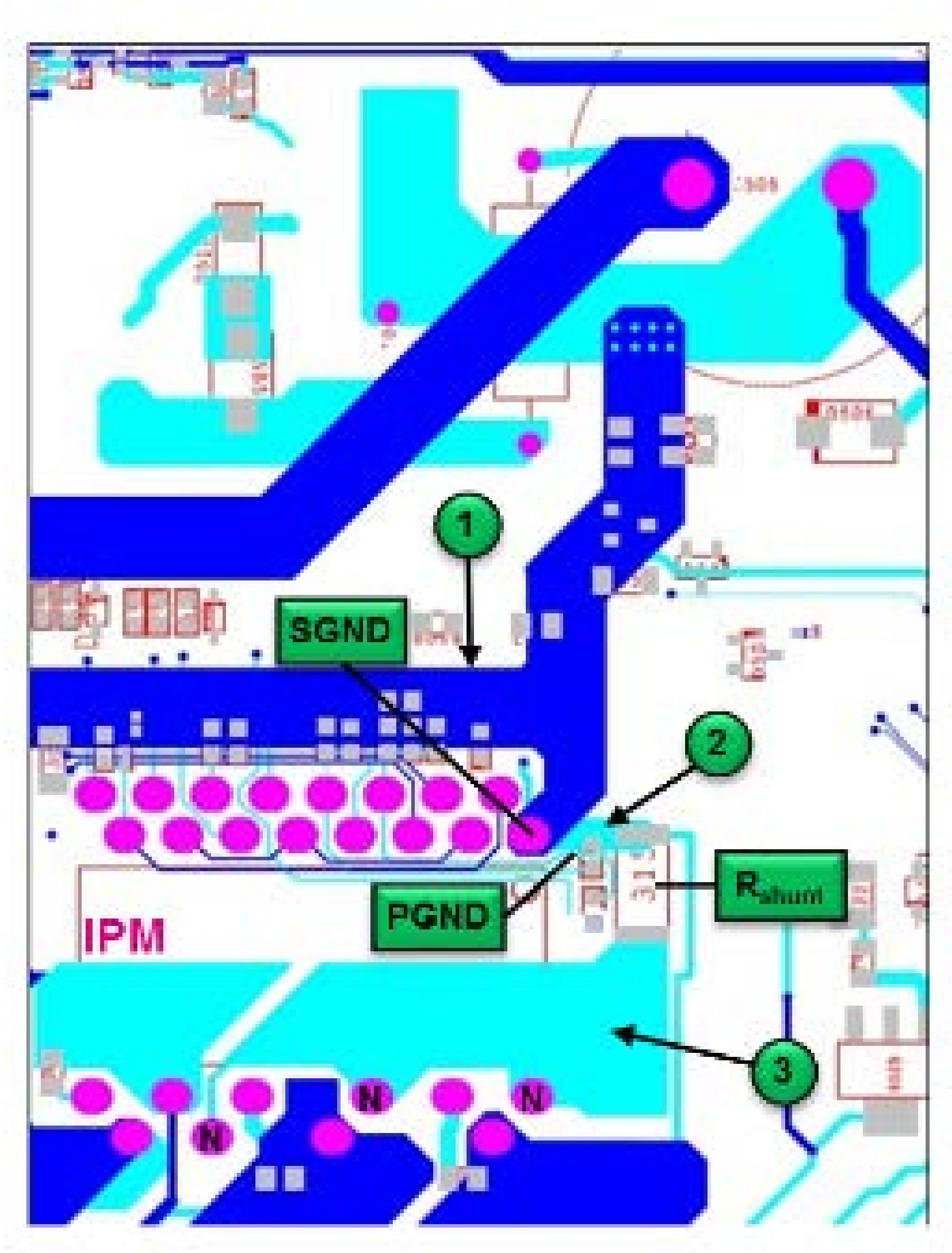
1. ground tracks form a loop (white dashed line in *Figure 31: "Case 2"*) which may increase EMC problems by introducing noise into the ground (due to high voltage switching tracks) and affect driver or application performance
2. the signal ground (SGND) of the IPM is connected away from the power ground (PGND); all signal grounds must be connected in a star configuration to the power ground
3. the shunt resistor is too far from the N pins of the IPM

4. the power ground tracks are too narrow; this may increase parasitic inductance

