

AN10897

A guide to designing for ESD and EMC

Rev. 02 — 19 January 2010

Application note

Document information

Info	Content
Keywords	ESD, EMC, PCB design
Abstract	An introductory approach to designing for ESD. Understanding the ESD pulse, how passive components react over frequency, and PCB layout techniques are exposed.

Revision history

Rev	Date	Description
02	20100119	application note; second release
Modifications:		
		<ul style="list-style-type: none"> • Title of application note modified: appended “and EMC” • Section 2.3.5 “Filters”, 1st bullet item, 2nd sub-bullet: changed from “high filter capacitors” to “high speed capacitors” • Section 2.3.6 “Ground”, 2nd equation following 3rd paragraph changed from “$V = 20 \times 10e-9 \times (10A/10e-9)$” to “$V = 20 \text{ nH} \times (10 \text{ A} / 10 \text{ ns})$” • Section 3.1 “Board structure - functional grouping”: 3rd paragraph re-written. • Section 3.1.1.1 “4-layer board stack-up”, 2nd sentence: changed from “use a wide core” to “use a thick core”. • Figure 18 “Recommended stack-ups” modified (adjusted height of stack-up layers) • Section 3.1.2 “Ground planes”: <ul style="list-style-type: none"> – deleted 3rd and 4th bullet items following the 3rd paragraph. – 4th paragraph (immediately follows Figure 23) was re-written (now 4th, 5th and 6th paragraphs). • Figure 26 “Ground island and vias” modified (added label “vias to ground” on left side). • Figure 31 “Ground plane boundary” modified (added “20 × layer spacing” measurement)
01	20091214	application note; initial version

Contact information

For more information, please visit: <http://www.nxp.com>

For sales office addresses, please send an email to: salesaddresses@nxp.com

1. Introduction

This application note is written to give the user an introductory approach to dealing with ESD. Be it a Pachinko machine or a washing machine or a hand held remote control, ESD is a threat to the operation of the system and it needs to be addressed head-on.

Given the nature of the product, it will be exposed to ESD events that range from the normal charge build-up in human beings using the equipment, charge build-up by moving parts (due to triboelectric charging), or by malicious attacks by individuals.

It is important to note that this manual is a set of guidelines that will improve the system immunity to ESD events and that it is not a guarantee that it will prevent an ESD event from altering the performance of your product or eliminating potential damage.

When it comes to ESD, keep the following quote from Og Mandino in mind: “Never has there been a map, however carefully executed to detail and scale, which carried its owner over even one inch of ground.” It is upon you to take action into implementing the recommendation presented and testing the results of the recommendation in your application. You must also test for ESD at various points in the design phase to guarantee performance of the final product.

2. The basics

2.1 The ESD pulse

2.1.1 Characteristics of the ESD pulse—the IEC model

In this guide the ESD pulse we will be referring to is the one defined in IEC 61000-4-2. The typical test circuit is shown in [Figure 1](#) and the characteristics of the pulse is shown in [Figure 4](#).

There are other specifications such as the Human Body Model (HBM) (see [Figure 2](#) for typical HBM test circuit) and the Machine Model (MM). These models are device level specifications and are used to qualify IC components and intended for customer assembly production lines. Device-level ESD pulses (HBM, MM, CDM) and system-level ESD pulses (IEC) should be considered with separate test acceptance criteria. There is a disconnect in the EMC (ElectroMagnetic Compatibility) world between system manufacturers testing systems for upset and device manufacturers testing devices for failure. Some system level manufacturers are pushing device manufacturers to test semiconductor devices using system-level compliance standards—specifically, IEC 61000-4-2 for ESD (ElectroStatic Discharge). Product manufacturers would like to believe that if devices are qualified to IEC standard(s), finished products will likewise be qualified. Unfortunately, there is a fundamental difference between system-level and device-level testing. [Table 1](#) shows a comparison between the peak current of HBM versus IEC 61000-4-2 standards.

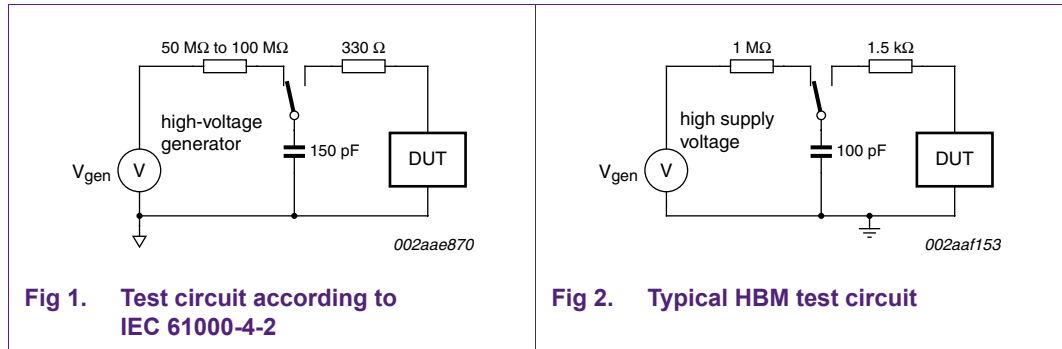


Fig 1. Test circuit according to IEC 61000-4-2

Fig 2. Typical HBM test circuit

Table 1. Peak current of HBM versus IEC 61000-4-2 ESD standards

Applied voltage (kV)	Peak current (A) Human Body Model	Peak current (A) IEC 61000-4-2
2	1.33	7.5
4	2.67	15.0
6	4.00	22.5
8	5.33	30.0
10	6.67	37.5

The human body has the ability to charge and store energy with a charge that can reach several thousands of volts (8 kV to 10 kV is common) and peak currents from 1 A up to 200 A. The timing in which this charge is delivered varies as follows:

- Rise time: $t_r = 200 \text{ ps to } 20 \text{ ns}$
- Spike time: $t_{\text{spike}} = 0.5 \text{ ns to } 10 \text{ ns}$
- Total time: $t_{\text{tot}} = 100 \text{ ns to } 2 \mu\text{s}$

The IEC specification ESD waveform has similar characteristics, however, the amount of energy delivered is far greater than a HBM pulse. Figure 3 and Table 2 show the current levels at different time points.

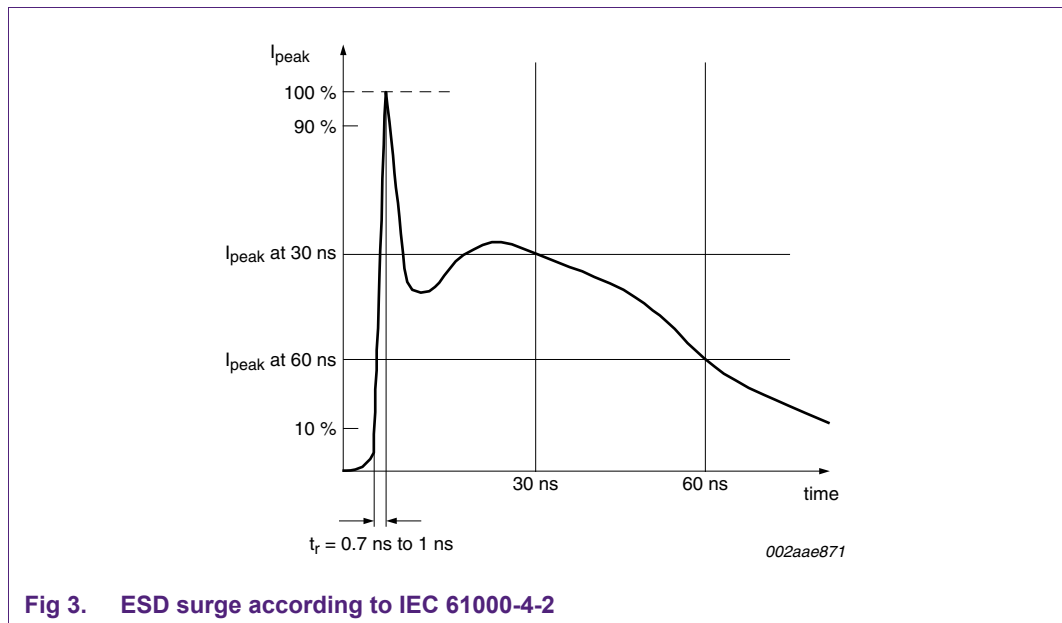


Fig 3. ESD surge according to IEC 61000-4-2

The IEC-61000-4-2 also specifies compliance levels, with a Level 4 being the most severe and representing a 8 kV contact discharge or a 15 kV air discharge onto the Equipment Under Test (EUT).

Table 2. IEC 61000-4-2 ESD surge classification

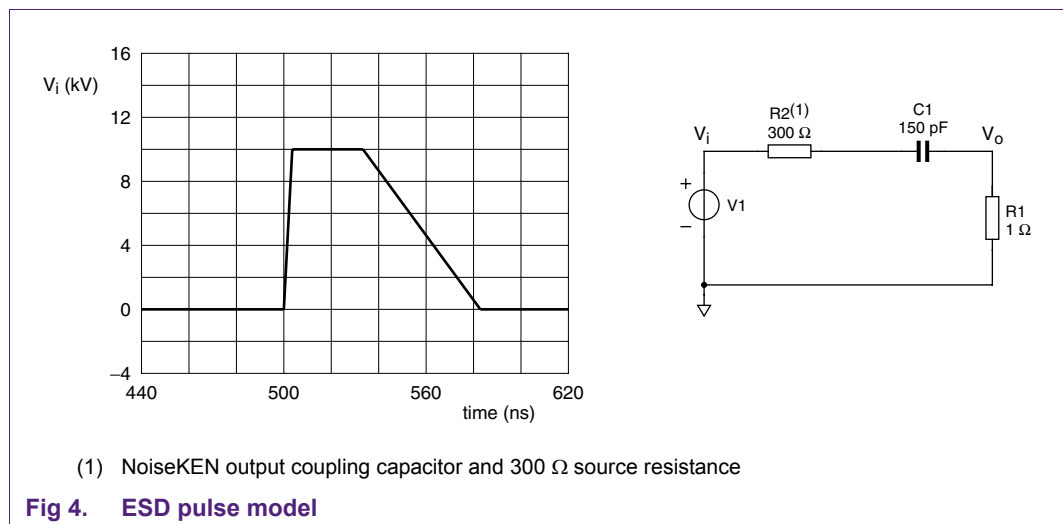
1a - Contact discharge			1b - Air discharge	
Class	Test voltage (kV)	Maximum current (A)	Class	Test voltage (kV)
1	2	7.5	1	2
2	4	15	2	4
3	6	22.5	3	8
4	8	30	4	15
X	special	special	X	special

When a discharge into equipment occurs, the discharge current will flow through the equipment through different paths. The high frequency components will flow radially, whereas the low frequency components will seek the path of least resistance to ground. This current flow will excite any existing antennas (antennas are described later) in their path. The efficiency (gain) of these antennas are primarily dependent on their size. Since a 1/4 wave length antenna is very efficient, it can easily have a dimension of 1.5 cm to 150 m. For ESD events, even a 1/20 wave length antenna may conduct a significant amount of energy.

