

Technical White Paper

Mitigation Techniques for EMI Sources for EMC Compliance



Abstract

This white paper provides a comprehensive analysis of electromagnetic interference (EMI), a critical challenge in the design and operation of modern electronic systems. As device density increases and operating frequencies rise, the risk of performance degradation and system failure due to unwanted electromagnetic disturbances grows. This study first categorizes the primary sources of EMI, distinguishing between natural phenomena and human-made emissions from high-speed digital circuits and switching power supplies.

Furthermore, the paper evaluates essential *mitigation techniques*, including the application of *shielding*, *filtering*, and *grounding* strategies to contain or block interference. Special attention is given to *printed circuit board (PCB) layout* optimization, such as minimizing loop areas and maintaining impedance control, as a foundational preventative measure. Finally, the research outlines industry practices and compliance standards, offering a systematic framework for engineers to maintain electromagnetic compatibility (EMC) in complex environments.

Table of Contents

1 Introduction	2
How Does an Electromagnetic Field Start in a Circuit?	2
Near-Field and Far-Field Electromagnetic Fields	4
2 Common Sources of EMI in Electronic Designs	5
2.1 Resistors	5
2.2 Capacitors	5
2.3 Inductors	6
2.4 Summary of Discrete Component EMI Considerations	7
2.5 Design Deficiencies	8
2.6 Sources Specific to MCUs	9
2.7 Other Common Sources	11
3 Mitigation Techniques to Minimize EMI and Optimize Board Performance	12
3.1 Power Supply Capacitor Selection and Placement	12
3.2 Capacitor Placement	13
3.3 Layer Stack-Up Optimization	15
3.4 Trace Routing	19
3.5 Passive Pi Filtering	19
3.6 Shielding, Grounding, and Cable Boundaries	21
3.7 Transient and ESD Current Paths	21
4 EMC Verification and Troubleshooting	22
4.1 Emissions and Immunity as Port Problems	22
4.2 Pre-Compliance Measurement Toolkit	22
4.3 A Frequency-to-Structure Debug Method	22
4.4 Interpreting Common Spectral Signatures	22
4.5 Integrated Design Review Checklist	23
5 Conclusion	23

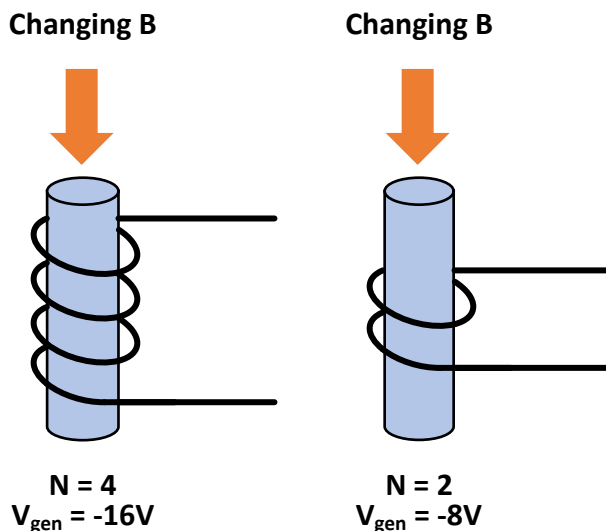
Trademarks

All trademarks are the property of their respective owners.

1 Introduction

How Does an Electromagnetic Field Start in a Circuit?

Any change in the electrical potential of a wire, circuit board trace, or any component that has inductive properties causes a voltage or electromagnetic field (EMF) to be *induced* in the circuit. This EMF produces a current if a load is present. No matter how the change is produced, a voltage is generated and subsequently a current develops across any load. The change can be produced by changing the magnetic field strength, moving a magnet toward or away from the coil, moving the coil into or out of the magnetic field, rotating the coil relative to a magnet or field, and so forth. In simpler terms, any $\Delta V/\Delta t$ can develop electromagnetic lines of force, so if the voltage moves, pulses, ripples, or changes in any way, the voltage can be considered a source of EMF.



$$\frac{\Delta(BA)}{\Delta t} = 4 \text{ Tm}^2/\text{s}$$

Figure 1-1. EMF Induced by Changing Magnetic Flux

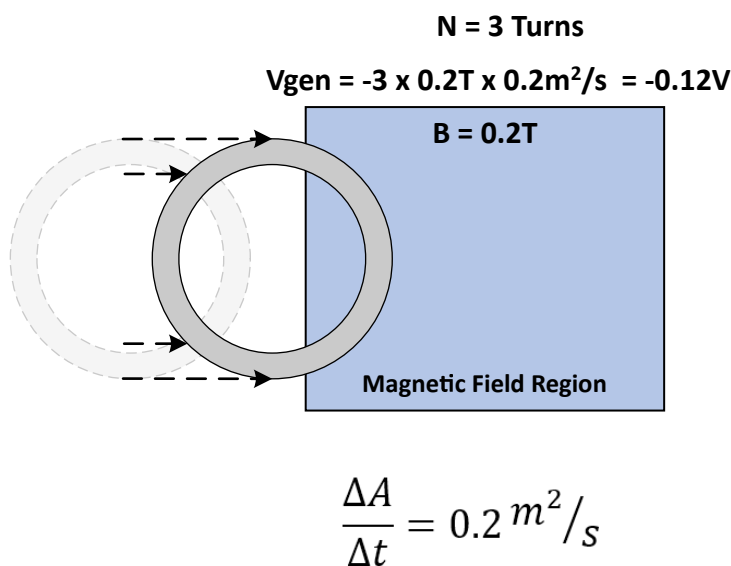


Figure 1-2. EMF Induced by Changing Loop Area

$$N = 5 \text{ Turns} \quad A = 0.002 \text{ m}^2$$

$$V_{\text{gen}} = -5 \times 0.4 \text{ T/s} \times 0.002 \text{ m}^2/\text{s} = -0.004 \text{ V}$$

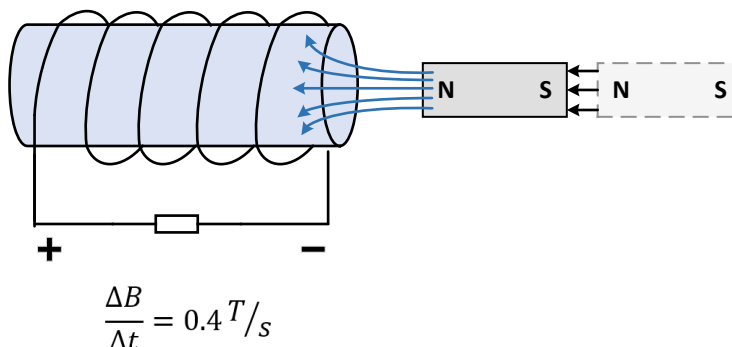
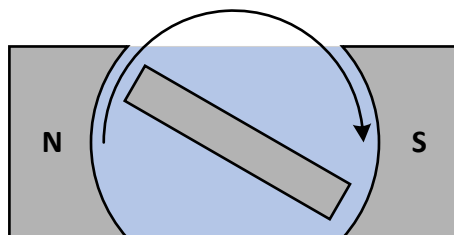


Figure 1-3. EMF Induced by Relative Magnet-Coil Motion

$$N = 20 \text{ Turns}$$

$$V_{\text{gen}} = -20 \times 0.2 \text{ T} \times 0.2 \text{ m}^2/\text{s} = -0.8 \text{ V}$$



$$\frac{\Delta A}{\Delta t} = 0.2 \text{ m}^2/\text{s}$$

Figure 1-4. EMF Induced by Coil Rotation

A schematic net describes connectivity but does not show the physical return path. Every signal current leaves a source and returns to that source. The outgoing trace and return path form a loop whose inductance and field exposure increases with enclosed area. At high frequency, current follows the path of least impedance, which is typically the path of least inductance rather than the path of least DC resistance. A useful mental model is to think in loops, not just nets. The outgoing current on the signal trace and the return current in the reference plane form a loop. The smaller and more co-located that loop is, the less magnetic field is exposed to the outside world. The same idea applies to switching regulator input loops, gate-drive loops, ESD clamp loops, decoupling loops, and cable common-mode loops.

At low frequency, return current tends to spread according to resistance. At high frequency, return current concentrates under or near the signal path because that path minimizes inductance. If the reference plane is continuous, this is beneficial: the outgoing and return fields cancel well. If the plane is split, the return current is forced to detour, the loop expands, and the board can radiate or become susceptible to external fields.

Rise time is often a better warning sign than clock frequency. A 10MHz clock with 500ps edges can contain meaningful harmonic energy well into the hundreds of MHz. For EMC reviews, estimate the highest important spectral content from edge rate and then ask whether the return path, via spacing, enclosure aperture, and cable boundary still look small at that frequency.

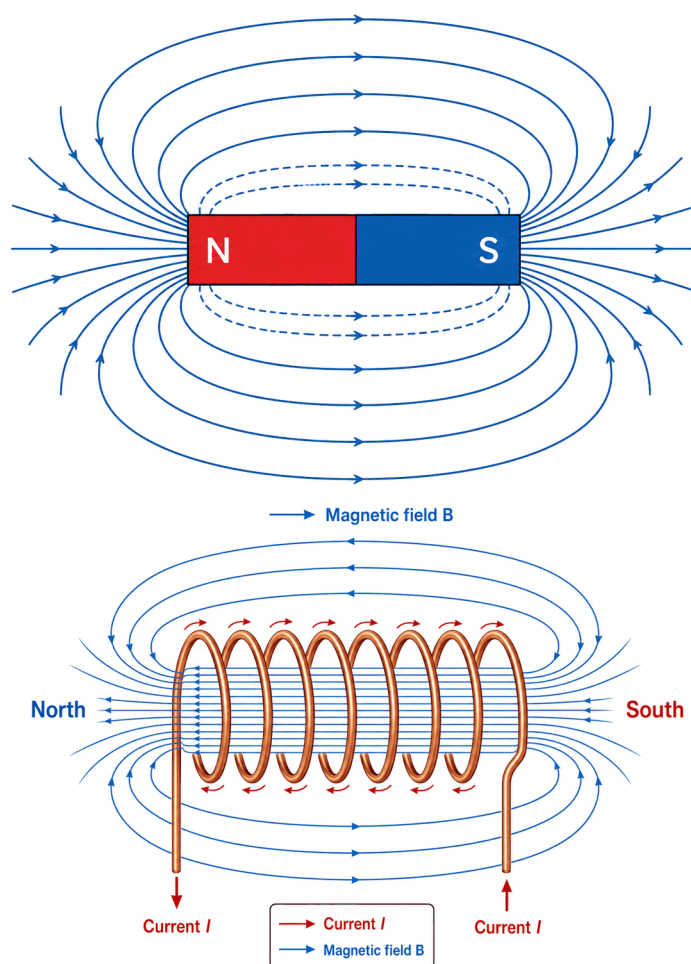


Figure 1-5. Magnetic Fields Induced by Change in Current

Near-Field and Far-Field Electromagnetic Fields

Near-field energy is stored around a source and can be predominantly electric or magnetic. Far-field energy propagates away as a wave with electric and magnetic fields related by the wave impedance of the medium. The transition is gradual and depends on source size, geometry, and observation distance.

