

PCB Design Guidelines For Reduced EMI

SZZA009
November 1999



IMPORTANT NOTICE

Texas Instruments and its subsidiaries (TI) reserve the right to make changes to their products or to discontinue any product or service without notice, and advise customers to obtain the latest version of relevant information to verify, before placing orders, that information being relied on is current and complete. All products are sold subject to the terms and conditions of sale supplied at the time of order acknowledgement, including those pertaining to warranty, patent infringement, and limitation of liability.

TI warrants performance of its semiconductor products to the specifications applicable at the time of sale in accordance with TI's standard warranty. Testing and other quality control techniques are utilized to the extent TI deems necessary to support this warranty. Specific testing of all parameters of each device is not necessarily performed, except those mandated by government requirements.

CERTAIN APPLICATIONS USING SEMICONDUCTOR PRODUCTS MAY INVOLVE POTENTIAL RISKS OF DEATH, PERSONAL INJURY, OR SEVERE PROPERTY OR ENVIRONMENTAL DAMAGE ("CRITICAL APPLICATIONS"). TI SEMICONDUCTOR PRODUCTS ARE NOT DESIGNED, AUTHORIZED, OR WARRANTED TO BE SUITABLE FOR USE IN LIFE-SUPPORT DEVICES OR SYSTEMS OR OTHER CRITICAL APPLICATIONS. INCLUSION OF TI PRODUCTS IN SUCH APPLICATIONS IS UNDERSTOOD TO BE FULLY AT THE CUSTOMER'S RISK.

In order to minimize risks associated with the customer's applications, adequate design and operating safeguards must be provided by the customer to minimize inherent or procedural hazards.

TI assumes no liability for applications assistance or customer product design. TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right of TI covering or relating to any combination, machine, or process in which such semiconductor products or services might be or are used. TI's publication of information regarding any third party's products or services does not constitute TI's approval, warranty or endorsement thereof.

Contents

	<i>Title</i>	<i>Page</i>
ABSTRACT		1
1 Background		1
1.1 RF Sources		1
1.2 Surface-Mount Devices vs Through-Hole Components		1
1.3 Static Pins vs Active Pins vs Inputs		1
1.4 Basic Loops		2
1.4.1 Proportionality of Loops and Dipoles		3
1.5 Differential vs Common Mode		3
2 Board Layout		4
2.1 Grounds and Power		4
2.1.1 Inductance		4
2.1.2 Two-Layer vs Four-Layer Boards		4
2.1.3 Microcomputer Grounds in One- and Two-Layer Designs		5
2.1.4 Signal Return Grounds		5
2.1.5 Analog vs Digital vs High Power		5
2.1.6 Analog Power-Supply Pins and Analog Reference Voltages		6
2.1.7 Power Plane Do's and Dont's for Four-Layer Boards		6
2.2 Power Distribution for Two-Layer Boards		7
2.2.1 Single-Point vs Multipoint Distribution		7
2.2.2 Star Distribution		7
2.2.3 Gridding to Create Planes		7
2.2.4 Bypassing and Ferrite Beads		9
2.2.5 Keeping Noise Close to the Chip		11
2.3 Board Zoning		12
2.4 Signal Traces		13
2.4.1 Capacitive and Inductive Crosstalk		13
2.4.2 Antenna Factor Length Rules		13
2.4.3 Series Termination, Transmission Lines		13
2.4.4 Impedance Matching at Inputs		14
2.5 Cables and Connectors		14
2.5.1 Differential-Mode and Common-Mode Noise		14
2.5.2 Crosstalk Model		14
2.5.3 Number of Returns		15
2.5.4 I/O Recommendations for Off-PCB Signals		15
2.5.5 Keeping Noise and Electrostatic Discharge (ESD) Out		15
2.6 Other Layout Issues		15
2.6.1 Front-Panel PCB with Keypad and Display in Automotive and Consumer Applications		15
2.6.2 Layout for Susceptibility		16
2.6.3 Autorouters		16
3 Shielding		17
3.1 How It Works		17
3.2 Grounding the Shield		17
3.3 Cables and Bypassing to the Shield		17
3.4 Slot Antennas: Cooling Slots and Seams		18
4 Summary		18
5 Literature		19

List of Illustrations

<i>Figure</i>	<i>Title</i>	<i>Page</i>
1	Signals Below 50 kHz Are Not EMI Concerns	2
2	Examples of Loops	3
3	Differential vs Common-Mode Noise	4
4	Microcomputer Ground	5
5	Layout Considerations	6
6	Power Distribution	7
7	Gridding Power Traces on Two-Layer Boards	8
8	Gridding of Ground Fills and Traces to Form a Ground Plane	9
9	Ferrite-Bead Placement Closest to the Noise Source	10
10	Board Zoning	12
11	MOS Buffer Simplified Schematic	14
12	Front-Panel Gridding to Form Two Ground Planes	16
13	Mounting Filter Capacitors for External I/Os	18

List of Tables

<i>Table</i>	<i>Title</i>	<i>Page</i>
1	Termination Characteristics	13

ABSTRACT

General layout guidelines for printed circuit boards (PCB), which exist in relatively obscure documents, are summarized. Some guidelines apply specifically to microcontrollers; however, the guidelines are intended to be general, and apply to virtually all modern CMOS integrated circuits. This document covers most known and published layout techniques as applied in a low-noise, unshielded environment. Efforts have been made to target two-layer boards, and the maximum acceptable noise level is assumed to be 30 dB, or greater, more stringent than FCC Part 15. This level seems to be the upper limit of acceptable noise in European and U.S. automotive markets.

This document does not always explain the why's of a given technique because it is intended only as a reference document, not a teaching aid. The reader is cautioned against making the assumption that although on a prior design a given technique was not applied and the unit had acceptable performance, that the technique is not useful. Over time, as IC devices increase in speed and density, every method to isolate and reduce noise will be required.

1 Background

1.1 RF Sources

Design guidelines to be discussed concern radio-frequency (RF) noise from the microcomputer. This noise is generated inside the device and is coupled out in many different possible ways. The noise is present on all outputs, inputs, power supply, and ground at all times. Potentially, every pin on the microcomputer can be a problem.

The biggest problem is noise from the integrated-circuit (IC) input/output (I/O) pins. Because the area covered by traces connected to them on the PCB form a large antenna. These pins also connect to both internal and external cables. The noise from clock switching within the IC appears as "glitches" on a static output. The glitch is caused by the common impedance of the output pin and the clock drivers, that is, the shared pins that supply each power and ground. The synchronous nature of most devices causes all current-switching events to occur at the same time, making a large noise spike containing RF energy.