2.1.2 Putting the ESD problem into perspective

We will model an ESD pulse with a simple circuit:

- V1 = voltage source
- R2 = generator source resistance
- C1 = capacitance A
- R1 = load resistance



Given the following equations:

$$\text{Energy: } Q = CV \tag{1}$$

$$I \times t = CV \tag{2}$$

Then, $150 \text{ pF} \times 8 \text{ kV}$ is approximately $1 \text{ }\mu\text{J}$.

With a generator source resistance of $300 \text{ }\Omega$, I_{peak} for 50 ns event is therefore 20 A .

If the voltage is clamped, then the current and/or time will **increase** to maintain the $I_t = CV$ energy relationship.

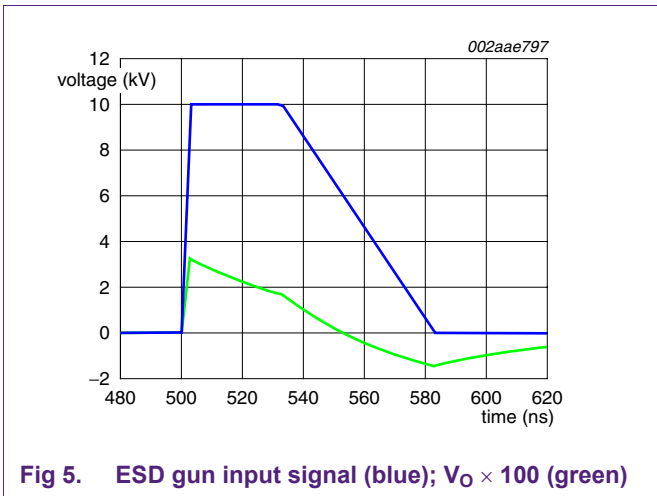


Fig 5. ESD gun input signal (blue); $V_O \times 100$ (green)

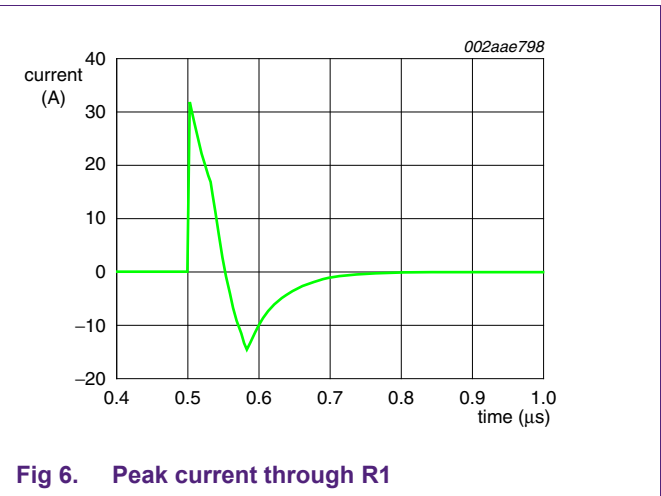


Fig 6. Peak current through R1

The current and voltage surge of an ESD pulse can cause malfunctions that can be classified into three general categories:

- Temporary errors that are automatically corrected by the EUT or its operating software.
- Temporary errors that require some intervention by the operator to recover to normal operation.
- Permanent damage that requires repairing or replacing part or all of the EUT.

These failure modes are caused by ESD current flowing directly through a vulnerable circuit, ESD current flowing in the circuit ground plane, and in sensitive circuits due to electro-magnetic coupling, or due the pre-discharge electric field.

2.2 Effects and failure modes

2.2.1 Sources, paths and receptors

In the field of EMI, interference can be broken down into:

- The source of interference
- The receptor of the interference
- The path of the interference

Sources can be microprocessors, video drivers, ESD, Audio amplifiers, power disturbances, lightning, motors, oscillators.

Receptors can be microprocessors, LED controllers, motor controllers, audio processors, reset circuits, low level analog, interrupt and control lines.

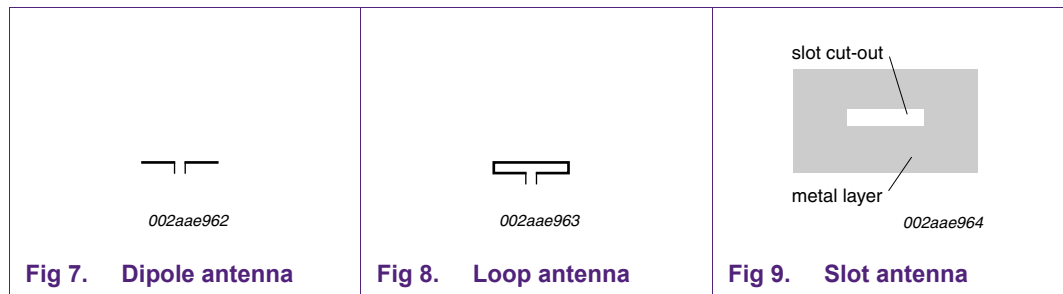
Paths can be divided into 2 categories:

- Radiated: EM fields, inductive or capacitive crosstalk
- Conducted: Signal, power, ground traces

2.2.2 Unintentional antennas

Unintentional antennas are formed in a system due to the signal carrying traces. For example you have:

- Transmission line
- Dipole antenna; typically formed by cables (see [Figure 7](#))
- Loop antenna: typically found on PCBs due to layout (see [Figure 8](#))
- Slot antennas; typically found in PCBs due to cut/split ground planes (see [Figure 9](#))



Critical components:

- Equal/opposite current minimize radiation.
- Overall dimensions (i.e., 1/4 wavelength and area) are key parameters
- Radiation varies with respect to current (I), frequency (f), area (A) or length (L).

Resonances can amplify the unwanted radiated emissions.

Table 3. Frequency versus wavelength

Frequency (MHz)	λ	$\lambda / 2$	$\lambda / 4$	$\lambda / 20$
1	300 m	150 m	75 m	15 m
10	30 m	15 m	7.5 m	1.5 m
30	10 m	5 m	2.5 m	50 cm
100	3 m	1.5 m	75 cm	15 cm
300	1 m	50 cm	25 cm	5 cm
1000	30 cm	15 cm	7.5 cm	1.5 cm

If we consider the rapid rise time of an IEC61000-4-2 ESD pulse to be equivalent of 300 MHz, then from [Table 3](#) you can see that a 5 cm trace is a critical length during an ESD event as it may be a very efficient $\lambda / 20$ antenna.

2.3 An overview of R, L, C and D components

2.3.1 Resistor (R) component characteristics

Resistors serve to reduce current and dissipate as heat.

Since they provide a voltage drop at DC only, small value resistors (<100 Ω) in series are practical in applications.

A resistor's end-to-end capacitance limits its overall impedance. A 1 M Ω resistor at DC is no longer the same at 100 MHz. Also, ESD voltages tend to arc across surface mount resistors, reducing their effectiveness. In this case, use carbon or thick film resistors for best results.

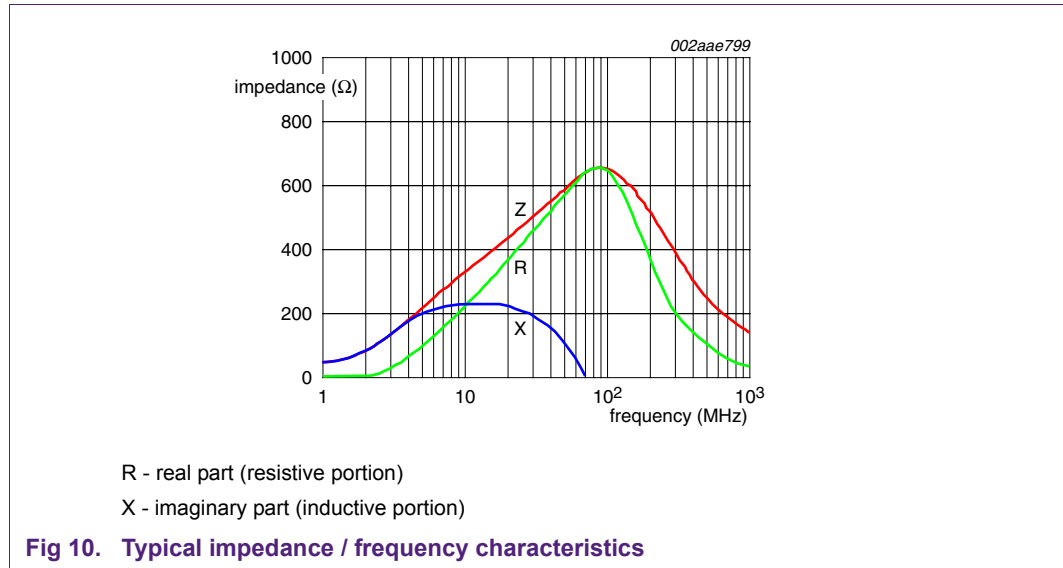
Resistors are low cost, but required values to limit I/O pin current will interfere with normal circuit operation. Example: to limit 6 kV ESD event current to 10 mA requires 600 k Ω resistor—not practical (ignoring IC peak current capability in this example).

2.3.2 Inductor (L) component characteristics

Wire wound inductors are commonly used EMI filters. They are good to about 50 MHz. For ESD and EMI applications, a ferrite is preferred at the higher frequencies.

SMD ferrites are one-turn inductors that are very useful as frequency dependent resistors. They allow low frequency signals to pass, while blocking high frequency pulses (i.e., ESD, EFT) and dissipating energy as heat.

A ferrite is usually chosen based on its resistive, rather than inductive, frequency range. For example, if we wanted to attenuate a 100 MHz EMI signal, then the ferrite shown in [Figure 10](#) would provide about 630 Ω of resistance at that frequency.



Ferrites have low impedance at typical system operating frequencies so they do not interfere with normal circuit operation. Ferrites also convert ESD energy to thermal energy very effectively.

2.3.3 Capacitor (C) component characteristics

Capacitors serve as filters. They can be used to filter the high frequency pulses produced by an ESD event. They also provide switching current to ICs and serve as energy storage bins that minimize voltage drop.

Capacitors are low cost and offer basic protection. There should be minimum one decoupling capacitor per IC on the board. More capacitors are better, and should be symmetrically placed around the ICs. Smaller size capacitors should be placed in parallel to larger capacitors to increase the frequency response.

Choose decoupling capacitors carefully. They become inductive above their resonant frequency as shown in [Figure 11](#).

A well-chosen high frequency capacitor is used for decoupling power and ground and signal filtering. Multi-layer ceramic tend to work well up to about 1 GHz, with lossy capacitors preferred for ESD applications due to less problems with resonance.

