Electro-Magnetic Interference and Electro-Magnetic Compatibility (EMI/EMC)

INTRODUCTION

Manufacturers of electrical and electronic equipment regularly submit their products for EMI/EMC testing to ensure regulations on electromagnetic compatibility are met. Inevitably, some equipment will fail, as the interference transmitted on cables connected to the equipment exceeds regulated limits, resulting in radiated emissions failure.

Additional problems can occur when connected equipment causes interference problems with the equipment under test resulting in component malfunction.

There are many ways to reduce the level of conducted and radiated interference, especially during the initial design of the circuit board.

These techniques include proper routing of tracks, proper use of ground planes, power supply impedance matching, and reducing logic frequency to a minimum.

Even with the most diligent employment of good EMI/EMC circuit design practices, not all interference or compatibility issues can be eliminated. At this point, additional components can be added, allowing the circuit to comply with design and regulation limits for EMI/EMC.

This engineering note will review both initial circuit board design practices and identify some after design components that can be used to solve EMI/EMC problems.

CIRCUIT DESIGN TIPS TO REDUCE EMI/EMC PROBLEMS

There are several areas where good circuit design practices are critical to the reduction or elimination of EMI/EMC problems. How the PCB layout is approached - not simply in the design but also the choice of components - directly affects the degree of EMI/EMC interference. Another area of concern is the circuit design of the power supply.

PCB Design Tips

- Avoid slit apertures in PCB layout, particularly in ground planes or near current paths
- Areas of high impedance give rise to high EMI, so use wide tracks for power lines on the trace sides
- Make signal tracks stripline and include ground plane and power plane whenever possible
- Keep HF and RF tracks as short as possible, and lay out the HF tracks first (Fig. 1)

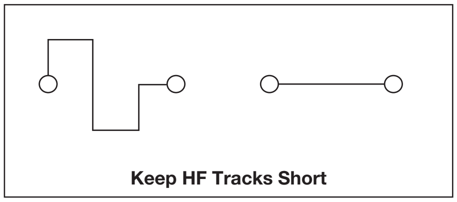


Fig. 1

- Avoid track stubs, as they cause reflections and harmonics (Fig. 2)

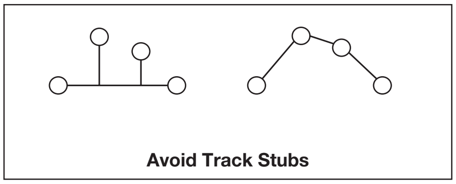


Fig. 2

- On sensitive components and terminations, use surrounding guard ring and ground fill where possible
- A guard ring around trace layers reduces emission out of the board; also, connect to ground only at a single point and make no other use of the guard ring (Fig. 3)

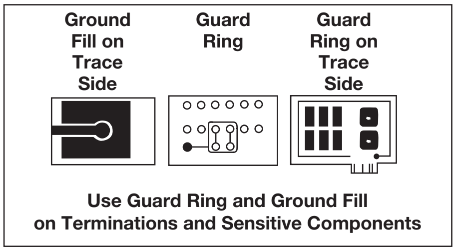


Fig. 3

- When you have separate power planes, keep them over a common ground to reduce system noise and power coupling (Fig. 4)

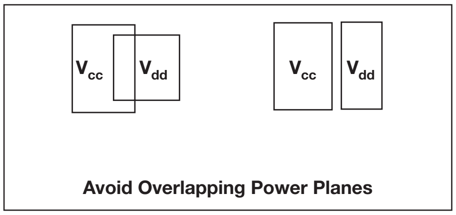


Fig. 4

- The power plane conductivity should be high, so avoid localized concentrations of via and through hole pads (surface mount is preferred mounting method)
- Track mitering (beveling of edges and corners) reduces field concentration
- If possible, make tracks run orthogonally between adjacent layers (Fig. 5)

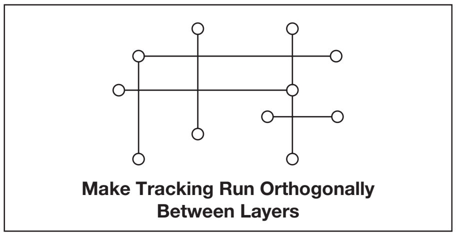
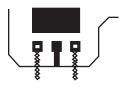


Fig. 5

- Do not loop tracks, even between layers, as this forms a receiving or radiating antenna.