The second most-important contributor is the power-supply system, which includes the voltage regulation and the bypassing capacitors at both the regulator and at the microcomputer. These circuits are the source of all the RF energy in the system, as they feed the clocked circuits inside the IC with the current required for switching.

The third noise source is the oscillator circuit, where the oscillator swings rail to rail. In addition to the fundamental frequency, harmonics are introduced on the output side because the output buffer is digital, which squares the sine wave. Also, any noise caused by internal operations, such as the clock buffers, appears on the output. If proper separation is maintained between the crystal and its tank circuits from other components and traces on the PCB, and the loop areas are kept small, there should be no problems with this noise source. But it has been shown that if ICs or passive components, such as the main V_{Batt} series inductor, are placed close to the crystal, harmonics of the crystal can couple and propagate.

The primary focus in this application report is on the first and second previously described noise sources. The way to deal with the third noise source has been addressed. Also, critical information is disclosed on board zoning (floor planning) and shielding.

1.2 Surface-Mount Devices vs Through-Hole Components

Surface-mount devices (SMD) are better than leaded devices in dealing with RF energy because of the reduced inductances and closer component placements available. The latter is possible due to the reduced physical dimensions of SMDs. This is critical to two-layer board design, where maximum effectiveness from noise-control components is needed. Generally, leaded capacitors all go self-resonant (become more inductive than capacitive) at about 80 MHz. Because noise above 80 MHz needs to be controlled, serious questions should be asked if a design is to be executed only with through-hole components.

1.3 Static Pins vs Active Pins vs Inputs

As mentioned previously, all lines have noise from the processor, to some degree. The total noise from a pin depends on how much noise the microcomputer provides it and its function in the system. For example, an output pin has the noise from the microcomputer's power rails and the noise capacitively coupled from adjacent pins and the substrate. If the pin's function is the system clock, that too is noise. Even if the pin were static at a one or zero level, one would still have to contend with noise from inside the chip.

In the case of an I/O pin in the input mode, the capacitance of the unused output transistors transfers noise from both power rails to the pin. The amount of noise is based on the impedance of whatever is connected to the pin. The higher the impedance, the more noise comes out of the microcomputer. That is why unused inputs should be tied to the lowest-impedance rail: ground, by direct short, if possible.

With respect to switching output signals, basically, only worry about signals that make an edge transition at a rate greater than 50 kHz (see Figure 1). If a pin changes its state at a rate of less than once per 100 instructions, this is acceptable because the contribution from switching is negligible. If the pin toggles, and toggles back on the next instruction, and remains static for 100 instructions, it, too, is acceptable because it contains the same amount of energy as in the previous example.

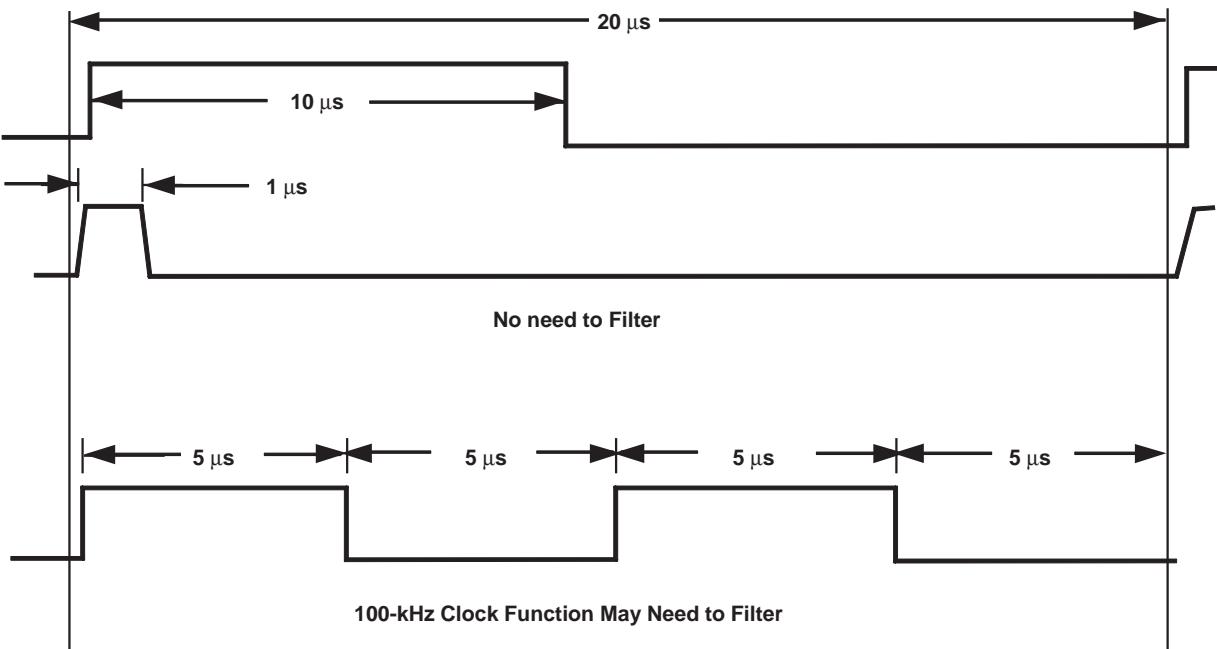


Figure 1. Signals Below 50 kHz Are Not EMI Concerns

1.4 Basic Loops

Every edge transition that is sent from the microcomputer to another chip is a current pulse. The current pulse goes to the receiving device, exits through that device's ground pin, then returns via the ground traces, to the ground pin of the microcomputer (see Figure 2). The pulse does not exit the ground lead of the receiving device and return to the battery, but travels in a loop to where it originates. Loops exist everywhere. Any noise voltage and its associated current travels the path(s) of lowest impedance back to the place where it was generated. This is a very powerful concept, because it allows you to mitigate noise propagation by controlling the shape and impedance of the return path.

A loop can be a signal and its return path, the bypassing loop between power and ground and the active devices inside the microcomputer, the oscillator crystal and its driver in the microcomputer, as well as the loop from the power supply or voltage regulator to the bypassing capacitors. Other more difficult loops are actually ambient field loops. For example, the crystal itself radiates energy that can be coupled into a wire running nearby. Then, the wire contains noise that tries to get back to the crystal loop. That may involve a very long and convoluted path, which serves as another antenna for noise from the crystal.