Wavelength becomes shorter as frequency rises. A physical feature that is electrically small at a low frequency can become an efficient resonant structure at a higher harmonic.

For a small source, a useful reactive near-field scale is on the order of $\lambda / (2 \times \pi)$. For physically large antennas, the conventional far-field criterion also depends on the largest source dimension. These boundaries are approximations to guide probe selection: magnetic loops are useful near high-current loops, while electric-field probes respond strongly near high $\Delta V/\Delta t$ nodes.

A conductor near one-quarter wavelength can be an efficient resonator, but cables and enclosures often radiate at much smaller electrical lengths because common-mode current excites the entire structure. The practical question is not only whether a PCB trace is a quarter wavelength, but whether unintended current reaches a structure that is large relative to wavelength.

Table 1-1. Near/Far Field Emission Considerations

Region Mode	Dominant concern	Useful diagnostic
Electric near field	High $\Delta V/\Delta t$ nodes and capacitive coupling	Small E-field probe; reduce switch-node area
Magnetic near field	High $\Delta I/\Delta t$ loops and mutual inductance	Shielded loop probe; minimize loop area
Far field	Structure acting as an antenna	Antenna and chamber test; identify common-mode current

2 Common Sources of EMI in Electronic Designs

2.1 Resistors

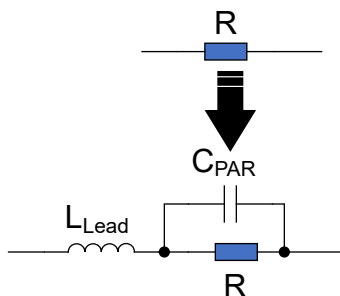


Figure 2-1. Resistor Parasitic Equivalent

In an ideal world, a resistor only has resistance. In reality, a real-world resistor (especially at high frequencies) acts as a complex RLC circuit due to its physical construction. Lead inductance increases impedance with frequency, while shunt capacitance eventually provides a bypass around the resistive element. Wire-wound resistors can be intentionally inductive; thin-film and small surface-mount geometries are typically better for fast damping and termination. A series damping resistor works best when placed at the driver so the resistor combines with source impedance before the wave enters the transmission line. Below are the main parasitic components associated with resistors and the respective impact and significance on a given circuit.

2.1.1 Equivalent Series Inductance (ESL)

Cause: Every conductor has inductance. In leaded (through-hole) resistors, the long metal leads act as inductors. In wire-wound resistors, the coiled resistive wire creates a significant magnetic field.

Impact: As frequency increases, the impedance of the resistor rises. This makes the resistor "choke" high-frequency signals.

Mitigation: Use Surface Mount (SMD) components (like 0402 or 0603 packages) because they have almost no lead length.

2.1.2 Shunt Capacitance

Cause: This is the capacitance between the two terminals of the resistor or between the individual turns of a wire-wound element.

Impact: At high frequencies, this creates a parallel path for current. The high-frequency energy "bypasses" the resistance, causing the total impedance to drop.

Result: This can lead to unwanted oscillations or phase shifts in high-speed circuits.

2.1.3 Parasitic Capacitance to Ground

Cause: The physical body of the resistor sitting over a PCB ground plane creates a small capacitor.

Impact: This can bleed high-frequency signals into the ground plane, causing signal integrity issues or EMI.

2.2 Capacitors

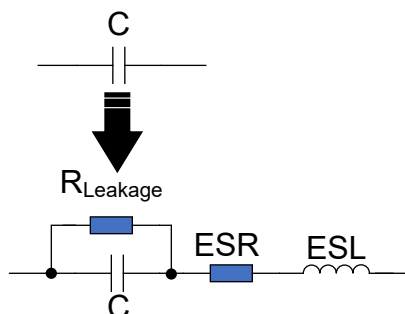


Figure 2-2. Capacitor Parasitic Equivalent

In an ideal world, a capacitor only provides capacitance. In reality, a physical capacitor behaves like a *Series RLC Circuit* due to its materials and construction and can be modeled as ideal capacitance in series with ESR and ESL. Below self-resonance, capacitive reactance dominates. At resonance, ESR limits minimum impedance. Above resonance, ESL dominates and the part behaves like an inductor.

Self-resonance occurs where the capacitive and inductive behaviors balance. The actual mounted resonance depends on the complete connection geometry, not only the component datasheet.

Multiple capacitor values broaden the low-impedance range, but their interaction with plane and via inductance can create anti-resonance peaks. A smaller nominal value is not automatically a better high-frequency capacitor if it is placed farther away or connected through a narrow neck. Below are the main parasitic components associated with capacitors and the respective impact and significance on a given circuit.

2.2.1 Equivalent Series Resistance (ESR)

Cause: The combined resistance of the internal conductive plates (electrodes), the leads, and the dielectric losses.

Impact: This is the primary source of heat generation. High ESR reduces the efficiency of decoupling.

Significance: In power supplies, high ESR can cause voltage ripples and premature component failure due to thermal stress.

2.2.2 Equivalent Series Inductance (ESL)

Cause: The physical loops formed by the internal structure of the capacitor and its mounting leads (or SMD terminals).

Impact: Inductance opposes high-frequency current. At very high frequencies, becomes greater than causing.

Self-Resonant Frequency (SRF): This is the frequency where $X_L = X_C$. Above this point, the capacitor is no longer effective at filtering noise.

2.2.3 Leakage Resistance

Cause: No dielectric is a perfect insulator. A tiny amount of current "leaks" through the dielectric between the plates.

Impact: This is modeled as a very large resistor in *parallel* with the capacitor.

Significance: Critical in battery-powered or precision analog circuits (like sample-and-hold) where it causes the capacitor to slowly discharge over time.

2.2.4 Dielectric Absorption (Soakage)

Cause: The tendency of a dielectric to retain a small charge after being discharged.

Impact: If you discharge a capacitor and let it sit, a small voltage can "reappear" on the terminals. This can cause errors in high-precision integrators or ADC circuits.

2.3 Inductors

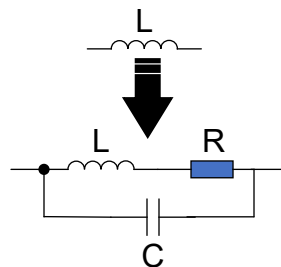


Figure 2-3. Inductor Parasitic Equivalent

In an ideal world, an inductor only provides inductance. In reality, a physical inductor is a complex RLC network due to the resistance of the wire and the proximity of the windings.

The DCR of an inductor causes copper loss, its inter-winding capacitance bypasses high-frequency energy, and its magnetic core introduces frequency-dependent and amplitude-dependent loss. Above self-resonance, the component is capacitive. Under high DC bias, the incremental inductance can decrease; near saturation the inductance can collapse, allowing a rapid current rise that worsens both functional stress and EMI.

Ferrite beads are intentionally lossy inductors. The datasheet impedance of ferrite beads often contains a large resistive component in the intended suppression band. That loss converts RF energy to heat and damps resonance, but the bead must also be checked for DC bias derating and interaction with load-side capacitance. Below are the main parasitic components associated with inductors and the respective impact and significance on a given circuit.

2.3.1 Direct Current Resistance (DCR)

Cause: This is the actual physical resistance of the copper wire used to wind the inductor.

Impact: DCR is the primary cause of power loss and heat generation. High DCR reduces the efficiency of the inductor, especially in power a or causing a significant voltage drop.

Significance: DCR limits the amount of current the inductor can carry before overheating or causing a significant voltage drop.

2.3.2 Parasitic Capacitance

Cause: Also known as inter-winding capacitance. Parasitic capacitance occurs because the individual turns of the wire are separated by an insulator (enamel), creating thousands of tiny "parallel plate" capacitors between the windings.

Impact: At high frequencies, this capacitance provides a "bypass" path for current, allowing high-frequency noise to skip over the inductance entirely. Self-Resonant Frequency (SRF): This is the point where the inductive reactance equals the capacitive reactance. Above the SRF, the inductor behaves like a capacitor.

Self-Resonant Frequency (SRF): This is the point where the inductive reactance equals the capacitive reactance. Above the SRF, the inductor behaves like a capacitor.

2.3.3 Core Losses (Magnetic Parasitics)

If the inductor has a physical core (ferrite, iron powder, and so forth), the inductor also introduces two additional non-ideal behaviors:

Hysteresis Loss: Energy lost as heat when the magnetic domains in the core flip back and forth with the AC signal.

Eddy Current Loss: Small circulating currents induced within the conductive core material, causing further heat.

Magnetic Saturation: While not a "component," magnetic saturation is a parasitic behavior where the core can no longer hold more magnetic flux. The core eventually reaches a flux level beyond which incremental inductance drops sharply. At this point, the inductance drops sharply, and the rapid current increase can potentially leading to massive current spikes.