Medium and low frequency capacitors such as Mylar are not acceptable for high frequency filters, and electrolytic low frequency capacitors are good only for energy storage.

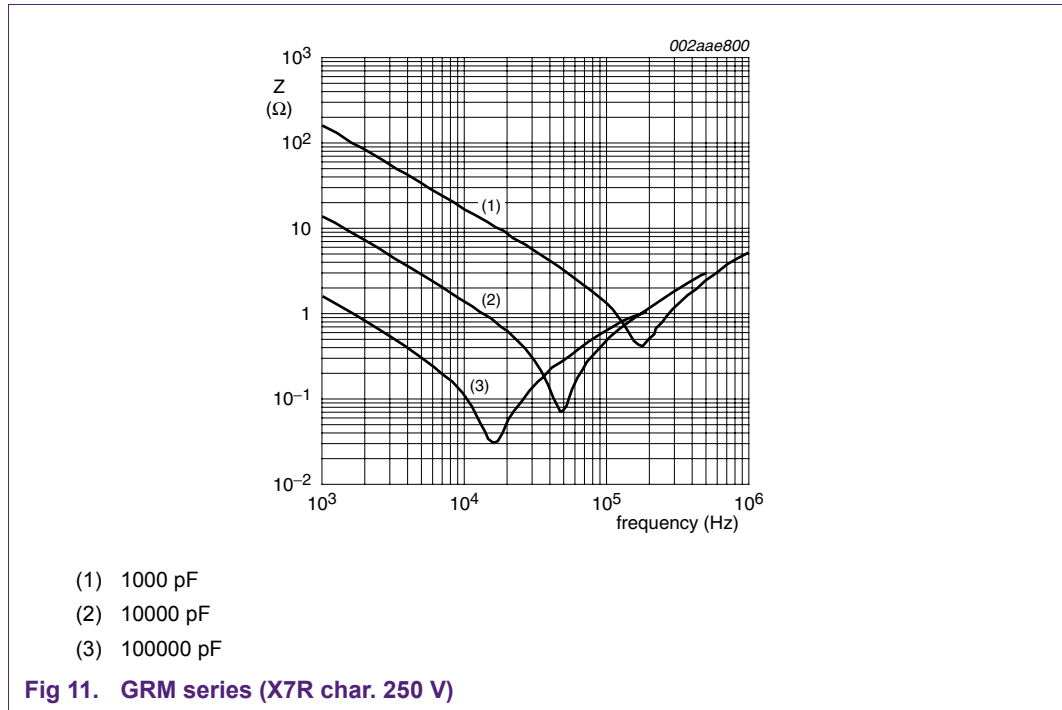
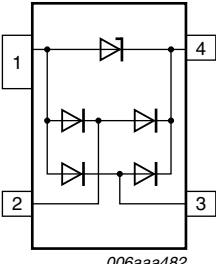
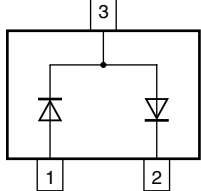



Table 4. Capacitor selection

Type	Maximum frequency
aluminum electrolytic	100 kHz
tantalum electrolytic	1 MHz
paper	5 MHz
mylar	10 MHz
polystyrene	500 MHz
mica	500 MHz
ceramic	> 1 GHz

2.3.4 Diodes (D)

Zener diodes and transorbs (transient voltage suppression diodes) help divert current to power or ground rails away from IC and provide voltage clamping.

 <p style="text-align: center;">006aaa482</p> <p style="text-align: center;">Ultra low capacitance double rail-to-rail ESD protection diode.</p> <p>Fig 12. PRTR5V0U2AX</p>	 <p style="text-align: center;">006aaa763</p> <p style="text-align: center;">High-speed switching diodes divert current away from sensitive circuits.</p> <p>Fig 13. BAV99</p>	 <p style="text-align: center;">002aae801</p> <p style="text-align: center;">Can be used to clamp voltages.</p> <p>Fig 14. Zener diode</p>
---	---	--

2.3.5 Filters

When constructing EMI filters consider the following:

- Filter components:
 - Shunt elements to be high speed capacitors
 - Use multi-layer ceramic capacitors with short lead lengths
 - Series elements to be resistors or ferrites.
 - Avoid use of low loss inductors; lossy ferrites are preferred, especially above 50 MHz.
- Filter termination:
 - The low-impedance capacitor should be connected to appropriate ground. Chassis ground for immunity (ESD), signal ground for emissions.

2.3.6 Ground

‘Ground’ is a return path for current, usually a point to which circuit voltage is referenced.

A ground plane or trace is a dynamic element. There are no guarantees that any two points on the plane or trace will remain at the same potential during an ESD event.

For example, for 20 nH ground inductance (= 20 mm trace) and a 10 A pulse lasting for 10 ns:

$$V = L(di/dt)$$

$$V = 20 \text{ nH} \times (10 \text{ A} / 10 \text{ ns})$$

$$V = 20 \text{ V enough to corrupt signals and cause major system problems}$$

Circuit or signal ground — may be logic or analog ground. Cannot be relied upon to be at the same potential as adjacent PCB boards.

Backplane or motherboard ground — common reference present on a board to which other boards are connected.

Enclosure ground — safety ground connection to earth, may or may not be connected to chassis ground.

Line neutral — In the USA, connected to earth ground at power entrance or an inverting transformers.

2.4 ESD mitigation

2.4.1 Hardware ESD management strategy

- ESD protection requires the peak voltage and current to be limited to safe levels for sufficiently short periods so as not to cause permanent damage to the chip.
- Damage is cumulative: 50 ESD events will cause more damage than a single ESD event or 3 ESD events (IEC spec).
- With no input ESD protection, either internal to the IC or augmented with external components when trying to meet more stringent tests, the IC will fail.

2.4.2 Software ESD management strategy

Software or firmware plays an important role in reducing the effects of ESD. Software is an effective way to recover from non-destructive ESD events.

As with a hardware ESD fix, a software ESD fix has its associated costs: increased development time, memory requirement, and processing time. This cost can be compared to a hardware fix alone, however, the redundancy that a software fix can provide may pay off in the future.

The general approach to writing robust software is to assume that the state of ports or registers has been unintentionally changed within a period of time. To address this, the programmer can:

1. Use refresh routines.
2. Use checking and restoring.
3. Unused interrupts return to main program.
4. Use an internal or external watchdog timer.

Recommendations to improve system robustness:

- Always assume the data has been corrupted since the last access.
- Use a hardware watchdog to reset the system in case of a hang-up.
- Re-write all IO register data regularly to ensure system has correct data.
- Unused interrupts should have a return to main.
- Use checksums or CRC on large blocks of data.

2.4.2.1 Refresh

When refreshing, checking for the validity of prior data is not conducted. The software simply re-writes data into registers that are required for proper operation. This is done even if the port/register has not been used since the last write.

When using refresh, keep in mind the order in which the refresh should occur. You must assume that the register values are incorrect, and that by enabling a port before setting its correct state may lead to undesired results.

2.4.2.2 Checking and restoring

You can also build in periodic checks to verify that all your components are running as programmed. For large blocks of data you can use checksums or Cyclic Redundancy Checks (CRC) for fast check. If these checks provide errors, then you must choose an appropriate recovery routine. You may want to restore only portions of your system or save data for diagnostics.

2.4.2.3 Unused interrupts and registers

You can cover the software 'holes' by placing all unused registers/ports in a known state. Pay special attention to unused interrupt vectors. A good practice is to place a 'return to main' statement in unused interrupt vectors, which allows the program to continue in case an ESD event causes the microcontroller to erroneously point to an unused interrupt.

2.4.2.4 Watchdog timers

In the event that the software enters an unrecoverable state, a watchdog timer can be used to force a reset or other recovery process.

External watchdog timers are also recommended. They come in stand-alone IC packages that can be used to recover the system if the host fails.

The key is to always leave a path for the software and hardware to return to a valid operating point so that it can continue execution.

3. A practical approach to schematic and PCB design

Electromagnetic compatibility and ESD immunity must be considered in the early design stage of a system. If ignored, problems encountered later in testing or in the field will become very difficult and expensive to fix.

A crucial element in EMC/ESD immunity is a solid PCB design. Good practices such as decoupling power and I/O lines to ground, voltage and frequency (bandwidth) limiting, wave shaping (edge-rate control), using negative feedback, refresh cycling, WDT, and fault tolerant software—all play collectively an important role in improving EMC and ESD immunity.

3.1 Board structure - functional grouping

A PCB board should be divided by functional groups: analog, digital, power supply, high noise, and IO groups. [Figure 15](#) shows the sample placement of each group.

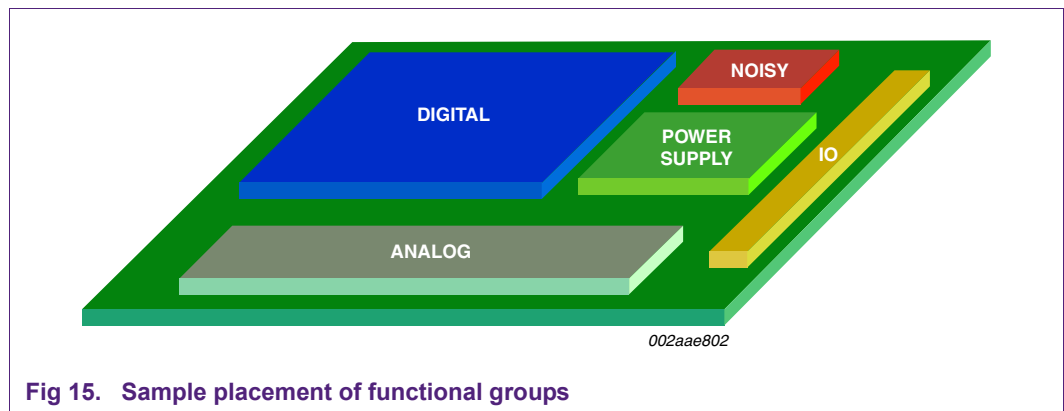


Fig 15. Sample placement of functional groups

In the schematic, each component should have a reference designator identifying the group to which it belongs. This will provide the layout engineer with the necessary information for the correct placement of components on the board. Traces of a given group should not cross over to a different group unless it is an interface trace between the groups.