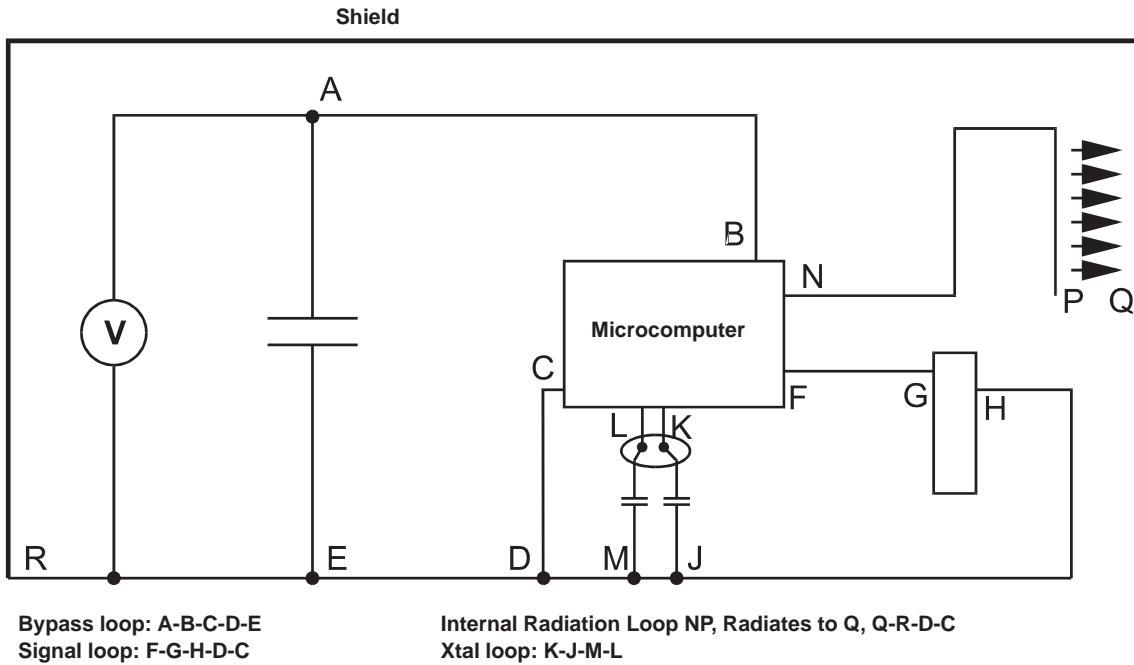


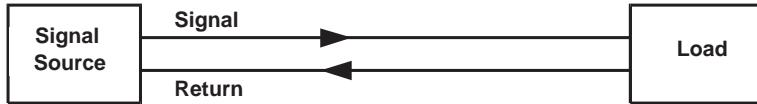
Figure 2. Examples of Loops

1.4.1 Proportionality of Loops and Dipoles

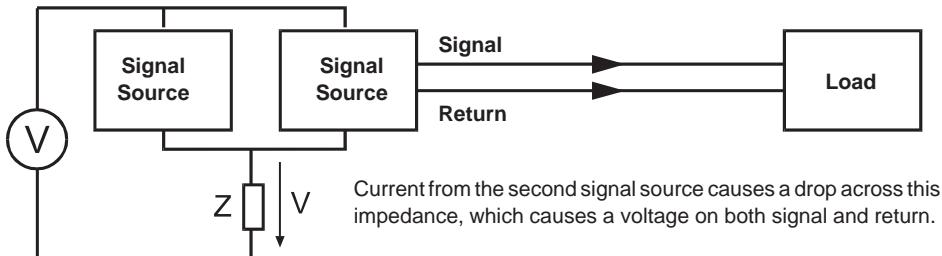
Loops and dipoles are antennas. Their radiating efficiency increases up to $1/4$ wavelength (λ) of the frequency of interest. Geometrically, that means, in the case of a loop, that the larger the laid-out area of the loop, the stronger the radiation until one or both legs of the loop reach $1/4$ wavelength. In the dipole, the longer the antenna, the more radiation, until the length of the antenna reaches $1/4$ wavelength. At 1 MHz, $1/4 \lambda = 75$ m. At 300 MHz, $1/4 \lambda = 25$ cm, or about 10 inches.

1.5 Differential vs Common Mode

Differential-mode noise is the noise of a signal as it travels down its trace to the receiving device, then back along the return path (see Figure 3). **There is a differential voltage between the two wires.** This is the type of noise that every signal must make in order to do its job. Make sure there is no more noise than needed to get the job done, in terms of both frequency content (rise and fall times) and the magnitude of the current. **In common mode, a voltage travels down both the signal and return lines at the same time.** There is no differential between the signal and its return. The voltage is caused by an impedance that is common to both the signal and the return. **Common impedance noise is the most common source of noise in most microcomputer-based systems that are not using external memories.**



Differential-mode noise is the “noise” voltage when a signal travels to its lead and returns. An output switching is an example of differential-mode noise.



Common-mode noise is the noise voltage that travels down both the signal and return caused by a voltage drop across a shared impedance. Ground bounce on outputs is an example of common-mode noise.

Figure 3. Differential-Mode vs Common-Mode Noise

2 Board Layout

2.1 Grounds and Power

The only non-dc current that should flow in the power routing of the PCB is the current required to replenish the bypassing capacitor. High-frequency current used inside the microcomputer that is switched on the input clock edges should come from the bypassing capacitors, not from the power supply.

2.1.1 Inductance

Inductance increases with increasing length, and decreases (at a slower rate) with increasing width of the conductor. In power routing, the inductance makes voltage drops that radiate and propagate.

Because it is not desirable for any trace to radiate RF energy, any trace carrying RF energy should be as low an inductance as possible:

- On a two-layer board, for both power and ground, the length-to-width ratio should not exceed 3:1 for any traces between the IC and the voltage source.
- Power and ground should be run directly over each other, which reduces impedance and minimizes loop area.

2.1.2 Two-Layer vs Four-Layer Boards



A two-layer board can achieve 95% of the effectiveness of a four-layer board by emulating what makes a four-layer board better:

- Make an extra effort to route ground underneath power.
- Grid power and ground, but be careful not to create unneeded common impedance connections or to violate an intended isolation, such as between high-power and digital grounds. See section 2.2.3, *Gridding to Create Planes*.
- Route returns for direct connections to the processor I/Os directly under the signal trace. Gridding is a space-effective way of doing this. See section 2.2.3, *Gridding to Create Planes*.
- Under the microcomputer, build a solid plane for ground that bypassing components and the oscillator loop can be tied to. Tie this ground to the ground pin and the power-supply bypass capacitor. This is called a microcomputer ground, which is discussed in section 2.1.3.