2.4 Summary of Discrete Component EMI Considerations

EMI sources can be organized into component non-idealities, PCB geometry, switching devices, digital interfaces, cables and enclosures, and external disturbances. The categories interact; a fast MCU edge excites package inductance, the resulting ringing enters a cable through connector imbalance, and the cable then becomes the dominant antenna.

Real components are frequency-dependent networks. A value printed on a schematic is only the low-frequency parameter. Package geometry, internal construction, mounting pads, vias, and nearby planes add inductance, capacitance, and loss. Component selection therefore requires impedance-versus-frequency data, not nominal value alone.

Table 2-1. Summary of Discrete Component Parasitics and EMI Considerations

Component	Dominant Non-Idealities	EMI Consequence	Selection Question
Resistor	Lead or package inductance; terminal and body capacitance	Impedance rises, then potentially falls as capacitance bypasses R	Is the part non-inductive and effective at the concern frequency?
Capacitor	ESR, ESL, leakage, dielectric loss	Decoupling becomes inductive above self resonance	What is impedance at the actual noise frequency and mounting geometry?
Inductor	DCR, inter-winding capacitance, core loss, saturation	Filtering collapses above SRF or under bias	Does inductance remain adequate at DC bias and frequency?

2.5 Design Deficiencies

- **Poor power input configurations:** A supply trace has resistance and inductance. A transient load current produces both IR drop and inductive voltage. If the power-distribution network cannot deliver current locally, the rail disturbance spreads through shared impedance and modulates other circuits. The same event appears as power-integrity noise in the time domain and conducted/radiated EMI in the frequency domain. The resulting disturbance contains a resistive component proportional to current and an inductive component driven by how quickly that current changes. At fast edges, the inductive term can dominate. Shared impedance is especially damaging when noisy digital or motor current uses the same return neck as an ADC reference, oscillator, transceiver, or cable shield. The resulting voltage is not 'ground noise' in the abstract, but rather the voltage created by known current through a finite impedance.

Adding capacitors to the trace at both ends helps reduce the amplitude of the RF emissions while lowering the impedance of the trace reduces the voltage drop across the trace and lowers the amplitude. Consider Microstrip or Strip line techniques to control the impedance of a trace that carries clock signals to minimize radiation.

- **Poor Grounding:** Ground is a network, not an equipotential symbol. At RF, plane spreading inductance, vias, connectors, and chassis bonds create voltage differences. A solid reference plane usually improves both signal integrity and EMC because it supports a compact return path. Splitting the plane under a fast signal forces current around the split, increases loop area, and commonly creates common-mode voltage.
 - Partition circuits by placement and current flow before partitioning copper.
 - Keep noisy current local to its source and load; do not route the current through quiet reference regions.
 - Connect shield current to the intended chassis or boundary path before the current spreads into digital ground.

Improperly designed ground planes or long, high-impedance return paths are primary causes of both conducted and radiated EMI. The current in the PCB circuit must return to the source or power supply at some point. The ground can also exhibit similar characteristics as the supply positive traces and radiate unintentionally. Creating a good ground plane to capacitively couple the EMF from traces helps to prevent them from radiating.

- **Large Loop Areas:** Every signal forms a loop with its return path; larger loops behave as more efficient antennas, strengthening magnetic fields and emissions. For electrically small loops, magnetic dipole strength increases with loop area and current. Reducing area is therefore a first-order control for switching regulator input loops, gate loops, ESD discharge loops, motor commutation loops, and high-speed signal-return loops. A compact loop also has lower inductance, which reduces overshoot and the energy available to excite resonances.
- **PCB Traces:** A trace must be treated as a transmission line when propagation delay is no longer negligible compared with edge time. The exact threshold depends on acceptable error, but a common design screen is to compare one-way flight time with roughly one-sixth of rise time. Once distributed behavior matters, impedance mismatch creates reflections regardless of the clock frequency.

The polarity and size of the reflected wave depend on the relationship among load impedance, source impedance, and line impedance. Good termination controls that relationship and prevents repeated energy exchange along the interconnect.

Conductors that carry a current at frequencies at or near one quarter of a wavelength or an odd multiple of a quarter wavelength long that are close to the length of the conductor, that conductor can efficiently radiate electromagnetic energy as far field emissions. These conductors or traces when connected to a data bus, clock

lines or communication lines which are often long become transmission lines. When transient currents flow on these transmission lines.

Conductors that carry a current at frequencies at or near one quarter of a wavelength or an odd multiple of a quarter wavelength long that are close to the length of the conductor, that conductor can efficiently radiate electromagnetic energy as far field emissions. These conductors or traces when connected to a data bus, clock lines or communication lines which are often long become transmission lines. When transient currents flow on these transmission lines and traces, these currents act as an efficient antenna, radiating high-frequency electromagnetic waves. If such a transmission line is unterminated, the radiated spectrum shows maximum peak voltages. Impedance mismatching causes a maximum current flow on the transmission line or trace, thus maximizing radiated emissions.

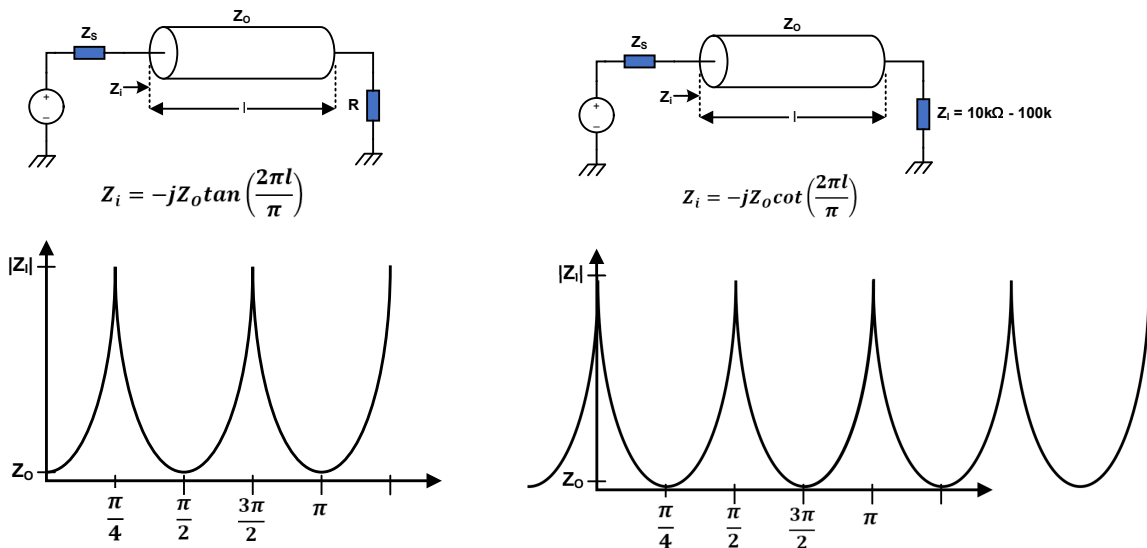


Figure 2-4. Transmission Line Model for Matched and Mismatched Termination Impedance

2.6 Sources Specific to MCUs

2.6.1 Clock Circuits and Oscillators

Clock circuits and oscillators are often the biggest generators of wide-band noise. High-frequency clock signals with fast rise and fall times create harmonic disturbances that can reach hundreds of MHz. Clock trees switch continuously and can synchronize noise across the system. Crystal pins are high-impedance analog nodes even though the pins generate a digital clock internally. Long traces, probing capacitance, or routing near aggressors can disturb oscillation and increase radiation. Keep the crystal loop compact, preserve the reference, and keep unrelated copper and fast signals away.

As the digital signals from an MCU rise and fall, they create current fluctuations and, if not properly accounted for, a dV/dT can show up as a radiated emission source.

A digital waveform contains a fundamental repetition frequency and harmonics created by its finite rise and fall times. The clock frequency predicts harmonic spacing; the edge rate predicts how far significant energy extends. A low-frequency clock with sub-nanosecond edges can therefore fail at hundreds of megahertz.

As rise time becomes shorter, meaningful spectral energy extends to higher frequencies. Edge rate therefore provides a better first estimate of EMC bandwidth than clock repetition rate alone.

Ringing adds a separate resonant frequency set by parasitic inductance and capacitance. Each edge excites the resonance; damping determines how long it persists. The measured spectrum can show narrow peaks at the ringing frequency and its mixing products, even if the logical data rate is much lower. Ringing frequency is approximately set by the effective inductance and capacitance of the driver, package, interconnect, receiver, and probe. The initial overshoot can forward-bias protection structures or create substrate current. Repeated ringing increases spectral content and can push energy into a cable or enclosure resonance. Ringing frequency rises

when effective inductance or capacitance is reduced. Damping controls how quickly the oscillation decays after each transition.