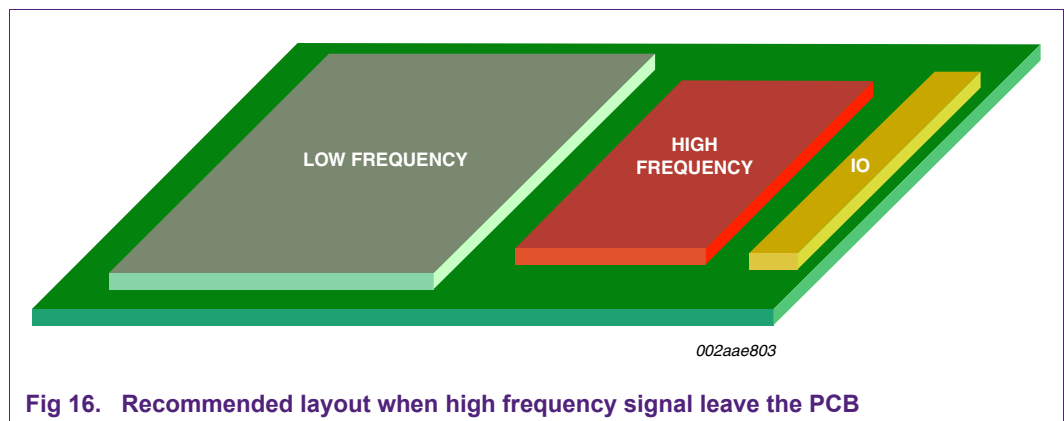


Fig 16. Recommended layout when high frequency signal leave the PCB

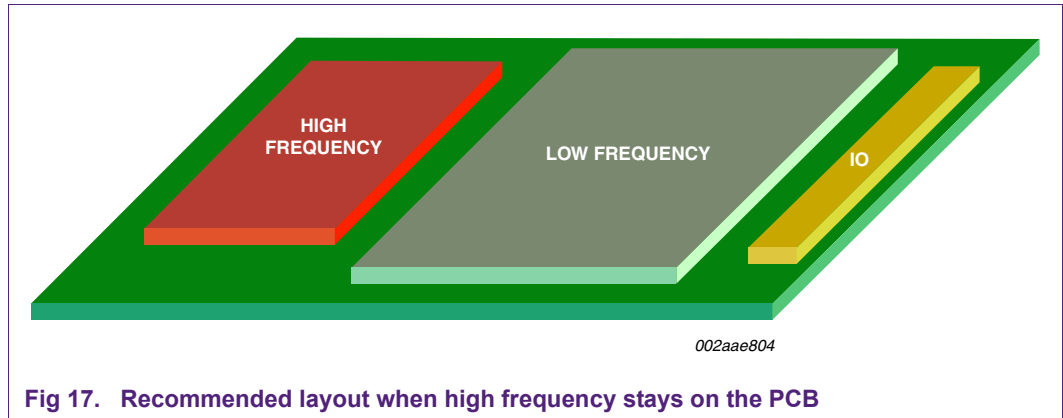


Fig 17. Recommended layout when high frequency stays on the PCB

High frequency circuits that produce signals that leave the PCB should be placed close to the I/O area to minimize their loop areas on the PCB (Figure 16). To minimize coupling into I/O interconnects, circuits that produce high frequency signals that do not leave the PCB are placed away from the I/O connectors to minimize coupling (Figure 17). The separation 'rule of thumb' is 2 inches (50 mm).

Routing signal traces perpendicular (90°) to each other on adjacent layers of a printed-circuit board may help to minimize crosstalk.

3.1.1 Multi-layer boards recommended stack-ups

3.1.1.1 4-layer board stack-up

The two external trace layers usually have orthogonal trace routing directions. To improve the performance, use a thick core and place the signal layers close to the planes.

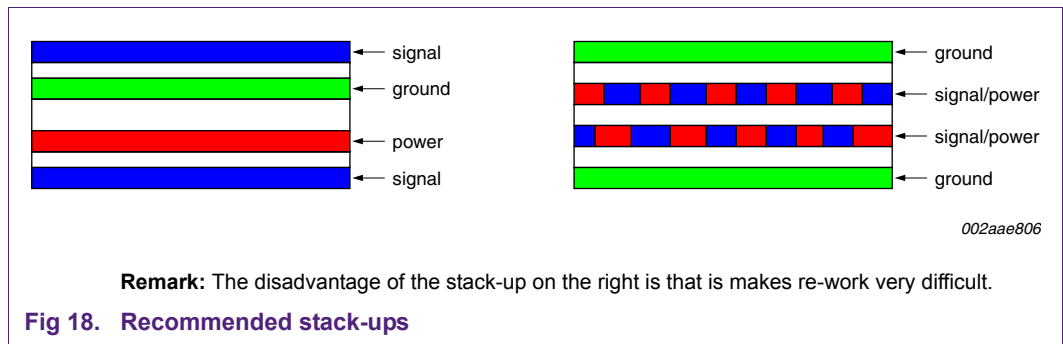
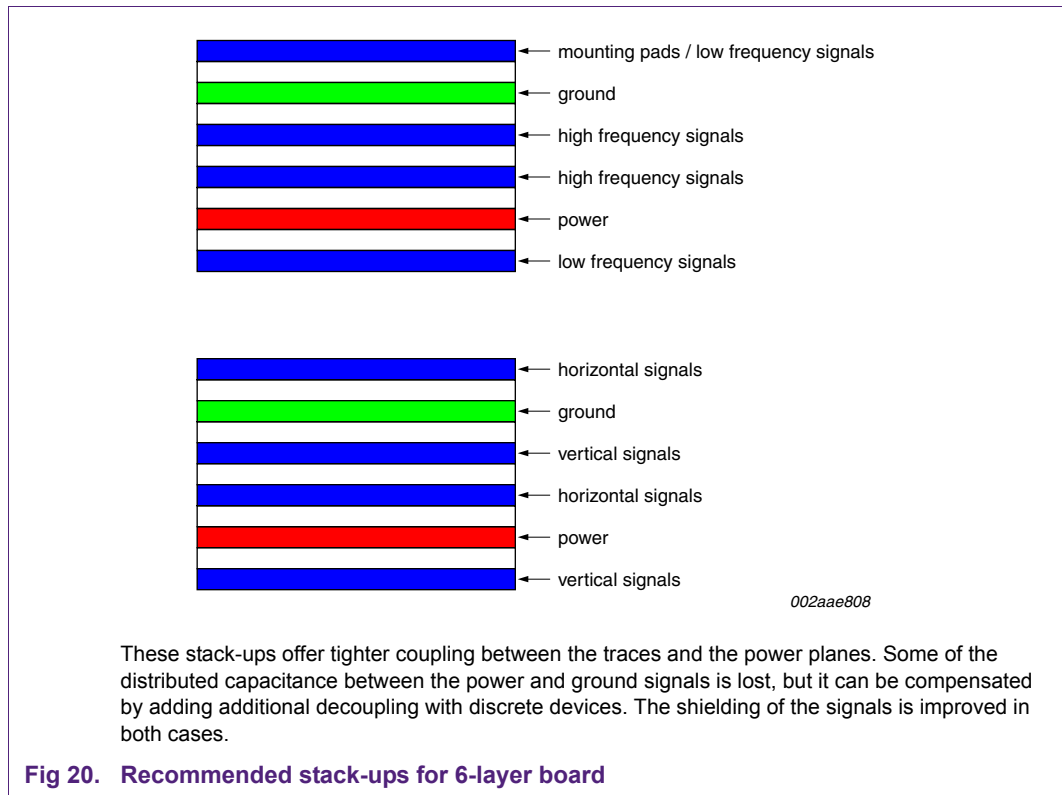
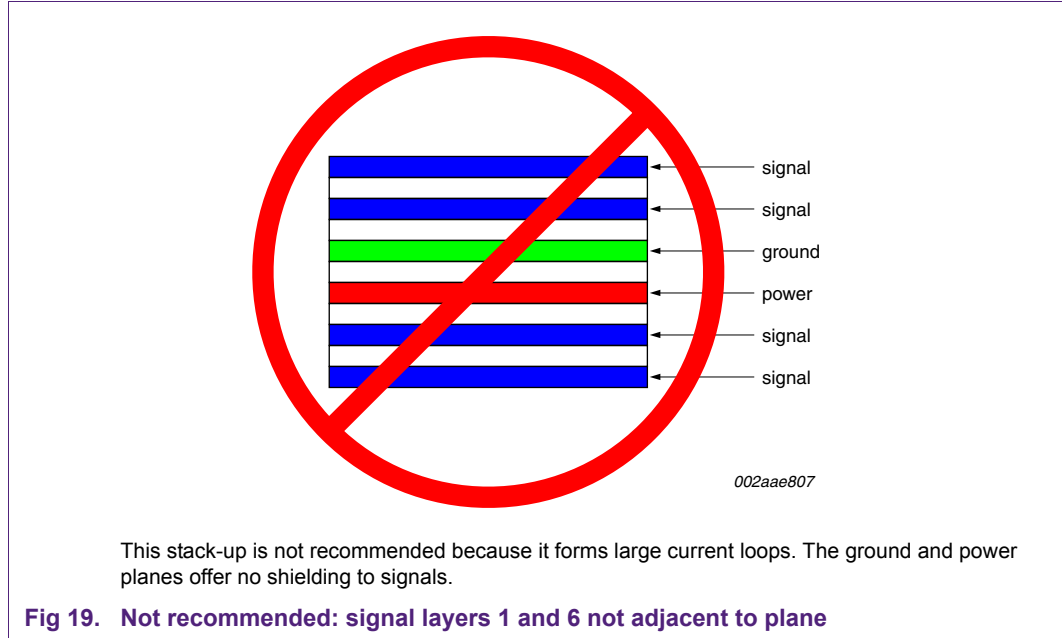


Fig 18. Recommended stack-ups

3.1.1.2 6-layer board stack-up

Do not use the following stack-up since it does not provide any shielding to the signals and signal layer 1 and signal layer 6 are not adjacent to the plane.



3.1.2 Ground planes

A solid ground plane (or ground grid) should exist under all components and traces. Avoid splitting the ground plane and maximize the copper areas in between connectors, pins and traces to allow for the largest ground area possible.

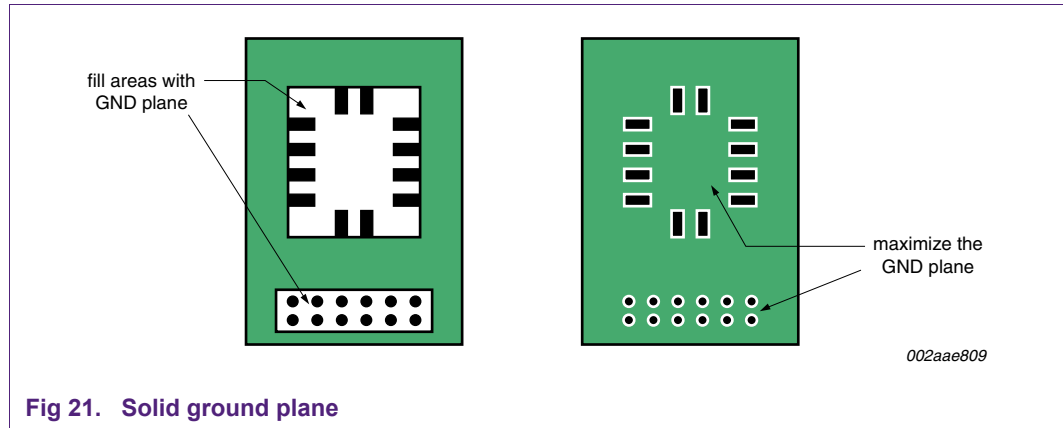


Fig 21. Solid ground plane

If you do split the ground plane and run traces across the split, current will flow in a big loop as there will be no return path near the trace. This current flow will result in radiation and increased ground inductance. If splitting the ground plane is necessary and you must run traces across the split, then you should connect the two planes at one point through a bridge. Route all traces across this bridge point so that they have a return path for the current directly underneath each of the traces, hence creating a very small loop area.