2.1.3 Microcomputer Grounds in One- and Two-Layer Designs

A microcomputer ground is a ground area on the bottom layer underneath the microcomputer that becomes a ground island for the noise made by the microcomputer. This area should extend about 1/4 inch outside the outline of the device and tie to the microprocessor ground. Ground connections for the power-supply bypassing capacitors and any bypassing capacitors on the pins also should tie to this ground. Additionally, the ground area should extend out and around the through holes for the oscillator leads, and the bypass capacitors tied in to provide the smallest possible loop area when viewed from the top. See Figure 4 for an example.

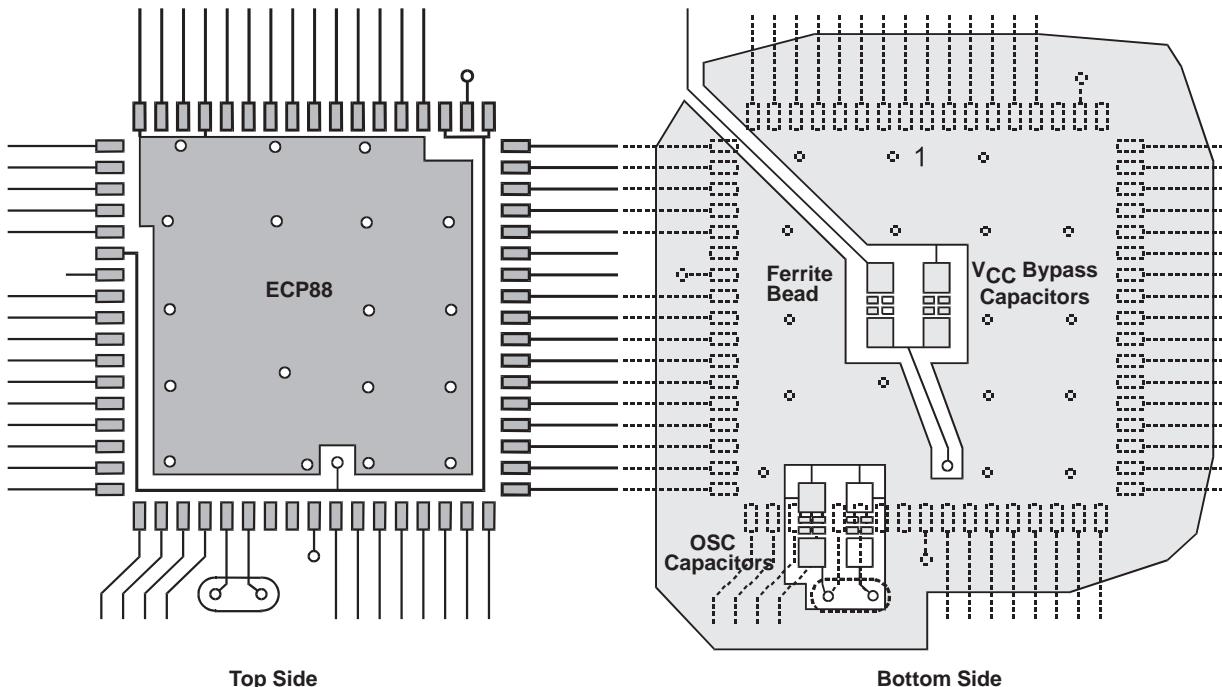


Figure 4. Microcomputer Ground

The topside traces are shown in dotted line form on the bottom side diagram for alignment purposes. Notice how the oscillator capacitors lay back over the traces between the device and the crystal. This eliminates loop area. The same is true for the placement of the ferrite bead and V_{CC} bypass capacitor, being centrally located with the main power lead running almost directly under the lead finger for the ground.

2.1.4 Signal Return Grounds

As mentioned previously in section 1.4, a loop is made by a signal, and the ground return path from the receiver device back to the signal source. Signal return paths present the most difficult design problem in PCB layout.

It would be difficult to route a ground return underneath each trace connected to a signal pin on the microcomputer. But, this is exactly what the ground plane of a four-layer board does. No matter where the traces run, there is always a ground return path running underneath it.

The closest approximation to having a ground plane in a two-layer board comes from gridding the ground, as described in section 2.2.3. As stated previously, radiation from the signal traces is the primary concern. Reducing the loop area by routing the return for the signal underneath the signal trace is most effective way of dealing with this problem. Therefore, creating a ground grid is the most important thing to do (after floor planning) in laying out the PCB.

2.1.5 Analog vs Digital vs High Power

Digital ground and power carry the RF energy that needs to be contained, so it is best to isolate it from any other power and ground, either analog, high power, or other unrelated trace. If noise from the microcomputer or any other circuit gets on an isolated ground, it can be returned by careful placement of a small RF capacitor in the 470 – 1000 pF range. Choosing the location of the capacitor is by trial and error, and is best done in the screen room.

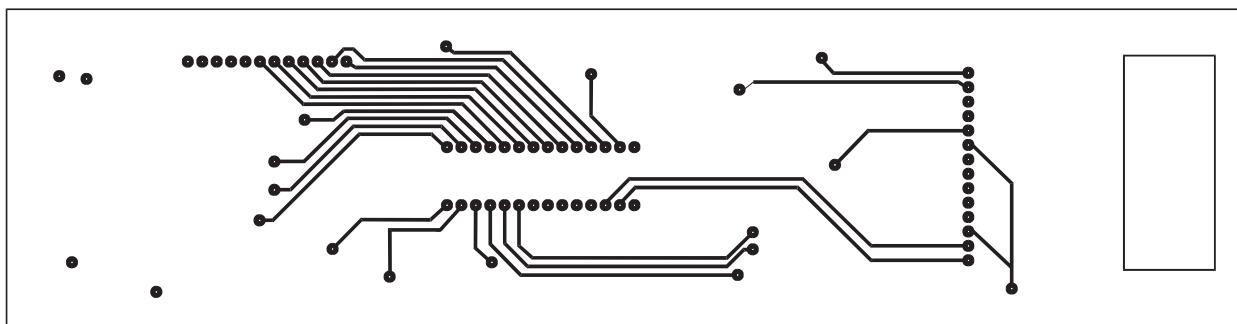
2.1.6 Analog Power-Supply Pins and Analog Reference Voltages

The reference voltage of an analog-to-digital (A/D) converter integrated into a microcomputer does supply a very small amount of clocked current; however, it is not enough to be concerned about from a noise-emissions standpoint. Most applications have the analog V_{ss}/V_{cc} tied to the digital V_{ss1}/V_{cc1} pins, which does not change significantly the noise characteristics of the A/D nor the radiated emissions, provided the power distribution is built to guidelines in sections 2.1.1 through 2.1.4.