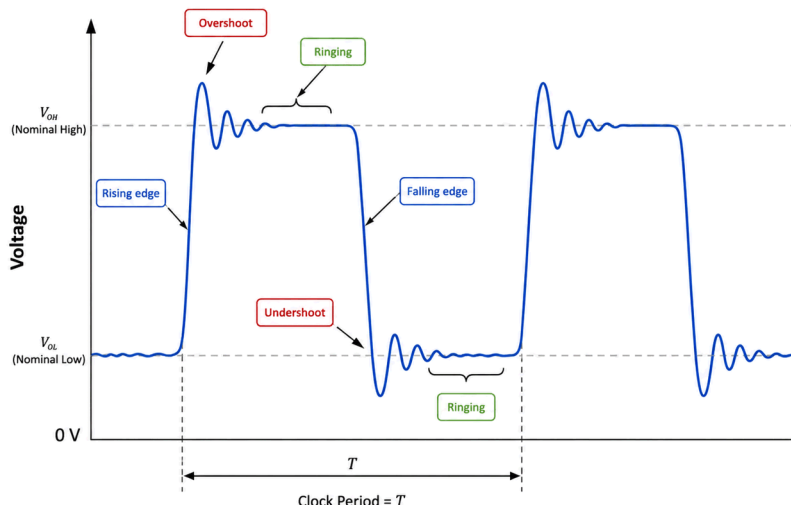


Figure 2-5. Digital Signal Ringing Components and Parasitic Ringing

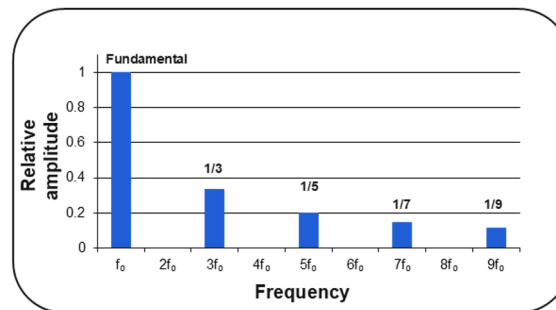


Figure 2-6. Emission Frequency and Relative Amplitude for Digital Signals

There can typically be 11 frequency components associated with a square wave or digital clock signal with the ringing that occurs during the shift from high to low and vice versa:

1. Period of the on time
2. Rise time
3. Fall time of first ringing element
4. Period of the second ring
5. Total period of square wave
6. Fall time
7. Period of the off time
8. Period of second ring
9. Rise and fall of each ringing element
10. Decaying frequency element of ringing
11. Decaying rise and fall times of each ringing element

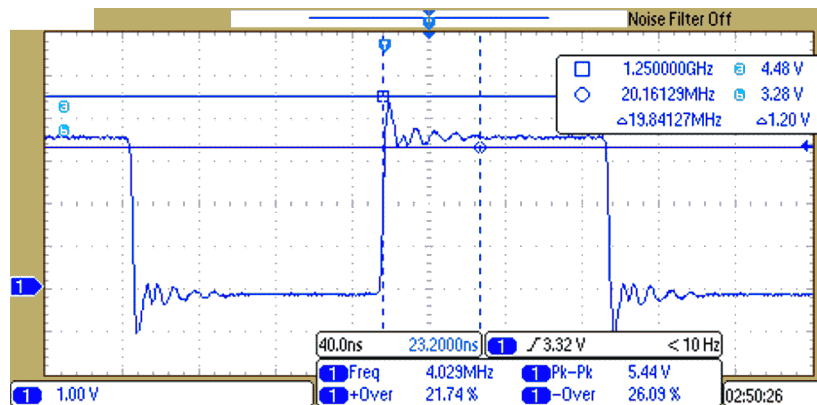


Figure 2-7. Clock Output With Higher Frequency Ringing Components

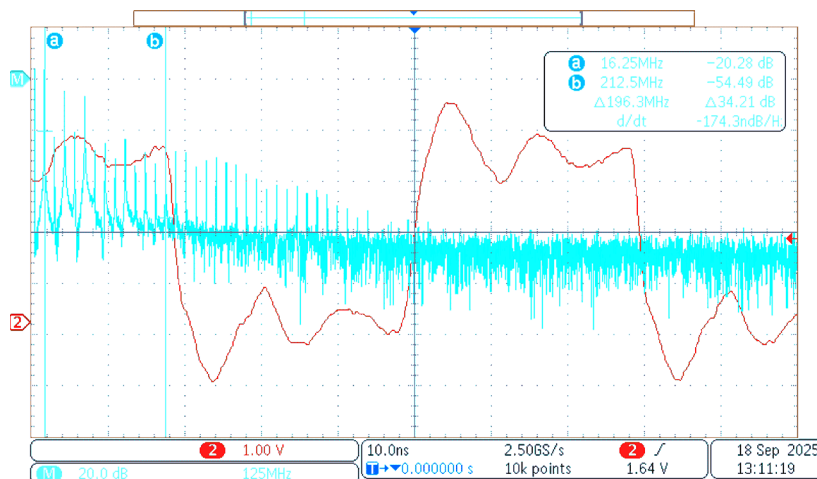


Figure 2-8. Digital Clock Signal with Superimposed Frequency Domain Peaks

2.7 Other Common Sources

2.7.1 High-Speed Digital Buses

External memory interfaces (like SDRAM) and data or address buses involve continuous transitions on multiple lines, which act as significant noise sources. Parallel buses produce simultaneous-switching output current. Package and plane inductance convert that current step into rail bounce. Byte-lane timing skew, drive strength, on-die termination, and return-path continuity all influence both data eye quality and emissions. Serial buses reduce pin count but move energy to higher frequencies, where connector and via asymmetry become more important.

2.7.2 I/O Output Transitions

Rapid switching of GPIO pins creates current spikes. Shorter transition times (faster edges) result in a richer frequency spectrum of noise. Unused performance margin is often an EMI liability. Maximum drive strength and fastest slew are rarely necessary for short local nets. Series damping, lower drive, shorter routes, and tightly referenced return paths reduce the energy launched by each transition.

2.7.3 Floating Components

Unconnected metal objects like heat sinks, brackets, or unused IC pins can act as antennas, radiating interference if not properly grounded. A heat sink, shield can, mounting bracket, cable drain, or unused copper island can acquire RF voltage through parasitic capacitance. If an unused pin lacks a low-impedance RF bond, the pin can resonate and re-radiate. A DC continuity check is not sufficient: a long wire bond can be a good DC ground and a poor RF ground. Use short, wide bonds or multiple attachment points when a structure is intended to remain at chassis potential.

2.7.4 External Environmental Sources

External Environmental Sources: External electromagnetic energy becomes a problem when it couples into a susceptible port and is converted into a functional disturbance. Motors and relays generate inductive transients; radio transmitters produce continuous RF fields; electrostatic discharge injects fast current; and trace-to-trace crosstalk couples energy locally. Susceptibility depends on bandwidth, nonlinear junctions that demodulate RF, protection paths, firmware recovery, and the physical route of the injected current.

- **Electromechanical Devices:** High-current loads such as motors, pumps, and relays generate significant transient noise and magnetic field coupling.
- **Communication Transmitters:** Nearby cellular phones, Wi-Fi routers, and radio stations (TV, HAM, and so forth) emit RF energy that can be picked up by unshielded MCU traces.
- **Crosstalk:** Closely spaced PCB traces can interfere with each other through inductive or capacitive coupling, where one signal (the "aggressor") disrupts another (the "victim").
- **Natural Phenomena:** Occasional but intense disturbances from lightning strikes, electrostatic discharge (ESD), or solar flares can overwhelm protection systems.

Table 2-2. Environmental EMI Troubleshooting

Disturbance	Primary Coupling Path	Typical Symptom	First Control
Motor or relay transient	Shared supply or ground; magnetic loop	Reset, ADC error, interface fault	Local suppression and separated current loops
Nearby RF transmitter	Cable or trace common mode; enclosure aperture	Demodulated noise or intermittent upset	Boundary filtering, shielding, balanced routing
Crosstalk	Mutual capacitance and inductance	False edge, timing jitter, analog error	Spacing, reference continuity, reduced edge rate
ESD type event	Direct or capacitive injection; chassis current	Latch-up, reset, permanent damage	Short protection path to boundary or chassis

3 Mitigation Techniques to Minimize EMI and Optimize Board Performance

Effective mitigation follows the current path. First reduce source energy, then contain the local loop, preserve the return path, filter at boundaries, and prevent large structures from carrying common-mode current. Shielding and ferrites are useful, but they cannot fully compensate for a source loop or return discontinuity that remains physically large.

One way to mitigate or greatly reduce the emissions that are a result of this digital signal ringing is to analyze the capacitors attached to the VDD line to see the impedance characteristics over frequency. The frequency here is the key word. Design deficiencies are typically easily resolvable and can be mitigated by understanding the potential sources of EMI of a design and implementing changes to reduce or remove emissions. In the following sections, we focus on some of the most common PCB design issues and mitigation techniques to optimize the design.

3.1 Power Supply Capacitor Selection and Placement

Power integrity and EMI are the same problem viewed from different instruments. A power rail that bounces at high frequency forces return current through unexpected paths and can modulate I/O thresholds, ADC references, oscillator edges, and cable shields. Decoupling is the local energy path that keeps transient current from spreading across the board.

One way to mitigate or greatly reduce the emissions that are a result of this digital signal ringing is to analyze the capacitors attached to the VDD line to see the impedance characteristics over frequency. The frequency here is the key word. Design deficiencies are typically easily resolvable and can be mitigate

When selecting capacitors (following device recommended values), one key parameter to consider is the ESR of the capacitor. As you can see from the impedance versus frequency plot below, the impedance has a low point where the impedance is the smallest over a given frequency, which is preferred since we want to minimize any voltage drop caused by a high impedance value. Select decoupling by mounted impedance across the disturbance band. ESR creates damping and sets minimum impedance; ESL sets the high-frequency limit. Package size, orientation, pad length, via count, and distance to the IC can matter as much as nominal capacitance.

- Use device-recommended capacitance as the starting point, then evaluate impedance at frequencies implied by edge rate and measured ringing.
- Do not assume a three-value capacitor stack is always superior. Check for anti-resonance peaks created by the combined network.
- Account for DC-bias derating, temperature, tolerance, and dielectric aging when estimating effective capacitance.