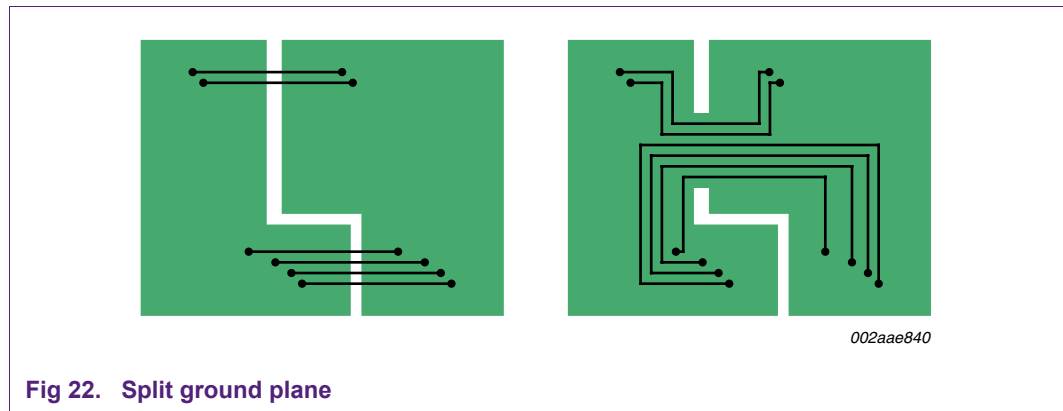


Fig 22. Split ground plane

Always route signal traces and their associated ground returns as close to one another as possible to minimize the loop area enclosed by current flow:

- At low frequencies current flows through the path of least resistance.
- At high frequencies current follows the path of least inductance.

Figure 23 shows a 22 inch (55.88 cm) long trace over ground plane.

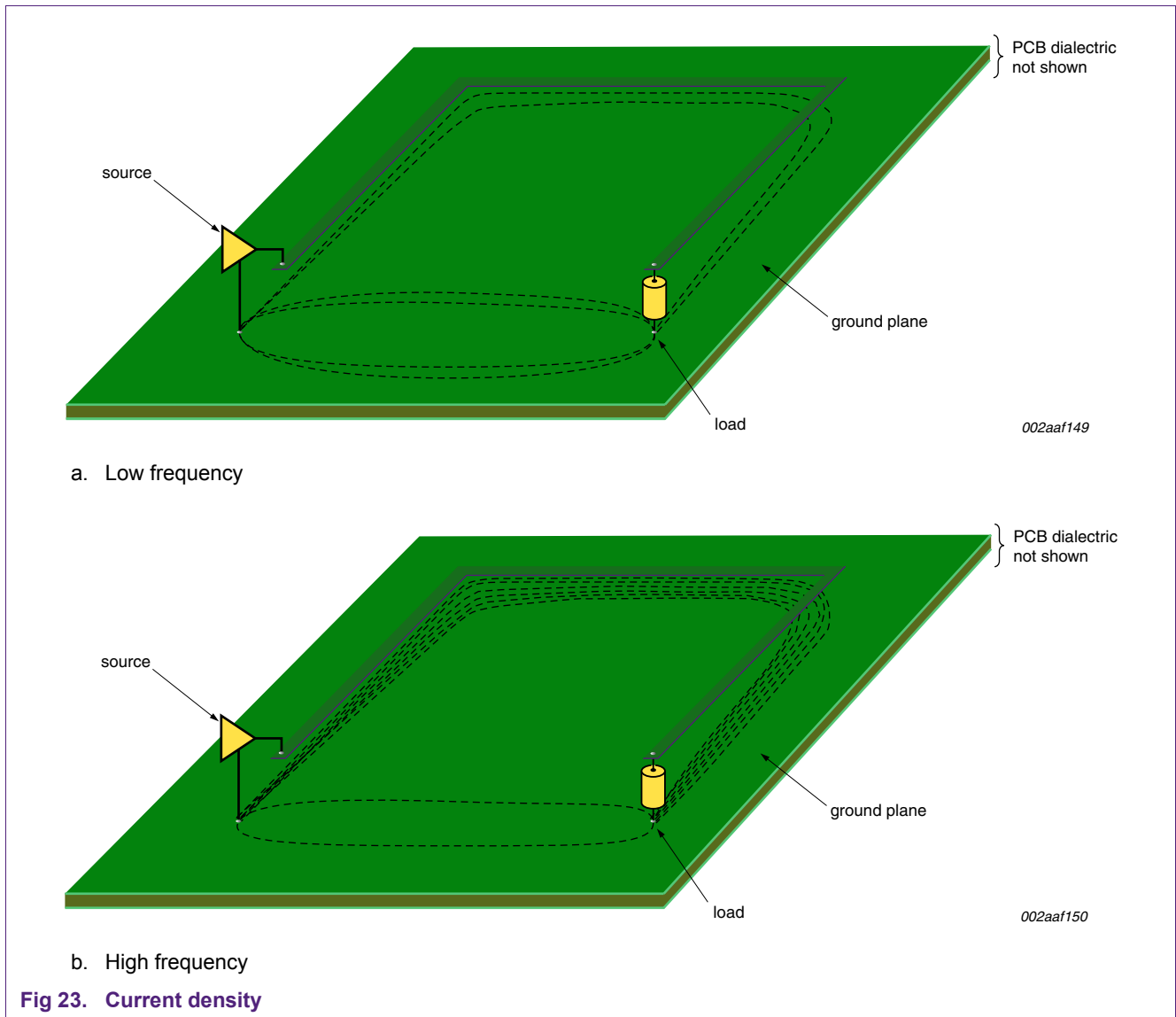


Figure 23a illustrates the low frequency current return path from the load back to the source. At low frequencies, the current follows the path of least resistance.

From Figure 23b you can see that high frequency ground return currents return directly underneath the signal trace. This is the path of lowest impedance (lowest inductance).

A high speed digital return current will stay on its return path and not flow through the analog portion of the ground plane and corrupt analog signal. Therefore, it is unnecessary to split the ground to prevent the digital current from doing something that it will not do.

The preferred method is to use only one ground plane and to partition the PCB as explained in Section 3.1. Analog signals can then be routed only within the analog section of the PCB on any layer, and digital signals can be routed only in the digital section of the

PCB on any layer. Do not route digital signals in the analog section of the board, or visa versa. The ground return paths for analog, digital, or power should be carefully designed so that they do not flow through each other's circuits.

All high frequency signal traces must be on layers adjacent to a plane.

Above 25 MHz, PCBs should have two (or more) ground planes.

3.1.3 Ground grids

In two-layer boards where it may difficult to have a ground plane, a carefully designed ground grid can provide a good performing ground plane.

The spacing between the ground grid is critical. [Figure 24](#) shows the inductance versus grid spacing. For reduced inductance, the ground grid spacing should be under 13 mm.

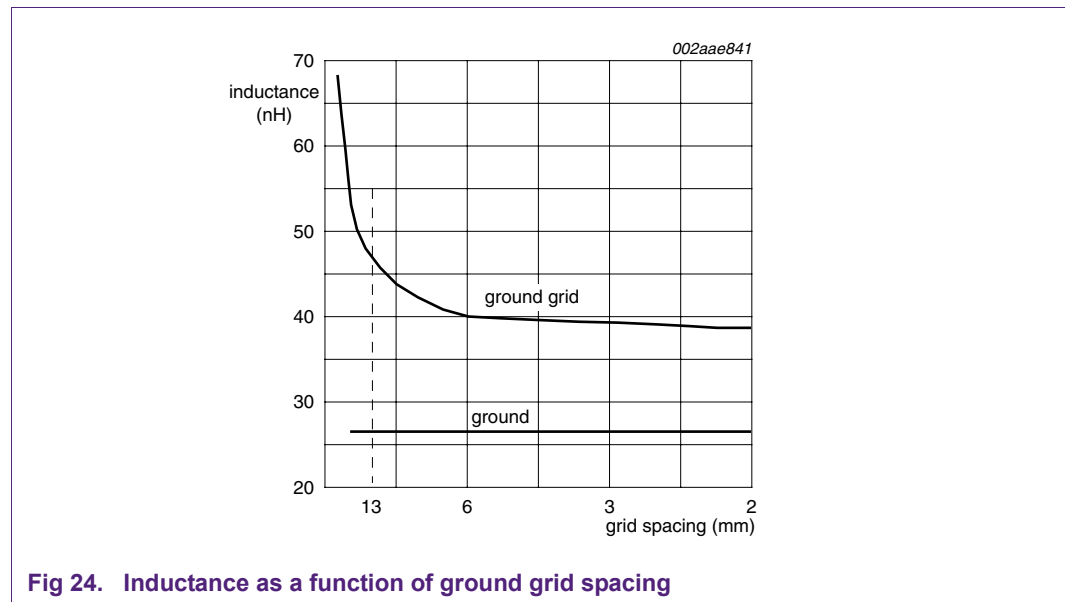


Fig 24. Inductance as a function of ground grid spacing

Ideally, the top and bottom side of the PCB should look like a ground plane with horizontal ground traces on one side, and vertical ground traces on the other side. It is not absolutely necessary that the traces be orthogonal.

The traces should be as wide as possible and the grid is achieved by connecting the vertical and horizontal traces with vias. Decoupling capacitors are recommended at these intersections.