The above layout can be improved by moving the bulk capacitor and shunt resistor closer to each other as shown in *Figure 32: "Case 2 improved layout"*, where:

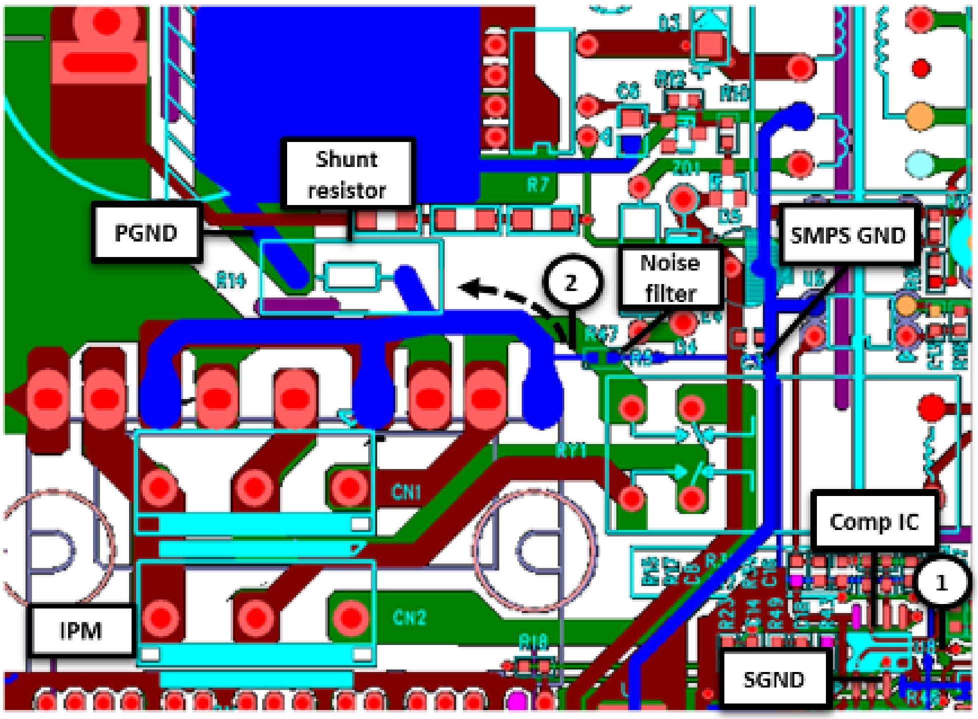
1. the ground path has been reshaped to remove any loops and increase the width/length ratio of the tracks
2. the shunt resistor ground has been changed to a star point configuration for both signal and power grounds
3. the shunt resistor has been relocated to shorten connections with the IPM N pins

Figure 32: Case 2 improved layout



6.7 Case study 3

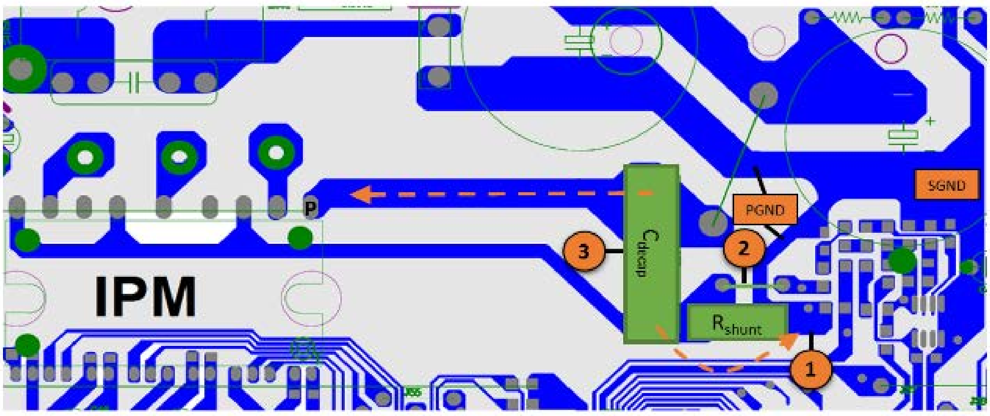
Figure 33: Case 3



1. the comparator ground for OCP (SGND) is too far from shunt resistor R14 (PGND) and it crosses the SMPS power supply ground; finally, the comparator IC should be closer to the shunt resistor and away from noisy tracks
2. the current sensing track should be connected directly to shunt resistor R14and the filtering capacitor must be placed as close as possible to the comparator pin to reduce the level of noise that could trigger false overcurrent protection