2.1.7 Power Plane Do's and Don'ts for Four-Layer Boards

The reasons for reduced noise from four-layer boards were mentioned in section 2.1.2. The following guidelines should maintain the advantages gained in the four-layer board.

- Pay utmost attention to how the holes and cutouts in the planes are done. They break up the plane and, therefore, cause increases in loop areas (see A and B in Figure 5).
- Avoid buried traces in the ground plane. If you have to use them, put them in the +V plane.
- When making through holes for 100-mil-center-spacing leads in the plane, place a small trace between each pin. Breaking up the plane with a row of holes is much better than having a long slot (see C and D in Figure 5).
- When splitting up the ground plane to make, for example, a digital and power ground, make sure that the signals connected to the microcomputer are still located entirely over the digital ground. Extending signal traces beyond the power ground hurts because the power ground does not work to reduce the loop area for digital noise signals.



- A POOR – Buried trace cuts ground plane into two parts
B BETTER – Buried trace around the perimeter
Best solution is no trace at all in the ground plane
- C POOR – Slot formed by 100-mil spacing cuts up ground plane and focuses slot antenna radiation into that connection
D BETTER – Ground plane extends between 100-mil centers

Figure 5. Layout Considerations

2.2 Power Distribution for Two-Layer Boards

2.2.1 Single-Point vs Multipoint Distribution

In a true single-point power-distribution system, each active component has its own separate power and ground, and these traces would remain separate until they meet at a single reference point. In multipoint systems, the connections are made in a daisy-chain fashion, so there are multiple 0-V reference points. It is clear that multipoint systems have the potential for common impedance coupling. While implementing a single-point system may be impossible, a combination of single point for devices generating RF and multipoint for everything else serves to reduce noise. The best scheme possible has a single point that ties together the regulator ground, microcomputer ground, battery negative, and chassis or shield (see Figure 6).

2.2.2 Star Distribution

Star distribution is much like single point. It looks like all points reference the same fixed point, which is located centrally, by about the same length of traces. Additionally, that same reference point may be attached, via a large single trace, to its source, which is not centrally located. Therefore, the major differences to single point are:

- The single reference point on a star can be a longer trace, instead of a point
- The point where the separate traces begin is near the center of the board, and each trace goes in its own direction, with the resulting trace length equal to that of all the others.
- The star is best applied to something like a system clock in a high-speed computer board. The signal originates on the edge connector and proceeds to the center of the board, where it then splits and goes to each place it is needed. Since it effectively originates at the center of the board, the delay in the signal from one area of the board to another is minimal. The name star sometimes is used to refer to single point, making the above clarification necessary.

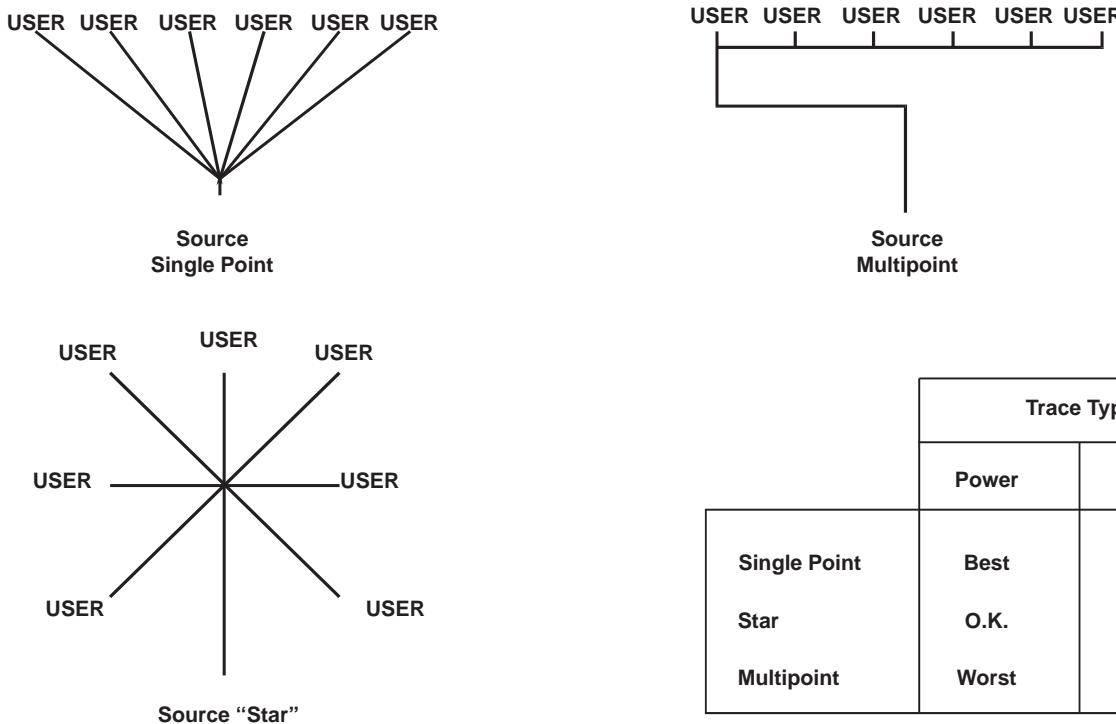


Figure 6. Power Distribution

2.2.3 Gridding to Create Planes

Gridding is the most critical design technique for two-layer boards. Much like a power utility grid, gridding is a network of orthogonal connections between traces carrying ground. It effectively creates a ground plane, which provides the same noise reduction as on four-layer boards. It serves two purposes:

- Emulates the ground plane of a four-layer board by providing a ground return path under each of the signal traces
- Lowers the impedance between the microcomputer and the voltage regulation

Gridding is done by expanding any ground traces and using ground-fill patterns to create a network of connections to ground across the PCB. For example, a PCB has most of the topside traces running vertically and most of the bottom traces running horizontally (see Figure 7). This already is working against having the return run directly under signal. First, every ground trace is expanded to fill up as much of the empty PCB space as possible. Then, all the remaining empty space is filled with ground. Place through holes where top-side traces cross bottom-side traces. Then do the same to the ground-fill patterns. Ground-fill patterns make a better contribution to the grid if they are tied to ground at both ends. A ground-fill-pattern geometry connected at only one point is just a ground shield, but if connected at two or more points, it becomes a conductor, and, therefore, becomes a contributor in the grid.