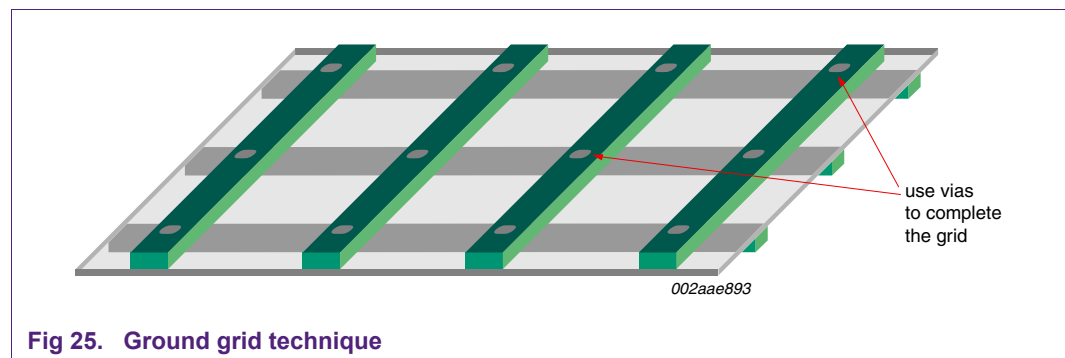
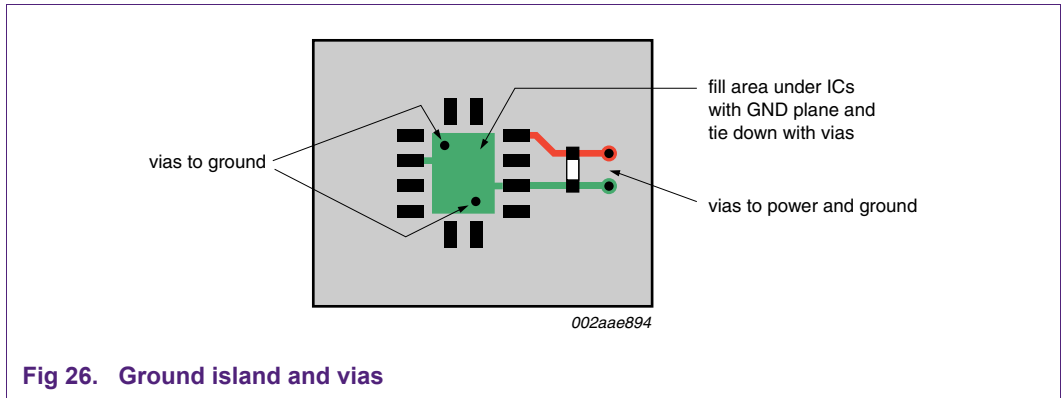


Fig 25. Ground grid technique

3.1.4 Ground points

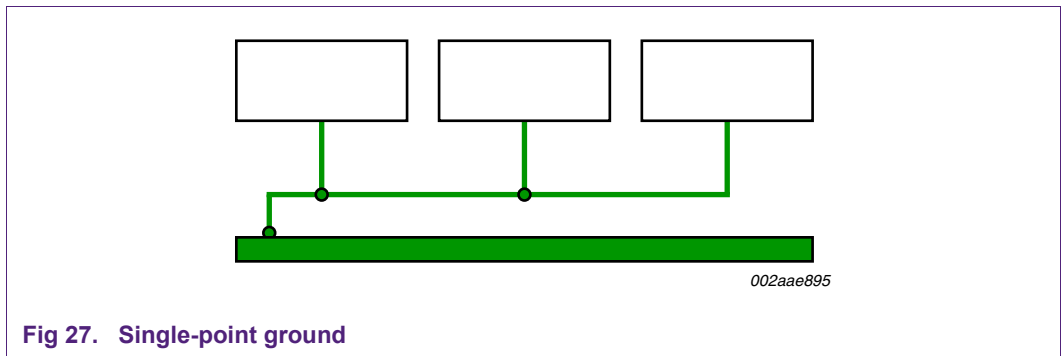
For dual-layer or multi-layer boards, a solid ground ‘island’ should exist underneath all high-speed integrated circuits. The island should be strapped to the ground plane by using vias. A ground via in close proximity of each ground pin should be used.



Keep all ground loops shorter than $\frac{1}{20}$ of a wavelength to maintain low emissions/reception and low-impedance.

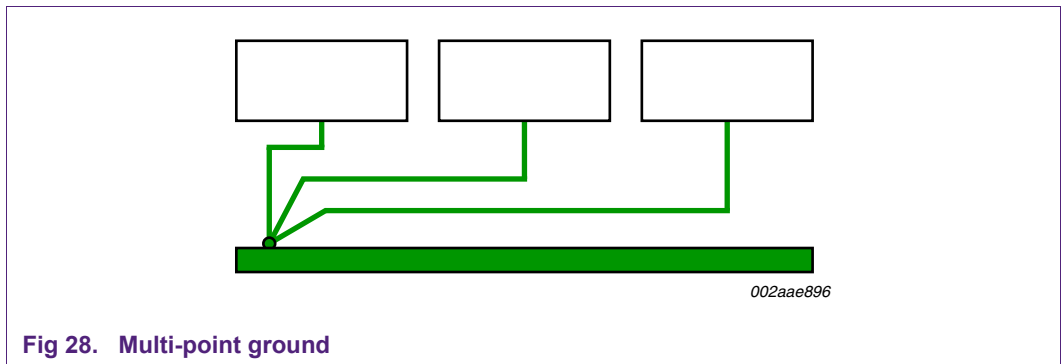
3.1.4.1 Single-point grounding

Single-point grounding schemes work well for low-level and low frequency circuits (below 1 MHz). However, they do not perform well under ESD due to high ground impedance.



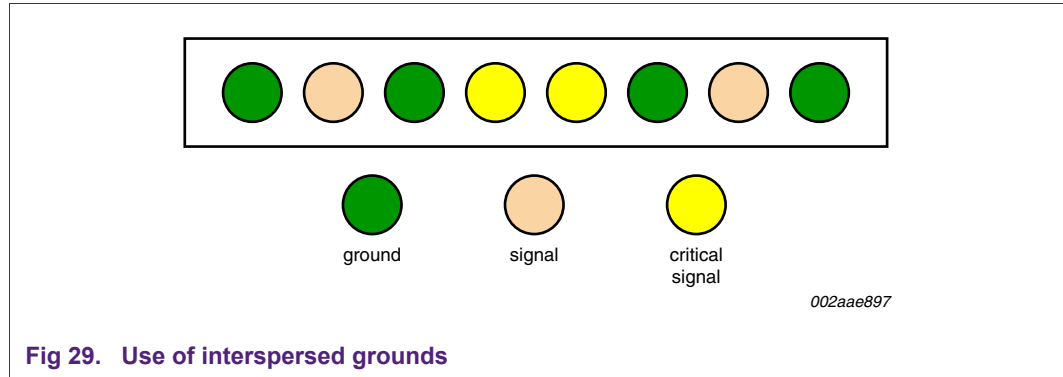
3.1.4.2 Multi-point grounding

A multi-point grounding scheme is preferred to improve ESD performance. The ground impedance is lower, giving the excess current an easy path away from sensitive circuits.



3.1.4.3 Ground pins and interconnect

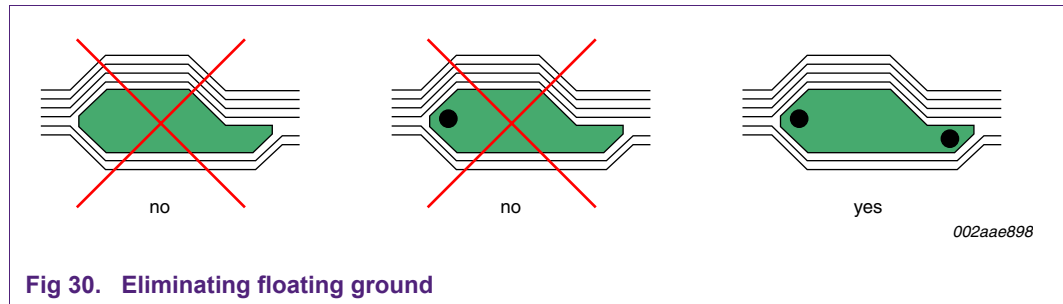
Ground pins should be evenly distributed across all connectors (board-to-board or board-to-cable). The number of connector ground pins should be determined prior to layout and should include the current capabilities of the connector as well as the current carrying capability of the cables connecting the boards.



A minimum of one ground-return track should be routed adjacent to each eight lines of address and data lines to minimize the loop area. You can increase the ratio to 1 : 5 for higher performance. All lines should be kept as short as possible. Route a ground trace adjacent to the Least Significant Bit (LSB) of address since this is usually a very active signal.

3.1.4.4 Metal islands

Avoid any floating metal on the PCB. If it is floating, it can easily store and reflect energy on the circuit. All ground segments with length-to-width ratio greater than 10 : 1 should have, at the minimum, one GND via at each end tying them to the rest of the PCB ground structure.



3.1.4.5 Ground plane area

Extend ground planes as far as possible beyond the boundaries of components and traces and power planes. For high performance circuits, ground planes should extend beyond power planes and any traces by at least 20 times the height of the layer spacing.

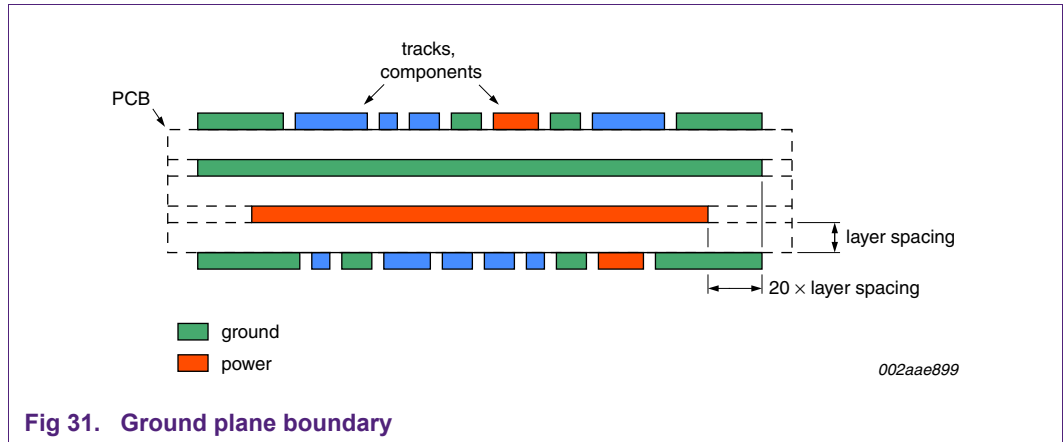


Fig 31. Ground plane boundary

3.1.4.6 Ground star configuration and spark gaps

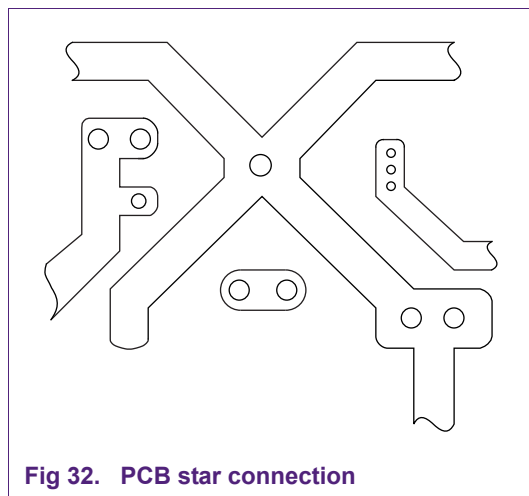


Fig 32. PCB star connection

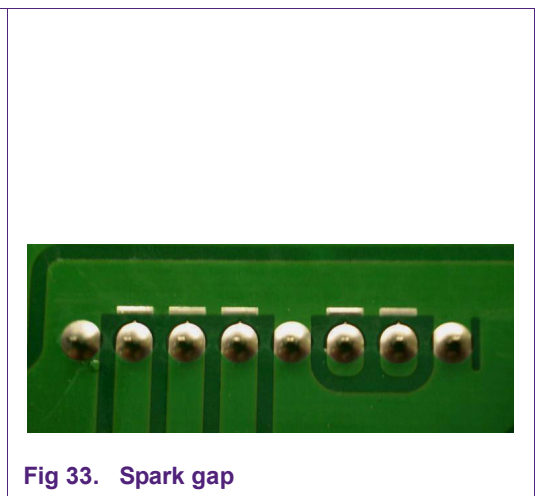


Fig 33. Spark gap

Earthing of connector grounds and such lines as spark gaps requires special consideration. Usually the best physical arrangement for these type of earth is the star configuration—where the earth happens at one place, and the lines go back to that place separately.