6.8 Case study 4

Figure 34: Case 4



1. separate the signal ground tracks from the power ground tracks and connect them at a single point by using the shunt resistors in a star configuration; widen the power ground track
2. jumpers on current sensing tracks should be avoided; the RC filter ground must be connected to the IC comparator ground
3. the decoupling capacitor on the bus voltage should be connected between the P pin of the IPM (as close as possible) and the power ground (on the shunt resistor)

6.9 Case study 5

Figure 35: Case 5

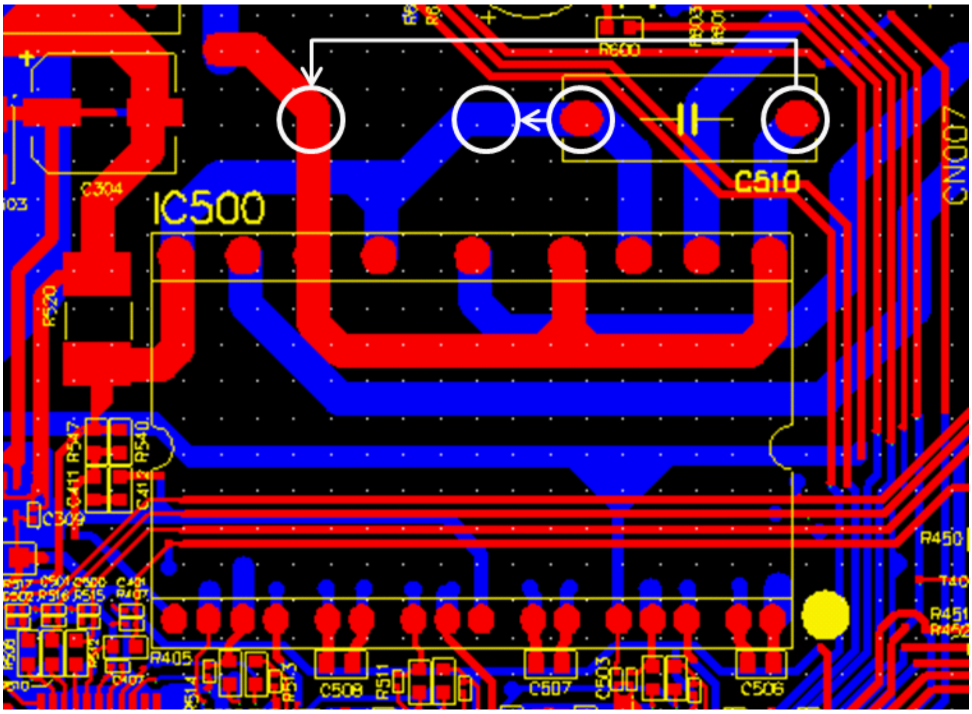


Figure 35: "Case 5" shows a PCB layout with decoupling capacitor C510 in parallel with the bulk capacitor to absorb the charge from an ESD zap before it can accumulate on the IPM, placed downstream of the device.

The effectiveness of the bypass capacitor can be improved by relocating it upstream of the IPM (white lines in *Figure 35: "Case 5"*) to detect the bus signal before it reaches the device.