- Grid as much as possible on a two-layer board. Look for places where small changes in the layout would allow another connection to be made in the grid.
- Use as many through holes as can physically fit.
- Lines do not have to be orthogonal, or of the same width.

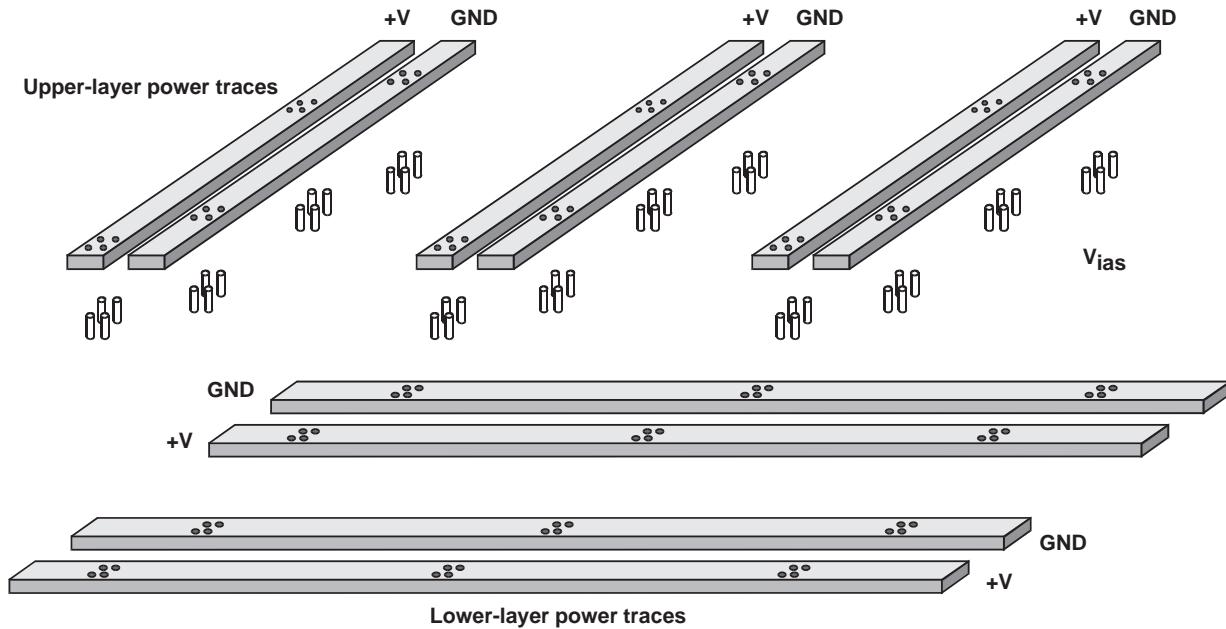


Figure 7. Gridding Power Traces on Two-Layer Boards

An example of gridding ground only to achieve the effect of a ground plane is shown in Figure 8. Note how the changes made in order to implement this were minor, indicating how a small effort can have a large payback.