It is also possible to supplement ESD protection with PCB spark gaps. The spark gap shown here is manufactured to a common PCB tolerance: it is an $\frac{8}{1000}$ inch gap. The contact area needs to be free of solder resist, in order to function as a spark gap. Breakdown of small spark gaps is approximately $V = (3000 \text{ pd} + 1350)$, where 'p' is pressure in atmospheres, and 'd' is distance in millimeters. This spark gap can be expected to have a peak voltage of about 2000 V to 2500 V.

3.1.4.7 Power distribution

Locate voltage regulators and power supplies close to the power entry point on the PCB. Decouple the power at this point and closely-routed traces should be used for power distribution.

Large bulk capacitors (i.e., electrolytic capacitors) should always be placed in parallel with at least one smaller high frequency ceramic capacitor. This is in addition to the small value decoupling capacitors that should be placed close to the device being decoupled.

When possible, use a power plane to distribute power. For two-layer boards, a star topology or power grid should be used. The 'star' point of power distribution should be the positive terminal of the bulk capacitor, which is the capacitor that stores the largest amount of energy to be delivered.

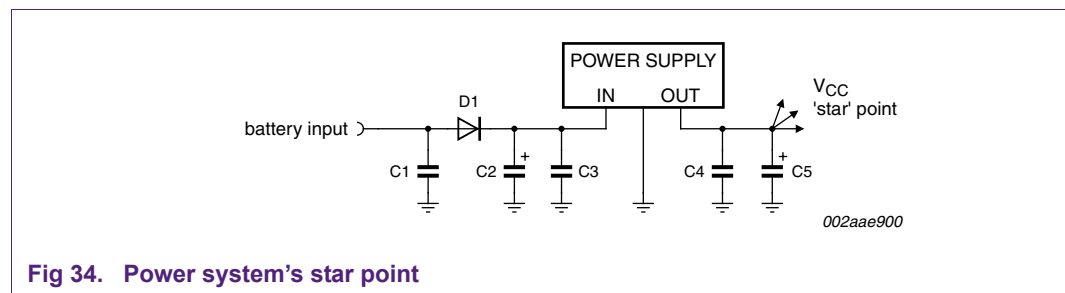


Fig 34. Power system's star point

The value of the bulk capacitor should be carefully calculated so that it is large enough to supply the necessary energy within the required acceptable voltage drop. The value is usually at least ten (10) times greater than the sum of all the values of decoupling capacitors.

3.1.4.8 Decoupling capacitors

To decouple ICs, multi-layer ceramic capacitors are recommended. The values should be $0.1 \mu\text{F}$ and a smaller $0.01 \mu\text{F}$ for ESD performance. The decoupling capacitors should be placed as close as physically possible to the IC's power pin. The further away the capacitor is placed, the higher the inductance in the trace and the less effective the capacitor will be.

The decoupling capacitors are an integral part of the power distribution system, which should provide sufficient current, in time, for the device to function properly. High-peak currents while output switching is a requirement that should be considered when placing capacitors around the ICs.

The minimum number of capacitors is one per IC. Cost and space permitting, adding additional decoupling capacitance is better.

3.1.4.9 Crosstalk and trace spacing

To prevent crosstalk, or coupling, traces carrying high current with fast rise/fall times (for example, under 10 ns) should maintain proper spacing from other traces that run in parallel to them, or have a ground guard trace in between them. This is particularly important if critical signals are close to the offending trace.

Power and ground signals should always be routed in parallel (side-by-side) or on top of each other (on adjacent layers) to minimize loop area, thus reducing loop impedance (Figure 35). This is especially important when constructing a power grid on a two-layer board.

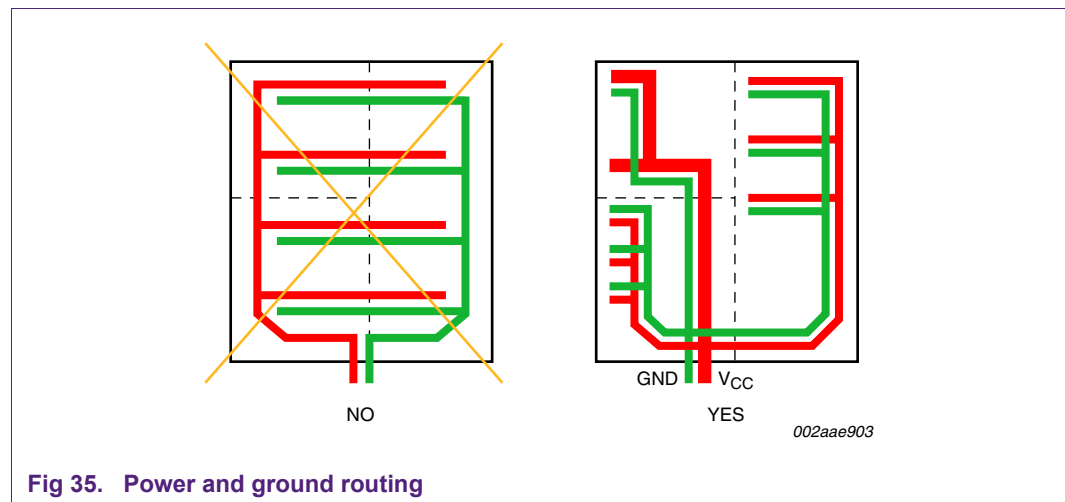


Fig 35. Power and ground routing

Do not route power signals in parallel to high noise signals, as the noise may couple into the power plane. Choose the lowest power and slowest switching speed that meets the circuit requirements to minimize radiated emissions.

Power, ground, and signal traces on the PCB should be kept short and as wide as possible. Give special attention to pass-through signals on the boards as they may need a higher width-to-length ratio (minimum recommended is 1 : 10).

Placing ferrite beads on power traces may provide attenuation of unwanted signals above 1 MHz. When properly sized, these beads can be very effective in damping high-frequency switching transients or parasitic ringing due to line reflections without causing a DC loss.

Remark: Use caution, as using ferrites may impede AC current flow.

Devices such as Zener diodes, MOVs or transorbs should be placed at the power entry point to PCB, as their function is to limit/clip transients and spikes. Assure low-impedance connection to ground.

3.1.4.10 Power supply circuit placement

All Switching Mode Power Supply (SMPS) traces should ideally be routed on one layer of PCB with the SMPS reference plane placed directly on adjacent layer to minimize the loop area. Power switching and high current circuits should be kept separate from digital and low-level analog circuits.

Remark: Ground all the heat-sinks.

3.1.5 Digital circuit considerations

Digital clock connections are a source of EMI. They should be carefully routed on the PCB with all clock traces adjacent to a ground plane. No traces other than ground should run under crystals or any other inherently noisy circuits. Bury clock signals between power and ground planes whenever possible.

All critical nets, such as clocks, data strobes, etc., should be routed physically adjacent to ground traces or ground plane.

3.1.5.1 Oscillators

Keep oscillators and clock-generating ICs away from I/O connectors and close to the chips they service. Always choose the lowest clock frequency and slowest rise and fall time for digital signals that meet system requirements.

Filter (series terminate) the output of clock drivers to slow down their rise/fall times and to reduce ringing (typically 33 Ω to 70 Ω).

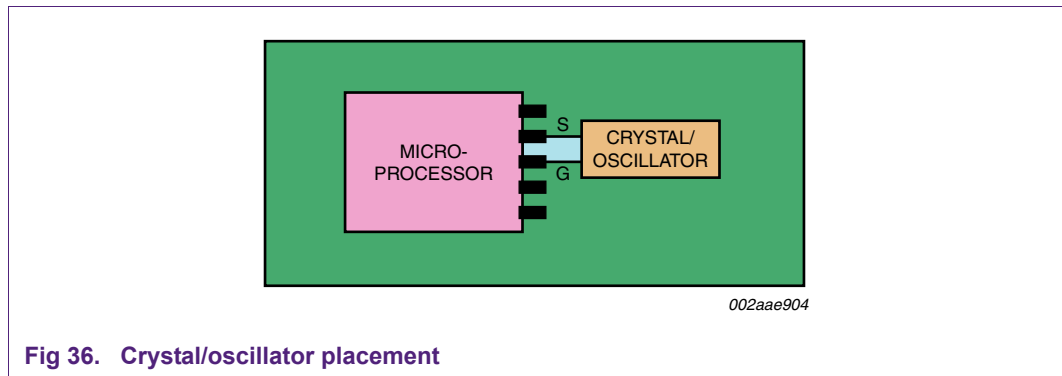


Fig 36. Crystal/oscillator placement

3.1.5.2 Digital bus

Terminators should be used for traces where the rise time is greater than twice the propagation delay.

Route differential signal lines adjacent to one another to take full advantage of magnetic field cancellation. Place ground guard traces on both sides of the entire length of the differential-pair signals.