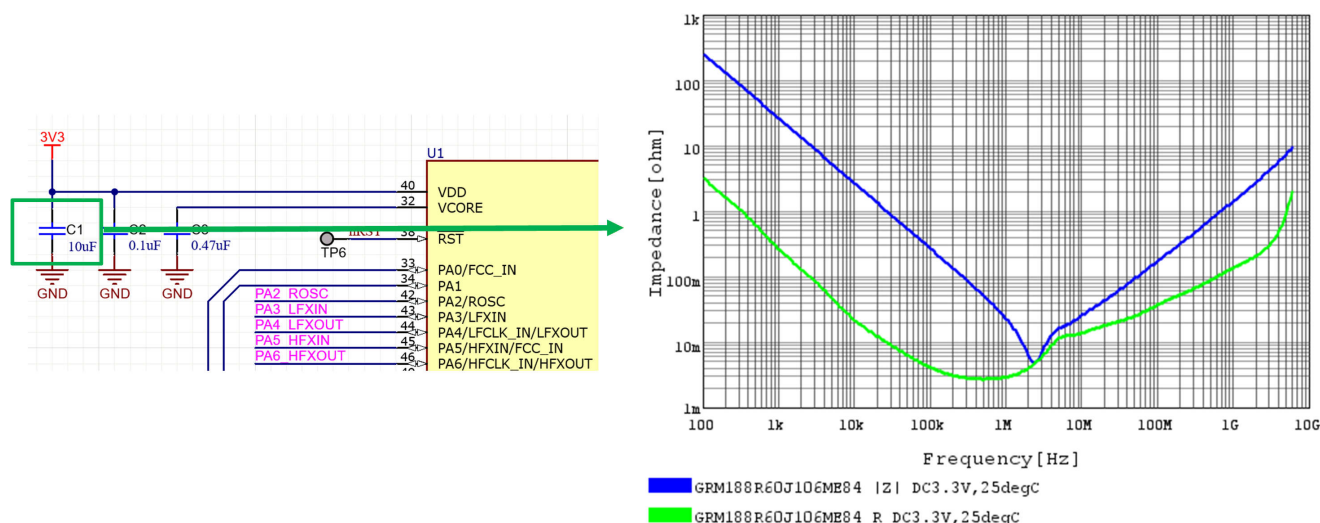


Figure 3-1. 10µF Supply Capacitor ESR Impedance Over Frequency

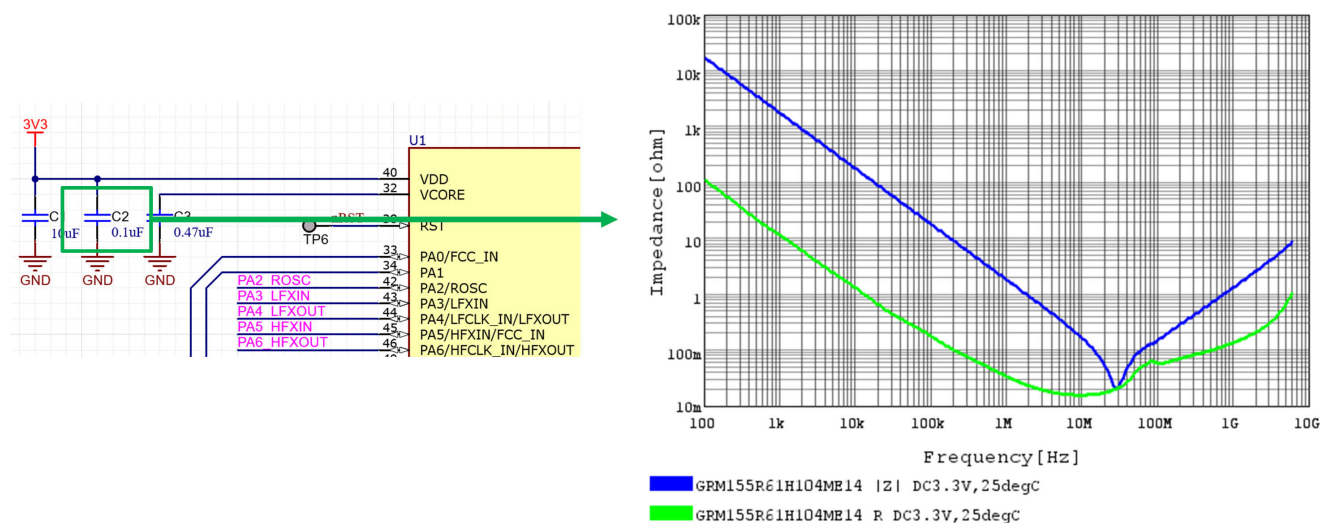


Figure 3-2. 0.1µF Supply Capacitor ESR Impedance Over Frequency

A decoupling capacitor does not 'absorb noise' in isolation. The capacitor provides a short local current path. The EMC value of the decoupling capacitor depends on the complete loop from IC power pin, through the capacitor, and back to the IC ground pin.

3.2 Capacitor Placement

Another often overlooked optimization factor is the placement of the capacitors. The preferred placement shown in Figure 3-3 has three capacitors with the lowest value capacitor as close to the device pin as possible to reduce parasitic resistance and filter out higher frequency noise.

The ground connection for each capacitor is connected with a via to the ground plane layer with very short traces from the pad of the capacitor and the via. This is done to minimize the loop inductance and suppress noise that can be a result of larger loops which can function as an EM radiator. Target impedance turns a transient requirement into a design metric. If the load current changes by ΔI and the allowed rail deviation is ΔV , the power-distribution network must remain below $\Delta V/\Delta I$ over the relevant bandwidth.

Target impedance is obtained by dividing the allowable rail-voltage deviation by the expected load-current step. The PDN must stay below that impedance across the bandwidth of the transient.

The highest-frequency capacitor must be closest to the supported pins with short, wide connections and adjacent power and ground vias. Bulk capacitance supports slower energy demand; local small-package capacitance supports fast edge current. Embedded package capacitance and tightly spaced power and ground planes extend the response to still higher frequency.

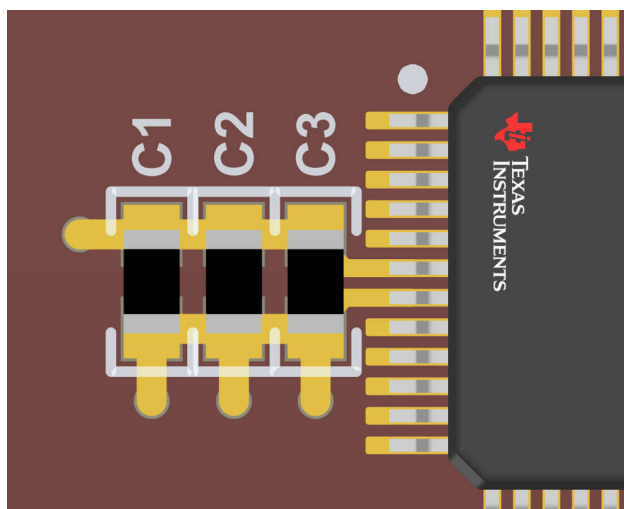


Figure 3-3. VIN Capacitor Placement Example Layout

Table 3-1. Capacitor Placement Considerations and EMI Outcomes

Placement Choice	Electrical Effect	EMI Outcome
Capacitor beside pins with adjacent vias	Small loop and low mounted ESL	Local transient current; less plane excitation
Capacitor connected through a long neck	Added inductance isolates capacitor at high frequency	Rail bounce spreads and excites common-mode current
Shared narrow return for noisy and quiet loads	Shared impedance converts current to voltage	Reference modulation and cross-domain coupling
Bulk capacitor only	Good low-frequency energy, poor edge response	Fast current still travels across board

Below is a summary of decoupling layout rules:

- Place the smallest-value, highest-frequency capacitor closest to the IC power and ground pins it supports.
- Give each high-speed IC local power and ground vias. Avoid routing through a long neck before reaching the planes.
- Place ground vias directly beside capacitor ground pads, and power vias directly beside capacitor power pads when the capacitor serves plane-to-pin current.
- Use multiple capacitor values only when their placement and parasitics are controlled. A poorly placed capacitor can be irrelevant above a few tens of MHz.
- Do not share a narrow ground return between a noisy load and a sensitive analog reference. Shared impedance turns current into voltage noise.

Bulk capacitors support lower-frequency load changes and prevent rail droop, but do not replace tight high-frequency bypassing. At the package and via scale, ESL often dominates. This is why a nearby 100nF capacitor can outperform a larger capacitor placed a few centimeters away for edge current.

Switching power stages deserve special treatment because they combine fast voltage edges, high current, magnetic fields, and nonlinear devices. Most regulator EMI problems originate from a few copper shapes: the switch node, the hot input loop, the gate-drive loop, and the diode/synchronous rectifier loop. Via stitching helps contain fields, but cannot compensate for a large switch loop or a switch node poured like a decorative logo.

Switching power stages combine high dv/dt , high di/dt , nonlinear devices, and magnetic components, so it is important to identify the commutation loops for each switch state. The highest priority loop is often the input ceramic capacitor through the switching devices and back. The switch node is a high- dv/dt electric-field source; the input and rectifier loops are high- di/dt magnetic-field sources.

- Place the input ceramic capacitor directly across the power-stage supply and return pins.
- Keep switch-node copper no larger than required for current and thermal performance.
- Route feedback and current sense away from the switch node and inductor; use quiet Kelvin sensing where appropriate.
- Keep motor current and recirculation paths out of logic and ADC returns.
- Use snubbers or gate-resistance tuning only after identifying the resonant elements and verifying device stress.
- Minimize the high- di/dt input loop from input capacitor to switch device and back. Put the input capacitor directly at the power stage pins.
- Keep switch-node copper as small as thermal and current requirements allow. Do not run quiet traces under or beside the switch node.
- Use a ground via wall around the converter region when it helps connect local ground pour to the reference plane and contain fields.
- Route feedback away from the switch node and inductor. Sense the output at the intended regulation point, not through a noisy power path.
- For motor drivers, separate power-current loops from control and sensing loops. Stitch the power-ground region well, but avoid injecting motor current into logic/ADC returns.

3.3 Layer Stack-Up Optimization

Layer stack-up can play an important role in the EMI performance of a PCB. One important consideration is which layer high speed signals are routed on and where this layer is in relation to a ground layer. By routing high speed signals on planes immediately adjacent to a solid ground plane, the return path is shortened and localizes magnetic fields.

A good stack-up places every fast signal layer adjacent to a continuous reference plane. Thin dielectric spacing tightens field coupling, lowers loop inductance, and reduces the field that reaches the board edge. A six-layer board offers more opportunities for close signal-to-ground coupling than a four-layer board, but layer count alone does not guarantee EMC: plane continuity and actual routing are decisive.