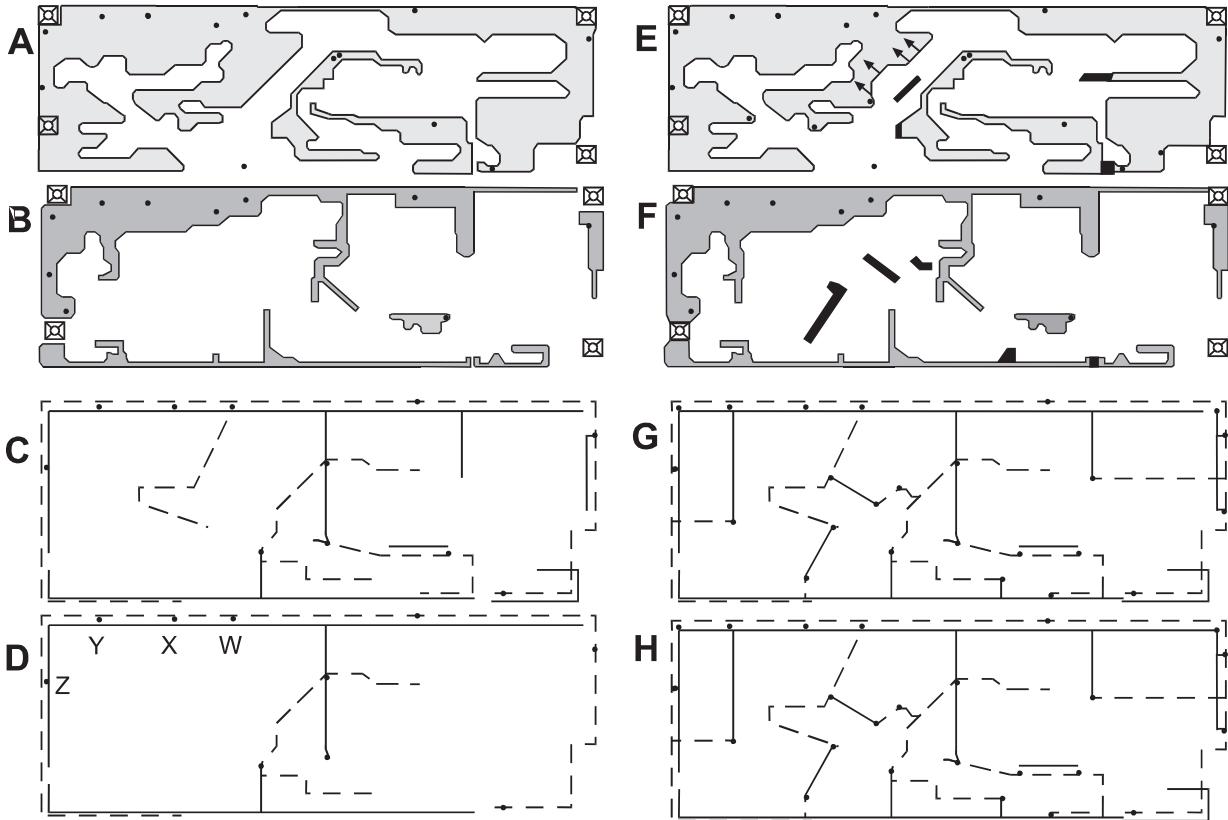


Figure 8. Gridding of Ground Fills and Traces to Form a Ground Plane

In the example in Figure 8, A and B represent the top and bottom sides, respectively, of a simple two-layer board. The +V traces and all interconnects have been deleted, leaving only the ground fill and ground traces, along with the vias between the front and back. Figure 8C is a simple stick diagram of the ground routing for the board. Each stick, or leg, represents the path of the ground conductor, as if the conductor has been shrunk down to a minimum-width trace. The top-side traces are represented by the dashed line, and the bottom-side traces by the solid line. It is easy to see in this diagram that most traces are dead ends. Most traces are connected at only one end. In Figure 8D, most of the single-ended traces have been removed. The result is a sparsely connected pattern that represents how ground is routed over the entire board. Excluding points W, X, Y, and Z in Figure 8D, there is only one path between any two points anywhere on the routing.

In Figures 8E, 8F, 8G, and 8H, the design has been modified very slightly, to achieve a gridded ground. In Figures 8E and 8F, the addition of some traces, shown in solid black, and slightly moving some geometries, as indicated by the arrows, has created an extensive network of interconnections that creates the desired grid. This is shown by the stick diagram of ground in Figure 8G. Closing the gaps around the mounting holes also contributes to the network. No longer are whole traces connected at only one end. Now, they connect at both ends, and form a more complete conductor. Figure 8H shows the density of the grid, which contrasts to the openness of Figure 8D. Also, notice how, in Figure 8H, no traces are dropped because they connect only at one end. Only one trace has this problem, and it is part of a geometry already connected in three other locations. This interconnected network is the goal of gridding ground. The result is nearly as effective as an actual ground plane.

2.2.4 Bypassing and Ferrite Beads

Bypassing between the +V and ground at the microcomputer is critical because the intent is for the capacitance to supply the current used in the device for switching. If the current is not available in the bypassing loop, because of too much inductance, the laws of physics say that the current should come to the lowest impedance, which then is from the leads connecting to the power supply. The distributed capacitance of the power routing becomes the source for the higher frequencies. Thus, the ferrite bead blocks the sourcing of RF current from the power line connection, forcing the microcomputer to live off the current available inside the ferrite bead.

It is of the utmost importance to realize, and always keep in mind, that the power-routing purpose is only to replenish the charge in the bypassing capacitor, and that the bypassing capacitor should supply all currents at or above the oscillator frequency. Keeping RF off the power distribution traces is accomplished using these measures (see Figure 9):

- Use a ferrite bead and a bypassing capacitor ($0.1 \mu\text{F}$ or $0.01 \mu\text{F}$), placing the capacitors inside the ferrite bead. Place a 1000-pF capacitor outside the ferrite bead, creating a PI filter. The ground connection for this capacitor should be the microground. However, if there is a lot of noise on this point, the capacitor could couple that noise back onto the $+V$ line.
- The ferrite bead is used only on $+V$, not on ground. If a through-hole ferrite bead is used, it is mounted with the exposed lead connected to $+V$.
- Apply the 3:1 length-to-width rule for traces in the bypassing loop, to minimize impedance in this high-frequency path.
- Make the bypassing loops as small as possible in area and length. When tying the bypass capacitors for the oscillator or $+V$ supply, try to extend the microcomputer ground rather than running a trace. Try to run any trace back over (or under) any other segment of the loop to reduce the radiating area when viewed from the top of the board.
- It is acceptable and beneficial to use ferrite beads and the same bypassing values on four-layer boards. The 1000-pF capacitor may not be needed on four-layer boards, but it should be drawn in the initial design, and deleted later if screen-room testing shows that it is not needed.

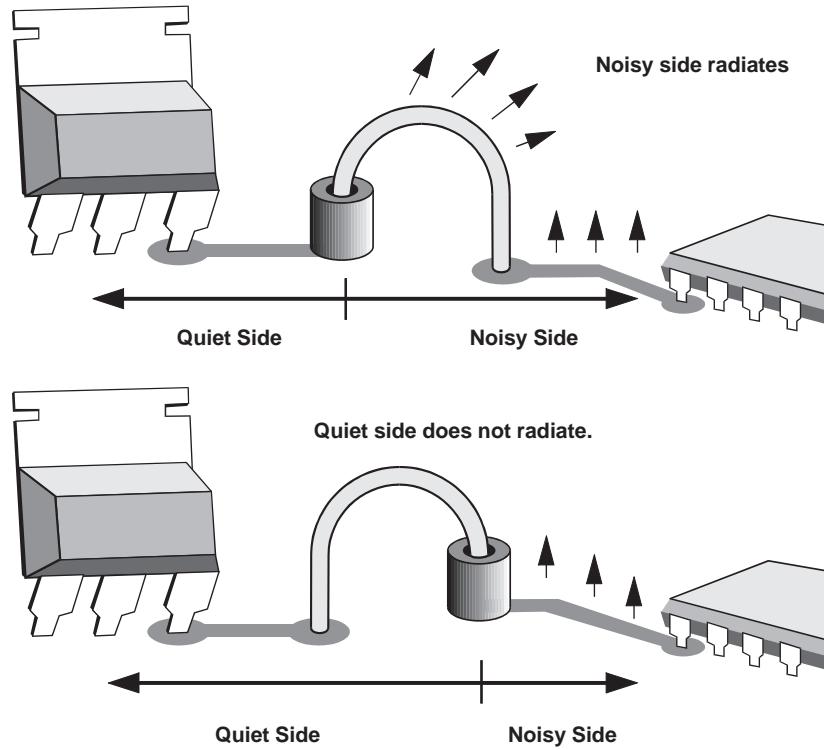


Figure 9. Ferrite-Bead Placement Closest to the Noise Source

2.2.5 Keeping Noise Close to the Chip

The following applies to pins that are used for simple digital I/O, not for pins used in the memory-expansion bus. The goal here is not so much to reduce the noise of the edge switching, but to mute the noise of the clock glitches when the pin is static.