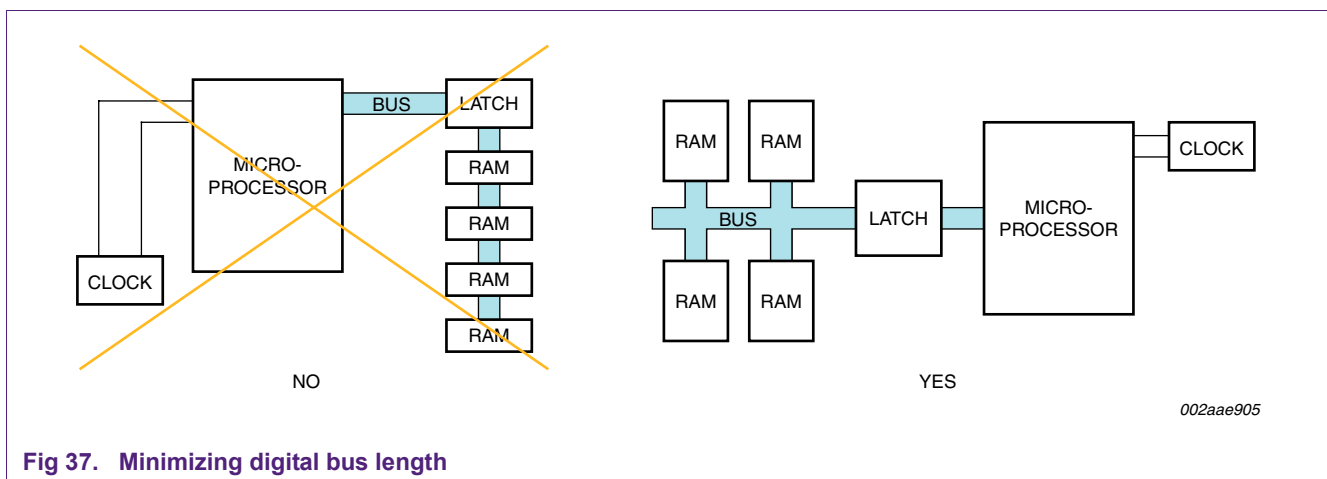


Fig 37. Minimizing digital bus length

High-speed digital signals such as data, address, and control lines of microprocessors should be grouped together and located as far from the I/O connector as possible.

Controlling the rise and fall times, the duty cycle, and the fundamental frequency of switched signals may help to minimize harmonic generation. All unused IC inputs should be terminated to prevent unintentional random switching and noise generation.

Keep high-speed traces, control signals, interrupts and reset signals away from the edge of a PCB.

3.1.6 Analog circuit considerations

Keep all analog or peripheral circuitry as close to the I/O connector as possible, and away from high-speed digital, high current, or power switching circuits. Analog signals should be confined to analog section of PCB only with low-pass filtering always used on all analog inputs.

Ground guard traces should always be routed adjacent to analog signals. Attach the guard traces at both ends with vias to sending and receiving circuits' ground.

If using suppression device across coils of relays and/or solenoids, the suppressor should be placed as close to the coil terminals as possible.

3.1.7 Shielding and cables

All metallic shields of a system should be interconnected and grounded. Each shield should have a low-impedance contact to ground in at least two places in order to prevent its noise potential from coupling to the enclosed object. An ungrounded shield's potential will vary with conditions and location, and therefore the noise coupled to the object inside will vary also.

Using twisted-pair wiring to the load should avoid creation of loop antennas that can radiate magnetic fields.

When routing a wire harness along sheet metal, keep it away from any openings as much as possible. Openings may act as slot antennas.

Use twisted-pair for sensitive low-frequency signals (below 1.0 MHz) and for circuits with impedances less than 1.0 k Ω to provide accurate reference voltages.

Coaxial cable should be used for transmission of RF (above 10 MHz) and where impedance match over a broad frequency range is important (such as video applications).

Circuits generating large, abrupt, current variations should be provided with a separate return lead to the ground in order to reduce transient pickup in other circuits.

Keep wire harness at least 10 cm away from electric field sources such as distributors and magnetic field sources such as alternators, motors, and solenoids. Always try to minimize the length of wire harness to reduce coupling and pickup. The wire harness should be away from ICs and oscillator circuits. Routing over or near ICs should always be avoided ([Figure 38](#)).

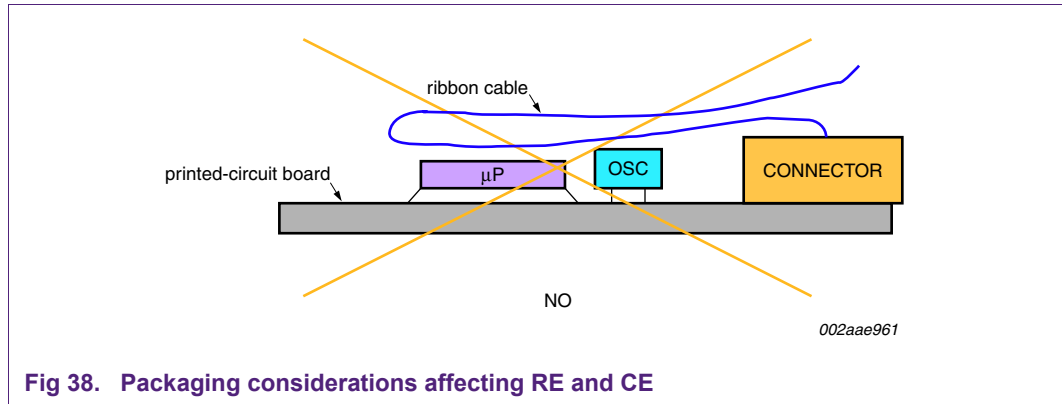


Fig 38. Packaging considerations affecting RE and CE

Critical signals should never be placed on the outside conductors of wire harnesses including shielded ribbon cables.

4. Abbreviations

Table 5. Abbreviations

Acronym	Description
CDM	Charged-Device Model
CE	Conducted Emissions
CRC	Cyclic Redundancy Check
DUT	Device Under Test
EFT	Electrical Fast Transient
EMC	ElectroMagnetic Compatibility
EMI	ElectroMagnetic Interference
ESD	ElectroStatic Discharge
EUT	Equipment Under Test
HBM	Human Body Model
I/O	Input/Output
IC	Integrated Circuit
LED	Light-Emitting Diode
LSB	Least Significant Bit
MM	Machine Model
MOV	Metal Oxide Varistor
PCB	Printed-Circuit Board
RF	Radio Frequency
SMD	Surface Mount Device
SMPS	Switching Mode Power Supply
WDT	WatchDog Timer

5. References

- [1] **Electrostatic Discharge and Electronic Equipment** — Warren Boxleiter; 1989; IEEE Press; ISBN 0-87942-244-0.
- [2] **ESD Immunity in System Designs, System Field Experiences and Effects of PWB Layout** — Douglas Smith and Ed Nakauchi; Electrical Overstress/Electrostatic Discharge Symposium Proceedings 2000; IEEE Catalog No. 00TH8476, Ordering No EOS-22.
- [3] **EMC Design Guide** — 2000, Henry W. Ott
- [4] **EMC Design Issues in Appliances** — November issue 2006; Conformity; Ronald Brewer
- [5] **EMC Guide for Printed Circuit Board** — 2002; Ford Motor Company; Document ES-3U5T-1B257-AA
- [6] **The Designer's Guide to Electromagnetic Compatibility** — 2005; Daryl Gerke and Bill Kimmel; published by Kimmel Gerke Associates, LTD.
- [7] **Chip Monolithic Ceramic Capacitors** — September 2009; Murata Manufacturing Co.; Document Cat.No.C02E-15
- [8] **Designing PCBs for ESD and EMI Compliance** — 2009; Airborne Electronics
- [9] **IEEE 'Design Tips': Resistive vs. Inductive Return Current Paths** — 2008; Bruce Archambeault, Ph.D., IEEE Fellow.

6. Legal information

6.1 Definitions

Draft — The document is a draft version only. The content is still under internal review and subject to formal approval, which may result in modifications or additions. NXP Semiconductors does not give any representations or warranties as to the accuracy or completeness of information included herein and shall have no liability for the consequences of use of such information.

6.2 Disclaimers

General — Information in this document is believed to be accurate and reliable. However, NXP Semiconductors does not give any representations or warranties, expressed or implied, as to the accuracy or completeness of such information and shall have no liability for the consequences of use of such information.

Right to make changes — NXP Semiconductors reserves the right to make changes to information published in this document, including without limitation specifications and product descriptions, at any time and without notice. This document supersedes and replaces all information supplied prior to the publication hereof.

Suitability for use — NXP Semiconductors products are not designed, authorized or warranted to be suitable for use in medical, military, aircraft, space or life support equipment, nor in applications where failure or malfunction of an NXP Semiconductors product can reasonably be expected to result in personal injury, death or severe property or environmental damage. NXP Semiconductors accepts no liability for inclusion and/or use of NXP Semiconductors products in such equipment or applications and therefore such inclusion and/or use is at the customer's own risk.

Applications — Applications that are described herein for any of these products are for illustrative purposes only. NXP Semiconductors makes no representation or warranty that such applications will be suitable for the specified use without further testing or modification.

Export control — This document as well as the item(s) described herein may be subject to export control regulations. Export might require a prior authorization from national authorities.

6.3 Trademarks

Notice: All referenced brands, product names, service names and trademarks are the property of their respective owners.

7. Contents

1	Introduction	3	3.1.7	Shielding and cables	26
2	The basics	3	4	Abbreviations	27
2.1	The ESD pulse	3	5	References	28
2.1.1	Characteristics of the ESD pulse— the IEC model	3	6	Legal information	29
2.1.2	Putting the ESD problem into perspective	5	6.1	Definitions	29
2.2	Effects and failure modes	7	6.2	Disclaimers	29
2.2.1	Sources, paths and receptors	7	6.3	Trademarks	29
2.2.2	Unintentional antennas	7	7	Contents	30
2.3	An overview of R, L, C and D components	8			
2.3.1	Resistor (R) component characteristics	8			
2.3.2	Inductor (L) component characteristics	8			
2.3.3	Capacitor (C) component characteristics	9			
2.3.4	Diodes (D)	11			
2.3.5	Filters	11			
2.3.6	Ground	11			
2.4	ESD mitigation	12			
2.4.1	Hardware ESD management strategy	12			
2.4.2	Software ESD management strategy	12			
2.4.2.1	Refresh	13			
2.4.2.2	Checking and restoring	13			
2.4.2.3	Unused interrupts and registers	13			
2.4.2.4	Watchdog timers	13			
3	A practical approach to schematic and PCB design	14			
3.1	Board structure - functional grouping	14			
3.1.1	Multi-layer boards recommended stack-ups	15			
3.1.1.1	4-layer board stack-up	15			
3.1.1.2	6-layer board stack-up	16			
3.1.2	Ground planes	17			
3.1.3	Ground grids	19			
3.1.4	Ground points	20			
3.1.4.1	Single-point grounding	20			
3.1.4.2	Multi-point grounding	20			
3.1.4.3	Ground pins and interconnect	21			
3.1.4.4	Metal islands	21			
3.1.4.5	Ground plane area	22			
3.1.4.6	Ground star configuration and spark gaps	22			
3.1.4.7	Power distribution	23			
3.1.4.8	Decoupling capacitors	23			
3.1.4.9	Crosstalk and trace spacing	24			
3.1.4.10	Power supply circuit placement	24			
3.1.5	Digital circuit considerations	25			
3.1.5.1	Oscillators	25			
3.1.5.2	Digital bus	25			
3.1.6	Analog circuit considerations	26			

Please be aware that important notices concerning this document and the product(s) described herein, have been included in section 'Legal information'.



© NXP B.V. 2010.

All rights reserved.

For more information, please visit: <http://www.nxp.com>

For sales office addresses, please send an email to: salesaddresses@nxp.com

Date of release: 19 January 2010

Document identifier: AN10897_2