Table 3-2. Optimal Stack-Up Traits and Considerations

Trait	Why it helps	Failure mode if absent
Continuous ground reference	Provides compact high-frequency return	Detours, loop expansion, common-mode conversion
Thin signal-to-plane dielectric	Tighter fields and lower loop inductance	More fringe field and stronger coupling to other structures
Power plane adjacent to ground	Adds distributed plane capacitance	Higher PDN impedance and broader current spread
No fast routing over voids	Maintains local return path	Reflections, crosstalk, and radiation

CAUTION

Plane splitting is often overused. For mixed-signal boards, physical partitioning and disciplined current routing over one solid ground plane are frequently safer than cutting the reference plane beneath fast signals.

In general, any signal and the signal return path form a loop. As this loop increases in size, the resulting radiated emissions become stronger as well. This is why minimization of these loops becomes important for EMI sensitive

designs. 6-layer boards are often preferred when designing for EMI optimization as this allows a return reference ground plane to be located below both the top and bottom layers of the PCB, reducing the loop significantly, not only through proximity to the outer layer, but also due to the plane being closer to the outer layer due to the additional internal layers. This continuous ground plane provides a direct and low inductance return path, keeping the fields contained between the top layer and the second layer. Figure 3-4 shows a good 6-layer board stack up for EMI conscious designs. This makes sure that each signal plane always has an adjacent GND plane.

	Top Overlay
	Top Solder
1	Top Layer
	Dielectric 1
2	Ground Return Plane 1
	Dielectric 2
3	Signals and Power 1
	Dielectric3
4	Signals and Power 2
	Dielectric 4
5	Ground Return Plane 2
	Dielectric 5
6	Bottom Layer
	Bottom Solder
	Bottom Overlay

Figure 3-4. Example 6-Layer PCB Stack-Up

A practical four-layer arrangement commonly uses outer signal/component layers with internal ground and power. Since there is no option to have an adjacent GND plane to each signal layer (assuming both internal layers are not a solid GND plane), add ground pours to the top layer and bottom layer with adequate via stitching. Make sure that the via stitching is dense on these ground pours in order to mitigate a poorly connected plane which can result in even more unwanted EMI as it acts as an antenna in this case. The ground plane and power plane are the internal layers, so on the layer without an adjacent ground plane, using guard traces and maintaining tightly coupled signal/return pairs is recommended to reduce potential EMI problems. The outer layer farther from ground needs careful ground pour, stitching, and routing discipline. In either case, do not route a fast signal where its nearest plane is broken by an anti-pad field, slot, or keepout.

- Prefer high-speed microstrip over the closest solid ground plane or stripline between solid references.
- Use ground pours only when they are well connected; floating or weakly stitched copper can resonate.
- Treat connector regions as boundary structures with explicit return and shield-current paths.

	Top Overlay
	Top Solder
1	Top Layer
	Dielectric 1
2	Ground Return Plane
	Dielectric2
3	Signal And Power
	Dielectric 3
4	Bottom Layer
	Bottom Solder
	Bottom Overlay

Figure 3-5. Example 4-Layer PCB Stack-Up

A grounded guard trace can reduce electric-field coupling and provide a nearby return path, but only if it is connected to a low-impedance reference with sufficiently frequent vias. Otherwise it can become a resonant conductor. A via fence connects reference copper layers and reduces fringe-field leakage at board edges, noisy zones, RF structures, apertures, and connector boundaries.

A common starting guideline is to keep stitching-via pitch much smaller than the guided wavelength at the highest frequency of concern. Practical spacing must also reflect source geometry and the continuity required at the boundary.

The wavelength rule is a starting point, not a universal optimum. The fence must be close to the field source or boundary, tied to a quiet reference, and free of long gaps. Dense vias far from the actual current path add cost without repairing a return discontinuity in the board interior.

To integrate guard traces for 4-layer boards where there are high speed critical signals, first make sure the signal trace is as short as possible. Placing grounded copper traces on both sides of a high-speed or sensitive analog trace "captures" the radiated electric and magnetic fields, minimizing crosstalk with adjacent traces. They are also able to provide a low-impedance path for return currents, which is vital for preventing noisy signals from interfering with the rest of the board. For the best shielding, stitching vias must be spaced at intervals no greater than $\lambda / 20$ of the wavelength of the highest-frequency signal being protected to make sure this "fencing" acts as an effective EMI barrier.

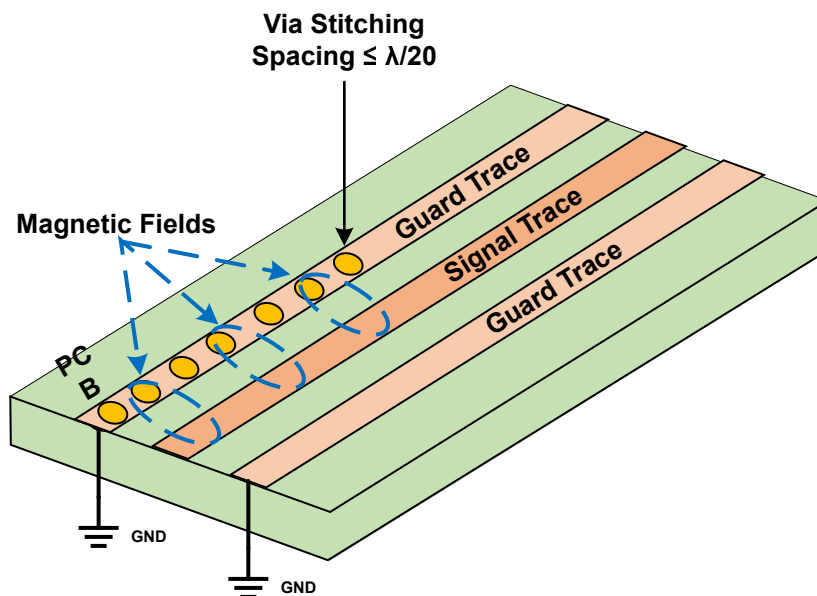


Figure 3-6. Via Stitching and Guard Trace Implementation

A via fence is a row or ring of ground vias used to connect reference copper layers and reduce the leakage of fringe fields. Via fences are useful at board edges, around noisy switching zones, around RF or oscillator regions, alongside high-speed channels, and around connector boundaries. Via fences must be treated as a field-management tool, not as a universal cure for every EMI problem.

Where fences are most useful:

- Along board edges near high-speed traces, especially when traces approach an edge or connector.
- Around switch-mode power supplies, class-D outputs, motor drivers, high-current pulses, and crystal/clock regions.
- Around RF traces or antennas where the fence is part of a defined coplanar waveguide or shield boundary.
- At chassis-to-digital-ground transition zones, when the grounding concept intentionally separates shield current from quiet circuit ground.
- Around internal cutouts, slots, mounting holes, and mechanical keepouts that interrupt return current.
- A fence is less useful if it is far from the field source, if it is broken by long gaps, if it is tied to a noisy copper island, or if it forces the current return to cross an unintended path. Dense via fences can also create manufacturing costs, routing congestion, and plane perforation. Use them where they close a real RF path.

3.3.1 Good Stack-Up Traits

- At least one uninterrupted ground plane for four or more layers.
- High-speed outer-layer microstrips referenced to the nearest solid ground plane.
- High-speed inner-layer striplines sandwiched between reference planes when emissions must be minimized.
- Thin dielectric between signal and reference plane for tighter field coupling and lower loop inductance.
- Power planes adjacent to ground planes so plane capacitance and decoupling work together.
- No routing over ground-plane slots, large voids, anti-pad fields, or split boundaries.

Plane splitting is often overused. Separating analog and digital copper can make sense for low-level measurement regions, but cutting the reference under a fast signal is typically worse than sharing a continuous plane with disciplined placement. The safer pattern is often a single solid ground plane, physical partitioning of noisy and quiet circuits, and controlled single-point or boundary treatment only where current paths are understood.

Another stack up and layer setup method to remove “fringing fields” is often referred to as the *20-H Rule*. The 20-H rule proposes recessing a power plane from a ground-plane edge by roughly twenty times the dielectric separation H to reduce edge fringing. The rule can be a useful geometric heuristic when plane-pair edge radiation is a known contributor. The 20-H rule is not a substitute for a close plane pair, local decoupling, continuous references, or control of cable common-mode current. Modern boards must validate the benefit against stack-up, power distribution, and manufacturing constraints rather than apply the rule automatically.

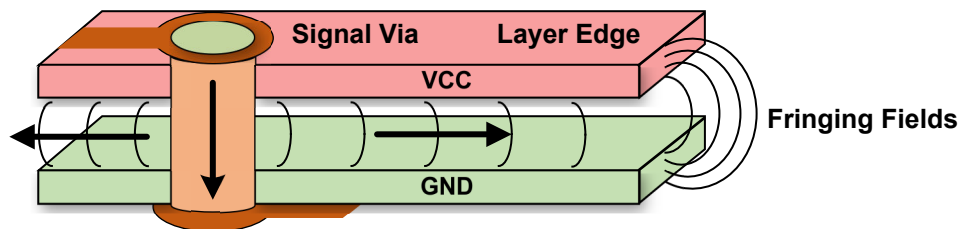


Figure 3-7. Example of Fringe Electromagnetic Emission Fields

3.3.2 Stitching and Shielding Optimization Techniques

When a signal changes layers, its return current must also change physical location. An adjacent ground via gives return current a short vertical path between ground planes. Without it, the current may travel to a remote plane bond or decoupling capacitor, creating a large three-dimensional loop. That detour increases loop area and can create mode conversion, crosstalk, and radiation.