- Do not leave floating conductor areas, as they act as EMI radiators; if possible connect to ground plane (often, these sections are placed for thermal dissipation, so polarity should not be a consideration, but verify with component data sheet). (Fig. 6)

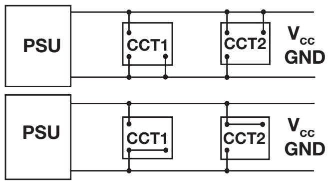


Do Not Leave Floating Conducting Areas

Fig. 6

Power Supply Considerations

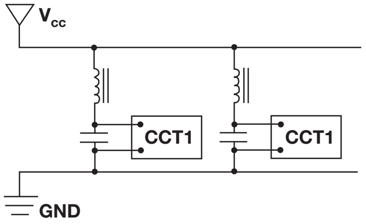
- Eliminate loops in the supply lines. (Fig. 7)



Eliminate Loops in Supply Lines

Fig. 7

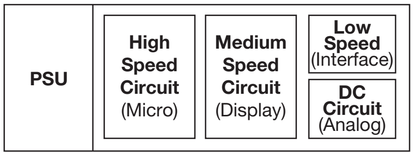
- Decouple supply lines at local boundaries. (Fig. 8)



Decouple Supply Lines at Local Boundaries

Fig. 8

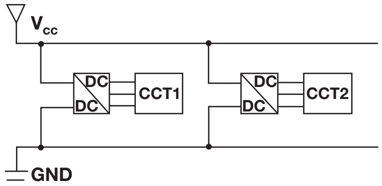
- Place high speed circuits close to Power Supply Unit (PSU) and slowest sections furthest away to reduce power plane transients. (Fig. 9)



Place High Speed Circuits Close to PSU

Fig. 9

- Isolate individual systems where possible (especially analog and digital systems) on both power supply and signal lines. (Fig. 10)

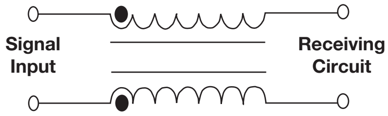


Isolated Individual Systems

Fig. 10

Component Considerations

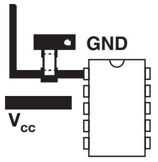
- Locate biasing and pull up/down components close to driver/bias points.
- Minimize output drive from clock circuits.
- Use common mode chokes (Vishay Dale series LPT4545 or LPT3535 or the LPE series of surface mount transformers) between current carrying and signal lines to increase coupling and cancel stray fields. (Fig. 11)



Use Common Mode Choke Between Signal Lines

Fig. 11

- Decouple close to chip supply lines, to reduce component noise and power line transients. (Fig. 12)



Decouple Close to IC Supply Lines

Fig. 12

- Use low impedance capacitors for decoupling and bypassing (ceramic multilayer capacitors, like those offered by Vishay Vitramon are preferred, offering high resonant frequencies and stability).
- Use discrete components for filters where possible (surface mount is preferable due to lower parasitic and aerial effects of termination's compared to through hole components).
- Ensure filtering of cables and overvoltage protection at the terminations (this is especially true of cabling that is external to the system, if possible all external cabling should be isolated at the equipment boundary).
- Minimize capacitive loading on digital output by minimizing fanout, especially on CMOS ICs (this reduces current loading and surge per IC).

If available, use shielding on fast switching circuits, main power supply components and low power circuitry (shielding is expensive and should be considered a "last resort" option).

MAGNETIC COMPONENTS FOR ELECTRO MAGNETIC INTERFERENCE REDUCTION AND ELECTRO MAGNETIC COMPATIBILITY

Products that use magnetics to reduce electro-magnetic interference and improve electro-magnetic compatibility within the circuit can be classified into several categories: inductors, chokes, transformers, ferrite beads, capacitors, and integrated passive devices that can incorporate any or all of the above devices. When considering any of these EMI/EMC components, it is necessary to identify circuit paths or areas likely to conduct or radiate noise.

Inductors

The most common magnetic EMI filter is the inductor or choke. Inductors are used for both line filtering and energy storage. If a circuit is suspected of being a source for EMI, often, selection of the right inductor can help eliminate the problem. For radiated interference, the choice of a shielded or toroidal inductor can often eliminate (or at least greatly reduce) the offending frequency. In fact, toroidal inductors like Vishay Dale's LPT-4545 and LPT-3535 surface mount, or Vishay Dale's TE, TD, or TJ series of leaded toroids virtually eliminate radiated fields because of the toroid's unique ability to contain the magnetic flux within its core.