6.10 Case study 6

Figure 36: Case 6

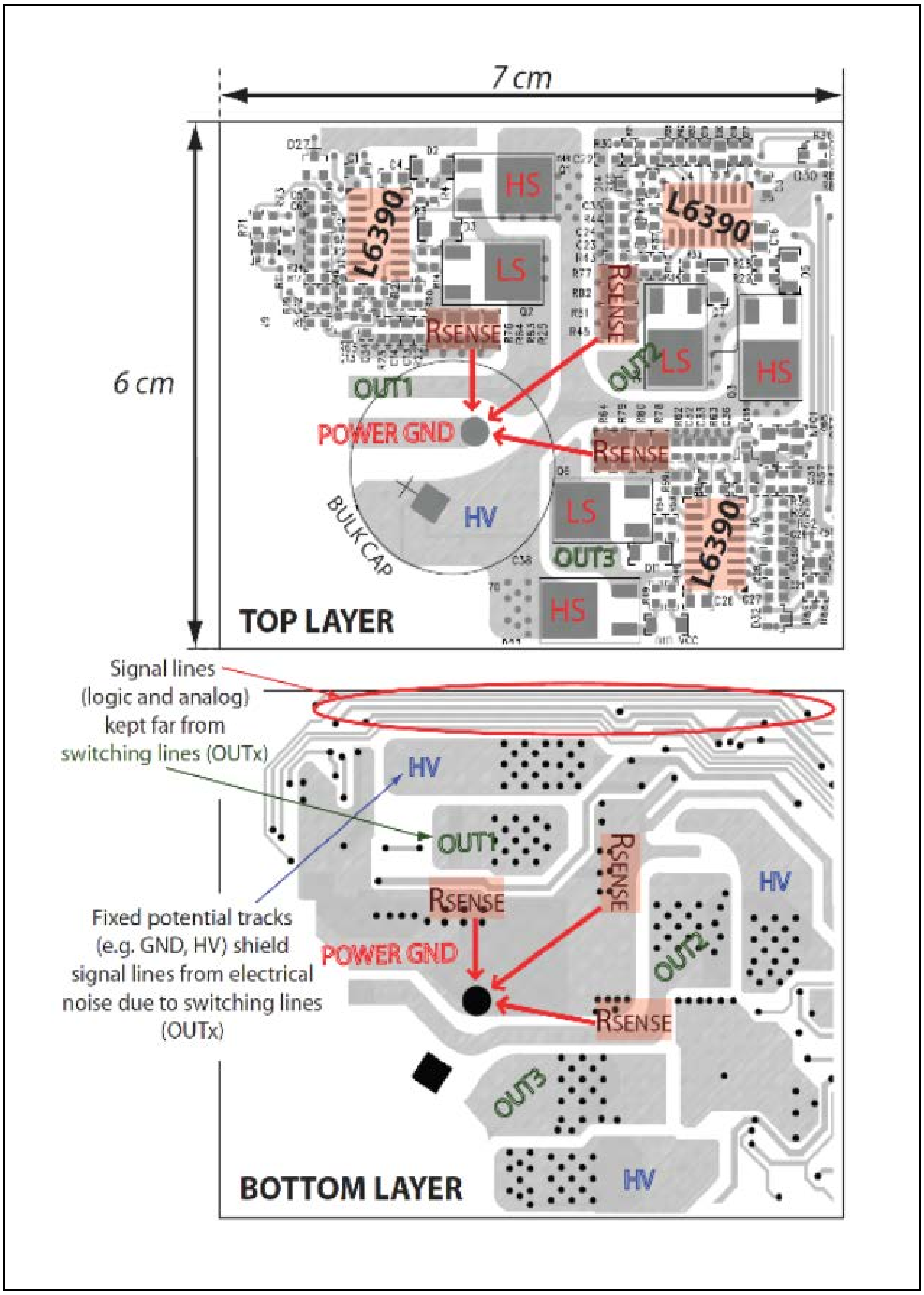


Figure 36: "Case 6" shows a practical example of a layout using the discrete approach. It represents the three-phase power stage section of the STEVAL-IHM021V1 demonstration

board, which includes three IC gate drivers (L6390), six IGBTs (or MOSFETs) in a DPAK package and a bulk capacitor.

The fixed voltage tracks such as GND and HV lines can be used to shield the logic and analog lines from the electrical noise produced by the switching lines (OUT1, OUT2 and OUT3). Each half-bridge ground is connected in a star configuration and the three RSENSE resistors are very close to each other and the power ground.

7 References

[1] ST AN3353 – IEC 61000-4-2 standard testing

[2] ST AN1709 – EMC design guide for ST Microcontrollers

[3] ST AN2738 – L6390 half-bridge gate driver

[4] ST AN3338 – SLLIMM™

[5] ST AN4043 – SLLIMM™-nano

[6] Electromagnetic Compatibility (EMC) Part 4-2: Testing and Measurement Techniques—Electrostatic Discharge Immunity Test (IEC 61000-4-2:2008 (Ed.2.0))

[7] Electromagnetic Compatibility (EMC) Part 4-4: Testing and Measurement Techniques—Electrical Fast Transient/Burst Immunity Test (IEC 61000-4-4:2012 (Ed3.0))

[8] EN 55 014 European limits and methods of measurement of radio disturbance characteristics of household appliances and power tools

8 Revision history

Table 3: Document revision history

Date	Revision	Changes
05-Jun-2015	1	Initial release.