Practical placement guidance:

- Place at least one ground stitching via adjacent to each high-speed signal via when the signal changes layers. For differential pairs, place ground vias symmetrically so the pair does not see an unbalanced transition.
- If the signal changes from a layer referenced to one ground plane to a layer referenced to another ground plane, add ground vias connecting those planes near the signal via.
- If the signal changes reference from ground to power, provide a high-frequency return bridge with a local decoupling capacitor between the two planes, placed close to the transition. Prefer avoiding this if possible.
- For clock, reset, fast GPIO, SPI, SerDes, USB, Ethernet, LVDS, MIPI, and RF nets, treat every via transition as an RF structure, not as a simple topological connection.
- Avoid via transitions over plane voids, near plane edges, or beside large anti-pad clearances that remove the local reference copper.
- A compact signal-via plus ground-via cluster behaves more like a controlled vertical interconnect. At very high data rates, the complete geometry matters: signal barrel diameter, ground via distance, anti-pad shape, backdrill depth, pad stack, plane spacing, and symmetry all influence impedance and resonance.
- Place a ground via near a high-speed signal via when both layers reference ground planes.
- For differential pairs, keep signal and return-via geometry symmetric to limit common-mode conversion.
- If the reference changes between power and ground, add a local high-frequency plane bridge with a capacitor, or preferably avoid the reference change.

- At very high data rates, include anti-pad shape, via stub, barrel diameter, plane spacing, and backdrill in the interconnect model.

Table 3-3. Via Implementations and Cautions

Use case	Initial pitch / distance	Why	Cautions
Signal via return stitching	1mm to 3mm from the signal via when practical	Keeps the return current transition local	Respect impedance, anti-pads, pair symmetry, and fabricator clearance
Differential pair layer transition	Two ground vias or symmetric via pair near transition	Reduces imbalance and common-mode conversion	Avoid asymmetric anti-pad removal near only one side of the pair
Board-edge ground fence	$\lambda / 20$ at highest concern; often 2mm to 5mm for general digital boards	Reduces fringing and edge radiation	Do not assume it fixes a bad return path in the interior
Noisy converter boundary	2mm to 5mm pitch around the hot region as a starting point	Contains fields and ties local pours to the main reference	Keep switch node copper inside the boundary and away from fence gaps
Connector shield/chassis boundary	As tight as mechanically reasonable near connector shell and ESD return	Prevents cable current from spreading into logic ground	Requires a defined chassis/digital ground strategy
Large quiet copper pours	10mm to 20mm grid for low-frequency equipotential support	Prevents floating copper and lowers plane impedance	Do not create isolated copper slivers or acid traps

3.4 Trace Routing

Spacing reduces capacitive and inductive crosstalk. The common '3W' rule - spacing traces by three times the width - is a useful screen, but stack-up and parallel run length determine actual coupling. Shortening the parallel region is often more effective than applying a fixed spacing everywhere.

Reference continuity is typically more important than corner style. A 90-degree bend creates a small local geometry change, but on ordinary digital boards, the added capacitance is often minor compared with a plane split, connector discontinuity, long stub, or poor termination. Use 45-degree or curved routing for controlled geometry and manufacturability, but do not treat corner shape as the primary EMC control.

- Trace Separation: Separate analog, digital, and high-power sub-circuits. Keep parallel traces spaced at least three times the trace width to prevent crosstalk.
- Layer Transitions: If you must change layers on a high-speed trace, place stitching vias next to the signal via to provide a continuous, low impedance return path
- Avoid 90° Corners: Route traces with 45° angles or curved corners, as sharp 90° bends can cause impedance discontinuities and radiate EMI

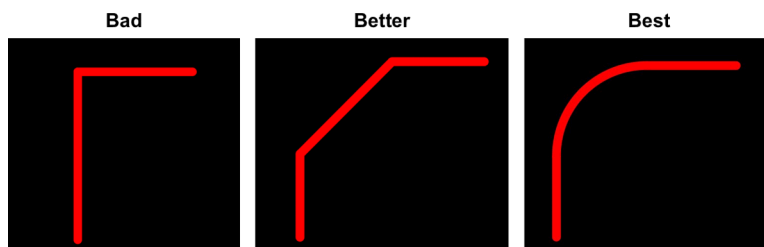


Figure 3-8. Trace Routing Styles

Preserve the return plane, control impedance, eliminate stubs, reduce edge rate, limit parallel coupling, and then refine corner geometry.

3.5 Passive Pi Filtering

Another method for reducing noise and EM radiation sources is to add a passive Pi filter. A Pi filter allows emission reduction for noise coming from both external as well as internal sources. [Figure 3-9](#) shows an input signal with high-frequency noise superimposed to mimic a VIN with an emission source.

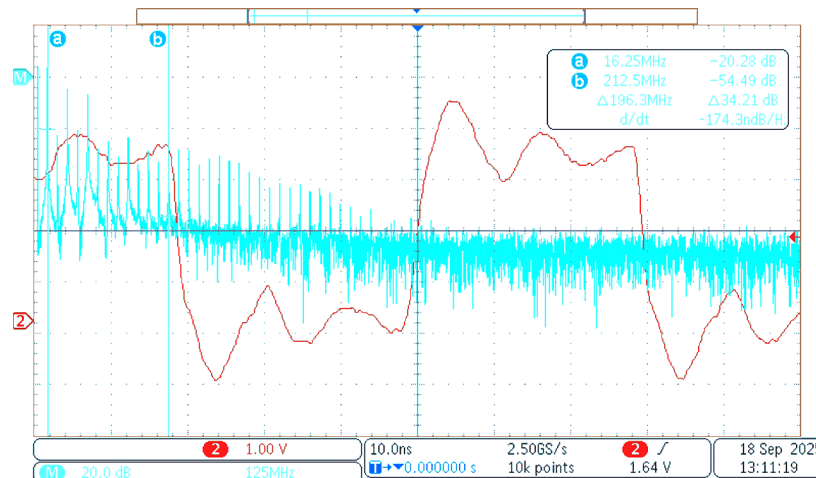


Figure 3-9. Clock Signal with Ringing and FFT Plot

A C-L-C pi filter presents shunt capacitance on both sides of a series impedance. The filter can attenuate noise traveling into or out of a subsystem, but the performance of the filter depends on source and load impedance, component parasitics, DC bias, and damping. An ideal lossless network can develop a high-Q resonance that amplifies noise near cutoff.

The filter has a natural resonant region determined by its effective inductance and capacitance. Source and load impedances, losses, and parasitics determine whether the response is well damped or exhibits peaking.

- Choose the inductor or ferrite for current rating, saturation margin, DCR, and impedance at the disturbance frequency.
- Place the load-side high-frequency capacitor close to the protected device and minimize the return loop.
- Add damping through ESR, a lossy ferrite, or an explicit damping network when the source/load combination produces peaking.
- Verify regulator stability. An input filter can interact with a converter's negative incremental input impedance.
- Measure insertion loss in the actual impedance environment; a 50 Ω bench result can not predict a low-impedance power rail.

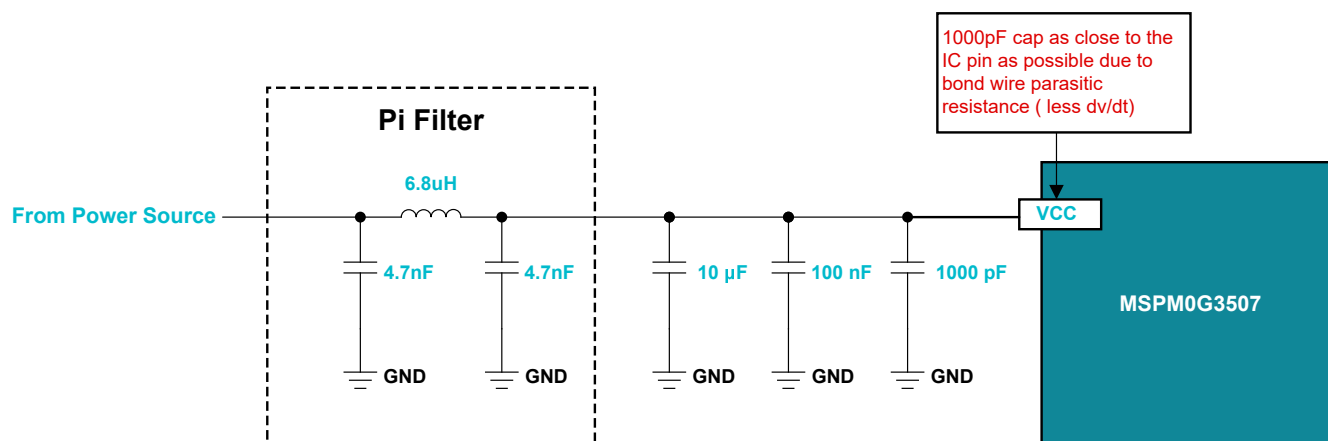


Figure 3-10. Bi-Directional EMI Filter Using a Pi Filter for High Frequency Noise

The typical passive pi filter shown in Figure 3-11 consists of two capacitors and an inductor connected between them and in series with the main power supply line. This is added in addition to the normal supply capacitors in between the existing capacitors and the power supply. The operating principle can be partitioned into:

1. The Inductor Acts as a Current Source Opposing the Drop: When the MCU suddenly demands more current, the supply voltage at the input of the inductor momentarily dips, causing a negative di/dt . The collapsing magnetic field induces a forward voltage ($V = L \times di/dt$) across the inductor. As a result, this induced voltage acts in series with the power supply, pushing stored current forward into the output side of the filter.

2. **Teamwork with the Output Capacitor Immediate Response:** The output capacitor responds first, instantly dumping its stored charge into the MCU to cover the microsecond-level spike. **Sustained Response:** As the capacitor discharges, the collapsing magnetic field of the inductor sustains the current flow, recharging the output capacitor and stabilizing the rail. The inductor cannot do this indefinitely; it only supplies current until its stored magnetic energy is completely depleted.
3. Without this action, quick digital switching inside the MCU can cause the supply voltage to drop below the minimum operating threshold. By integrating this type of filter, the filter essentially bridges the tiny time gap between the peak demand of the MCU and the ability of the main power supply to react, preventing accidental processor resets.