LPT 4545 Toroidal Inductor

The toroid is also less susceptible to induced noise from other components as the applied magnetic field would induce equal and opposite currents inside the toroid, thus canceling the induced interference.

Chokes

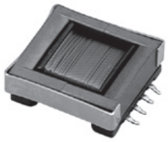
Common mode and differential mode chokes are used to eliminate noise on a pair of conductors. Common mode noise is defined as noise that is present or "common" to both conductors, and can be the result of induced noise caused by the "antenna" effect of a conductor or PC trace. Common mode noise is typically "in phase" within the conductors, while differential noise is present on only one conductor or present in opposite phase in both conductors. Common mode chokes use the properties of two closely coupled magnetic fields to eliminate the interference problem by canceling the noise within the magnetic fields. They are best employed to eliminate noise or EMI on cables or signal tracks. The choke should be located as close to the driver or receiver circuit as possible, or at the signal entry point of the circuit board. The proper selection of inductive component can also help in matching line impedance and can act as a bandwidth filter for the circuit. Vishay Dale's LPT and LPE series products can be configured in the common or differential mode depending on your application.

Transformers

The main benefit of using a transformer for EMI/EMC is that it can provide an isolation barrier between a signal line and the signal processing circuit (particularly where the signal line exits the board or system). This is true of signals being driven or received, since isolating the line reduces common mode noise and eliminates ground (or signal return) potential differences between systems.

One particular area where high noise immunity is essential is in thyristor/triac driving circuits. Here the transformer provides an isolation between the driven load and a logic based controller. The isolating pulse transistor provides much better noise immunity than an insulated gate bi-polar transistor (IGBT) due to inherently lower coupling capacitance (typically 10's of pF for a pulse transformer compared to nF for a power IGBT device). The lower coupling capacitance improves the circuit's immunity from noise from the main power supply or from power switching devices. Vishay Dale's LPE and PT transformers can be used to meet your transformer needs. Many more EMI/EMC configurations can be provided through our custom magnetic design department.

LPE Series Transformer



Surface Mount Ferrite Beads

Chip impeders, also called ferrite chip beads, perform the function of removing RF energy that exists within a transmission line structure (printed circuit board trace). To remove unwanted RF energy, chip beads are used as high frequency resistors (attenuators) that allow DC to pass while absorbing the RF energy and dissipating that energy in the form of heat.

ILBB/ILB Ferrite Beads

Surface mount ferrite beads have many advantages:

- Small and light weight
- Inexpensive
- High impedance values removes broad range of RF energy
- Closed magnetic circuit eliminates cross talk
- Beads are inherently shielded
- Low DCR ratings minimizes desired signal degradation
- Excellent current carrying capacity compared to alternatives
- Outstanding performance at removing RF energy
- Spurious circuit oscillations or resonances are reduced because of the bead's resistive characteristics at RF frequencies
- Broad impedance ranges (several Ω to 2000 Ω)
- Operates effectively from several MHz to 1 GHz

To chose the proper bead, you should consider the following:

1. What is the range of unwanted frequencies?
2. What is the source of the EMI?
3. How much attenuation is required?
4. What are the environmental and electrical conditions for the circuit (temperature DC voltage, DC bias currents, maximum operating currents, field strengths, etc...)?
5. What is the maximum allowable profile and board real estate for using this component?

Selection of the right bead for your particular frequencies is not a simple process. In most cases, since beads are only rated for impedance at 100 MHz, you will need to look at several graphs to determine the best bead for your frequency if it is different than 100 MHz.

This is a time consuming but necessary process to select the correct bead value since the highest impedance bead at 100 MHz is not necessarily the highest impedance bead at higher or lower frequencies. DC bias will also lower the effective impedance of the device.

EMI/EMC Component Selection

Before incorporating EMI/EMC components, it is necessary to identify the circuit paths and circuit areas most likely to conduct noise, and to identify circuit areas likely to act as antennas and radiate noise. At this point the most appropriate location for the chosen components can be determined.

The actual components chosen are determined by the frequency and signal level of the noise to be eliminated. Consideration should also be given for the frequencies that are to remain intact.