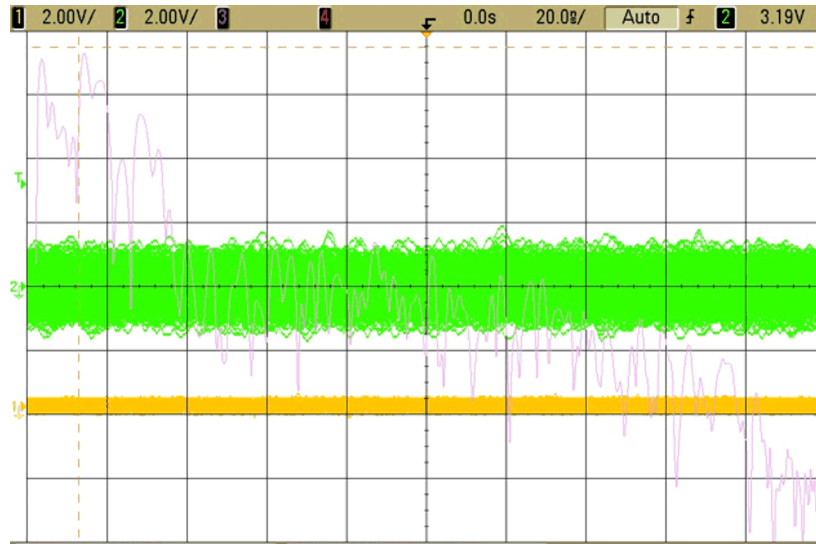


Figure 3-11. Pi-Filter Noise Filtering Performance

3.6 Shielding, Grounding, and Cable Boundaries

Shielding works by reflection, absorption, and redirection of current. At high frequency, enclosure seams and apertures often dominate material conductivity. A long narrow slot can radiate efficiently because current must detour around it. Bond panels with many short contacts, control aperture dimensions, and route shield current at the connector boundary rather than through circuit ground.

At higher frequency, current crowds nearer a conductor's surface. Higher conductivity and magnetic permeability reduce the penetration depth, but seams and apertures can still dominate enclosure leakage.

Cable filters must be located at the connector so disturbed current is diverted before it enters the board. Common-mode chokes act on common-mode current while allowing differential current to pass; their usefulness depends on mode-conversion sources and parasitic capacitance. Connector pin assignment must place returns beside fast signals and avoid forcing high-frequency current across the connector body.

3.7 Transient and ESD Current Paths

Protection is most effective when the transient is intercepted at the boundary and given a short path to the intended reference or chassis. A protection device placed far from the connector leaves a long unprotected trace that can radiate and couple into neighboring circuits. The clamping voltage at the IC equals the protector's dynamic clamp plus voltage across path inductance and resistance.

The voltage that reaches the IC includes the dynamic clamp of the protector, plus additional voltage created by path inductance and resistance. A long protection route can therefore defeat an otherwise suitable suppressor.

- Place transient suppression close to the entry point.
- Keep the discharge path short, wide, and separate from sensitive return paths.
- Coordinate protection capacitance and leakage with signal bandwidth and precision requirements.
- Design firmware to detect, log, and recover from non-destructive upset where appropriate.

4 EMC Verification and Troubleshooting

A design is not complete when the layout looks correct; a design is complete when measurements confirm acceptable margin. Pre-compliance work must identify dominant frequencies, current paths, operating modes, and cable configurations before formal testing.

4.1 Emissions and Immunity as Port Problems

Emissions testing asks how energy leaves the equipment through power, signal, enclosure, and cable ports. Immunity testing asks how energy entering those ports affects function. The same transfer path often works in both directions, so an emission hotspot can reveal a susceptibility path.

Table 4-1. General EMI-Sensitive Design Troubleshooting

Question	What to observe	Likely next action
Which frequency fails?	Narrow peak, harmonic comb, or broadband rise	Relate frequency to clock, edge, converter, or resonance
Which cable/configuration matters?	Large change when cable is moved or removed	Measure common-mode current and inspect connector return
Which operating mode matters?	Failure tracks load, bus activity, PWM, or radio state	Isolate the switching event and its current loop
Where is the near-field hotspot?	Localized E-field node or H-field loop	Reduce node area or loop area; verify return path

4.2 Pre-Compliance Measurement Toolkit

1. Oscilloscope with appropriate probing: confirm rail bounce, ringing frequency, edge rate, and timing. Use a short ground spring or differential probe; a long probe ground can create false ringing
2. Spectrum analyzer or FFT receiver: identify spectral peaks and compare changes after controlled modifications
3. Near-field E- and H-probes: localize high-dv/dt nodes and high-di/dt loops without claiming absolute compliance levels
4. RF current probe: identify common-mode current on cables, harnesses, and shield bonds.
5. Impedance or network measurement: characterize filters, PDN resonances, and component behavior in the relevant frequency range

Change one variable at a time and record the operating mode, cable placement, load, bandwidth, detector settings, and probe position. A repeatable relative measurement is more useful than an absolute number from an uncontrolled setup.

4.3 A Frequency-to-Structure Debug Method

1. Reproduce the failure with a stable operating mode and test configuration.
2. Identify the dominant frequency, bandwidth, and whether the signature is narrowband or broadband.
3. Map the frequency to candidate sources: clock harmonic, converter switching edge, ringing resonance, bus burst, or external injection.
4. Locate the source with time correlation and near-field probing; then locate the transfer path with cable-current and return-path checks.
5. Apply a reversible experiment such as added damping, temporary shielding, a local capacitor, reduced slew rate, or a short RF bond.
6. Confirm the mechanism by predicting and observing how the frequency or amplitude changes.
7. Implement the smallest robust design correction and retest across modes, temperature, load, cable position, and manufacturing variation.

4.4 Interpreting Common Spectral Signatures

Table 4-2. Spectral Signatures and Potential Sources

Signature	Common interpretation	Checks
Harmonic comb	Periodic clock or PWM source	Fundamental spacing, edge rate, clock enable state

Table 4-2. Spectral Signatures and Potential Sources (continued)

Signature	Common interpretation	Checks
Broadband floor increase	Fast asynchronous switching, arcing, or measurement overload	Time-domain correlation and analyzer input level
Single strong narrow peak	Resonance, oscillator, or cable/enclosure mode	Move cable, add damping, inspect resonant length
Peak shifts with load	Converter mode or LC resonance	Switch frequency, inductor bias, capacitance, control mode
Peak changes with probe cable	Test setup or common-mode path	Improve setup bonding and measure cable current

4.5 Integrated Design Review Checklist

Use this checklist at schematic review, placement review, routing review, and pre-compliance readiness review. A 'yes' answer must be supported by a current-path explanation or measurement, not only a visual impression.

Area	Review Question
Sources	Are unnecessary slew rate, drive strength, and switching-node area minimized?
Returns	Does every fast signal, power transient, and protection event have a short intended return path?
Planes	Are reference planes continuous beneath fast routes and across layer transitions?
PDN	Does mounted decoupling impedance meet the rail target over the needed bandwidth?
Converters	Are hot loops compact and feedback/sense routes isolated from switch nodes?
Transitions	Are return stitching vias or local plane bridges present at high-speed layer changes?
Boundaries	Are connector filters and transient paths located at the entry point?
Cables	Are differential balance and common-mode current controlled before cables leave the enclosure?
Mechanical	Are shields, heat sinks, and panels bonded with low RF impedance where intended?
Validation	Have worst-case modes, loads, cable placements, and recovery behavior been tested?

5 Conclusion

This paper examines electromagnetic interference (EMI) in MCU-based electronic systems, covering its physical origins, on-board and external sources, and practical mitigation techniques. EMI is not an isolated noise phenomenon; it is the visible result of time-varying current flowing through real geometry. Component parasitics set resonances, PCB layout determines loop and return-path geometry, cables and enclosures provide large radiating structures, and filters or shields redirect energy only when they are installed in the correct current path.

Any changing voltage on a conductor generates an electromagnetic field, and when conductor length approaches one-quarter the signal wavelength, near-field energy transitions to radiating far-field emissions. On-board sources fall into three categories: component-level parasitics (ESL and shunt capacitance in resistors, ESR and ESL in capacitors, DCR and inter-winding capacitance in inductors),

PCB design deficiencies (high-impedance power traces, inadequate ground planes, large signal-return loop areas, and impedance-mismatched transmission lines), and MCU-specific mechanisms (clock harmonics, high-speed bus transitions, fast GPIO switching, and floating conductive structures). External sources include electromechanical loads, nearby RF transmitters, trace-to-trace crosstalk, and transient events such as ESD. Mitigation begins at the component level with ESR-optimized decoupling capacitors placed as close as possible to device pins with minimal-length ground via connections.

At the board level, a 6-layer stack-up with ground planes adjacent to every signal layer provides the most effective loop area reduction; 4-layer designs require ground pours, dense via stitching, and guard traces

alongside high-speed signals. Every via transition on a high-speed net must be treated as an RF structure, with at least one ground stitching via placed adjacent to maintain a continuous low-inductance return path. Trace routing must maintain 3 times trace-width separation, use 45° corners to avoid impedance discontinuities, and employ via fences around noise-generating zones where they close a real RF path.

Finally, a passive pi filter on the power rail - two capacitors flanking a series inductor - suppresses both externally sourced supply noise and internally generated switching transients, bridging the gap between peak MCU current demand and the supply's response time to prevent undervoltage resets.

The most durable design strategy is hierarchical: reduce the source, shrink the loop, maintain a continuous reference, prevent differential-to-common-mode conversion, treat every connector as an EMC boundary, and validate with measurements that connect frequency to physical structure. When a mitigation works, the engineer must be able to explain which current moved, which impedance changed, and why the measured spectrum or immunity response improved.

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you fully indemnify TI and its representatives against any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to [TI's Terms of Sale](#), [TI's General Quality Guidelines](#), or other applicable terms available either on [ti.com](#) or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products. Unless TI explicitly designates a product as custom or customer-specified, TI products are standard, catalog, general purpose devices.

TI objects to and rejects any additional or different terms you may propose.

Copyright © 2026, Texas Instruments Incorporated

Last updated 10/2025