For attenuation less than 5dB inductive, EMI components are generally the best choice. For attenuation less than 5 dB, circuit type must first be considered.

Working with a high speed signal circuit, your best choice is a complex filter consisting of inductive and capacitive components (such as an LCR Filter). If your circuit is a general signal type (i.e., not a high speed circuit) grounding stability must first be determined. For stable grounds, capacitive EMI components are an excellent choice.

However, if the circuit has an unstable ground, high impedance inductive components should be considered for EMI suppression needs.

Designing equipment and choosing components is not an easy process. Often, the only measure of design success is the overall radiation level from your equipment. Trial and error is a long tedious process that can take several months to complete, and choosing the wrong component can waste time.

Here are three suggestions for more effective design:

- Always place EMI/EMC components as close as possible to the noise source.
- Select EMI/EMC components that match the impedance of the noise conduction path, not necessarily that of the circuit path. Remember that common mode noise often travels a different path than the circuit current.
- Start with EMI/EMC components that offer sufficient performance to meet your design standards. Component costs can be reduced once you have a working design.

VISHAY COMPONENTS FOR EMI/EMC COMPLIANCE

Surface Mount Ferrite Beads
ILB-1206, ILBB-0402 to ILBB-1812
Surface Mount, High Current Ferrite Beads
ILHB-0603 to IHLB-1812
Surface Mount Bead Arrays
ILAS-1206
Surface Mount Ferrite Inductors and Chokes
LPT-4545, LPT-3535
Surface Mount Transformers
LPE Series
Surface Mount Ceramic and Tantalum Capacitors
Ferrite Beads for EMI/EMC Compliance

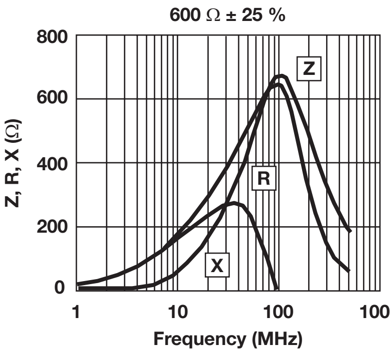
One of the simplest and most effective ways to reduce EMI is through the use of ferrite beads. Initially, EMI suppression consisted of a small bead-shaped ferrite (hence the name bead) with a hole through the middle. The ferrite bead was slipped over the suspected “noisy” wire or component lead and EMI was reduced.

Today, beads are available in a variety of styles including the original through-hole model, multiple apertures and surface mount configurations.

How Ferrite Beads Work

The best way to conceptualize a bead is as a frequency dependent resistor. An equivalent circuit for a bead consists of a resistor and inductor in series. The resulting change (of impedance over frequency) is directly associated with the frequency dependent complex impedance of the ferrite material.

At low frequencies (below 10 MHz) the inductive impedance is 10 Ω or less, as shown below. At higher frequencies, the impedance of the bead increases to over 100 Ω, and becomes mostly resistive above 100 MHz.



Since the bead's impedance is essentially resistive to high frequency circuits, the problem of resonance experienced by other EMI filtering choices like capacitors and inductors is eliminated. Often the bead is the only practical solution to an EMI problem.

When used as a high frequency filter, ferrite beads provide a resistive loss that attenuates the unwanted frequencies through minute heating of the bead's ferrite material due to eddy currents. At the same time, the bead presents minimal series impedance to the lower frequency or direct currents of the circuit.

USEFUL TABLES FOR EMI/EMC DESIGNS

DECIBELS

DB [1]	POWER RATIO	VOLTAGE CURRENT RATIO		DB [2]	POWER RATIO	VOLTAGE CURRENT RATIO
0	1.0	1.0		0	1.0	1.0
3	2.0	1.4		- 3	0.50	0.71
6	4.0	2.0		- 6	0.25	0.50
10	10.0	3.2		- 10	0.10	0.32
12	16.0	4.0		- 12	0.05	0.25
14	25.0	5.0		- 14	0.04	0.20
20	10^2	10		- 20	10^{-2}	0.10
30	10^3	32		- 30	10^{-3}	0.03
40	10^4	10^2		- 40	10^{-4}	10^{-2}
60	10^6	10^3		- 60	10^{-6}	10^{-3}
80	10^8	10^4		- 80	10^{-8}	10^{-4}
100	10^{10}	10^5		- 100	10^{-10}	10^{-5}
120	10^{12}	10^6		- 120	10^{-12}	10^{-6}
140	10^{14}	10^7		- 140	10^{-14}	10^{-7}

Notes

(1) $\mathrm{dB} = 10 \log_{10} \frac{P_1}{P_2}$

(2) $\mathrm{dB} = 20 \log_{10} \frac{V_1}{V_2} = 20 \log_{10} \frac{I_1}{I_2}$

ELECTRIC FIELD LEVELS

W	1 m	10 m	100 m	1 km	10 km
1	5.5 V/m	0.55 V/m	0.05 V/m	5.5 V/m	0.55 mV/m
10	17.4 V/m	1.7 V/m	0.17 V/m	17 V/m	1.7 mV/m
100	55 V/m	5.5 V/m	0.55 V/m	55 V/m	5.5 mV/m
1K	174 V/m	17.4 V/m	1.74 V/m	170 V/m	17 mV/m
10K	550 V/m	55 V/m	5.5 V/m	550 V/m	55 mV/m
100K	1740 V/m	174 V/m	17.4 V/m	1.74 V/m	174 mV/m

Notes

- Table assumes an antenna gain of one
- $E = \frac{5.5\sqrt{PA}}{d}$
- P = Power at antenna in W
  d = Distance from antenna in m (valid when d > $\lambda/2\pi$)
  E = Electric field in V/m
  A = Antenna gain (1 for table)

CISPR 22 LIMITS

FREQUENCY (MHz)	CLASS A	CLASS B
RADIATED		
30 to 230	40 dB πV/m	30 dB V/m
230 to 1000	47 dB πV/m	37 dB V/m
Quasi-peak, antenna at 10 m		
CONDUCTED		
0.15 to 0.50	66 dB πV/m	56 dB πV/m to 46 dB πV/m
0.50 to 5	60 dB πV/m	46 dB πV/m
5 to 30	60 dB πV/m	50 dB πV/m
Average		

FREQUENCY VS. WAVELENGTH

f	λ	$\lambda/2$	$\lambda/20$
10 Hz	30 000 km	4800 km	1500 km
60 Hz	5000 km	800 km	250 km
100 Hz	3000 km	480 km	150 km
400 Hz	750 km	120 km	37 km
1 kHz	300 km	48 km	15 km
10 kHz	30 km	4.8 km	1.5 km
100 kHz	3 km	480 m	150 m
1 MHz	300 m	48 m	15 m
10 MHz	30 m	4.8 m	1.5 m
100 MHz	3 m	0.48 m	15 cm
1 GHz	30 cm	4.8 cm	1.5 cm
10 GHz	3 cm	4.8 mm	1.5 mm

Note

- f = Frequency
  λ = Wavelength
  $\lambda/2\pi$ = Near field to far field distance
  $\lambda/20$ = Antenna effects of wires and slots

CAPACITOR SELF RESONANCE

FARADS	TOTAL LEAD LENGTH		
	1/4"	1/2"	1"
500 pF	100 MHz	72 MHz	50 MHz
1000 pF	72	51	36
0.01 F	23	16	11
0.1 F	7.2	5.1	3.6
0.3 F	4.2	2.9	2.1
0.5 F	3.2	2.3	1.6

Note

- $f = \frac{1}{2\pi\sqrt{LC}}$; L = 20 nH/inch

RISE TIMES - FREQUENCY - LENGTH

t_r	f_{eq}	L_{cross}	$L_{cross/2}$	L_{term}
1 ns	318 MHz	1.0 ft.	6"	3"
3 ns	95 MHz	3.0 ft.	1.5 ft.	9"
10 ns	32 MHz	10 ft.	5 ft.	2.5 ft.
30 ns	9.5 MHz	30 ft.	15 ft.	7.5 ft.
100 ns	3.2 MHz	100 ft.	50 ft.	25 ft.
300 ns	950 MHz	300 ft.	150 ft.	75 ft.
1 µs	320 MHz	1000 ft.	500 ft.	250 ft.

Note

- t_r = Rise time
  f_{eq} = Equivalent frequency $I/\pi tr$
  L_{cross} = Length of one rise time in free space
  $L_{cross/2}$ = Typical length of rise time on cable or printed circuit board (crosstalk)
  L_{term} = Length of terminate on cable or printed circuit board