Noise on the pins is coupled internal to the device through many paths that can change as the pin function changes. For example, the input pin in a keyboard scan has capacitively coupled noise from both the substrate and the power rails. Also, because it is high impedance, any ambient fields couple efficiently. When the key is pushed, the pin has a new set of noise sources because the signal line's impedance has changed. Thus, it is difficult to effectively develop a matrix of all possibilities; therefore, the following is recommended:

- Put a $50 - 100\text{-}\Omega$ resistor in series with every output pin, and $35 - 50\text{-}\Omega$ resistor on every input pin. If the system design calls for higher series resistance, use that value. Higher resistances are better for outputs, but usually do not improve characteristics of inputs. Place the resistor as close as possible to the microcomputer, overlapping the microcomputer ground if possible.
- Bypass any pin on the microcomputer to ground using a 1000-pF capacitor, provided the edge rate needed for the signal line is not faster than 100 ns . On outputs and pins that the system uses for both input and output, ground for the capacitor should be the microcomputer ground. The other end of the capacitor should be tied to the receiver side, not the microcomputer side, of the series resistor. Placing the capacitor inside the resistor makes the load seen by the microcomputer look like a short when it switches, which is not desirable. If adding the capacitor has to be traded off against placing the series resistor, because of space limitations, place only the resistor.
- On pins used for input only, place the capacitor inside, on the microcomputer side, of the resistor to reduce the loop area. Then, high frequencies originating in the microcomputer on the pin see less impedance to ground through the capacitor than through the resistor.
- Resets and interrupts are special functions, thus care must be taken not to reduce functionality.
- Do not apply any of the above remedies to oscillator pins. If proper spacing between the oscillator components and other unrelated components and traces is maintained, there should not be a need for oscillator signal conditioning.
- Unused pins should be configured as inputs and tied directly to the microcomputer ground. It is recommended that the watchdog be enabled to correct the unlikely event in which a device is disturbed, loses its program counter, and executes code to make the input become an output with a high level.

These rules take up space and add components, and so are not well accepted in production. The goal is to implement all rules on all I/O pins, but if that is not possible, then rank order the candidates least likely to cause noise and remove the application of these rules one pin at a time.

Filtering priorities from most needed to least needed are:

- Signals leaving the enclosure (see section 3.3, *Cables and Bypassing to the Shield*)
- Signals leaving the PCB to other boards inside the enclosure
- Signals staying on the PCB with high-impedance loads (i.e., driving another MOS input or open circuit)
- Pins of parallel I/O port designed to support high-speed data transfer, e.g., between the microcomputer and an external memory, need filtering over the remaining I/O pins, because of their faster rise and fall times.

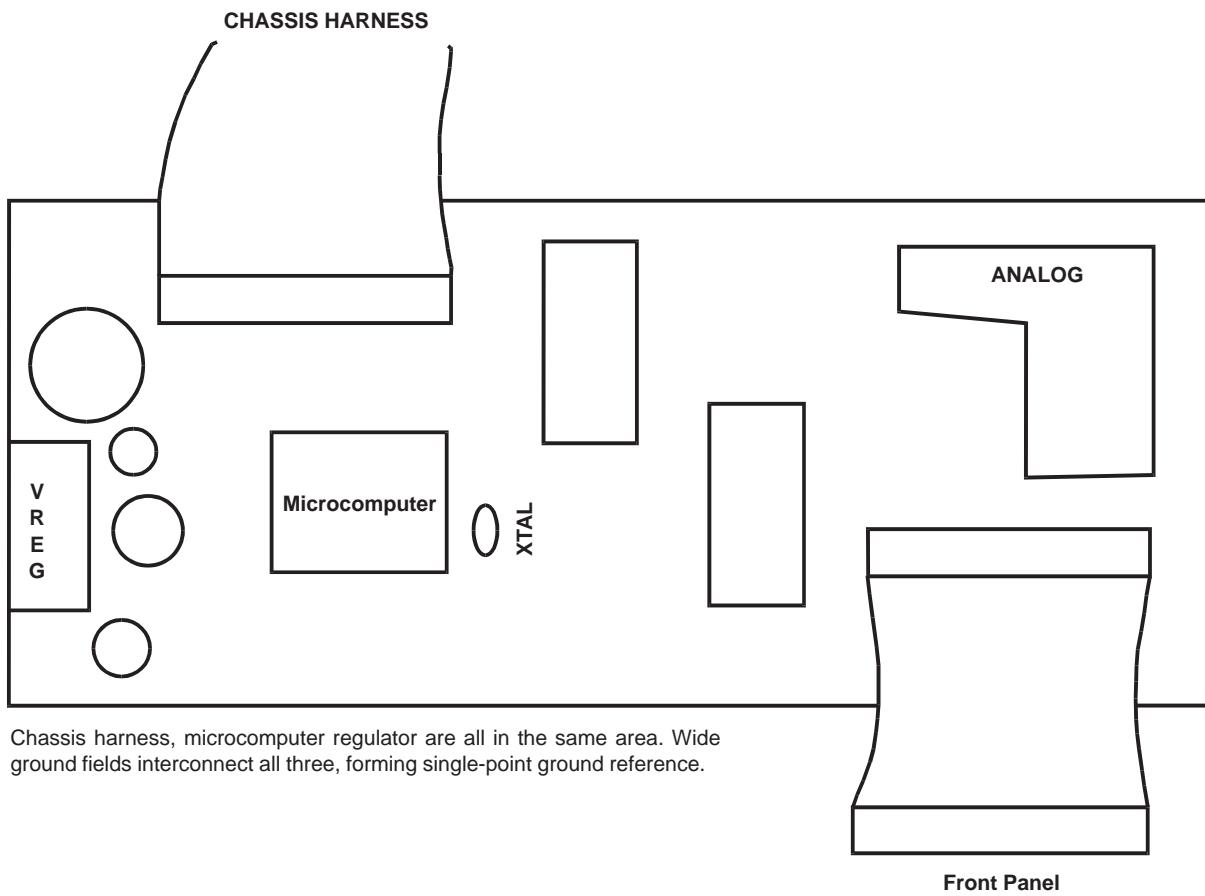
When the design is complete and first prototypes are built, an hour or two in the screen room removing each of the filtering components one at a time, identifies which are or are not needed to get the desired EMI level.

2.3 Board Zoning

Board zoning has the same basic meaning as board floor planning, which is the process of defining the general location of components on the blank PCB before drawing in any traces. Board zoning goes a little bit further in that it includes the process of placing like functions on a board in the same general area, as opposed to mixing them together (see Figure 10). High-speed logic, including micros, are placed close to the power supply, with slower components located farther away, and analog components even farther still. With this arrangement, the high-speed logic has less chance to pollute other signal traces. It is especially important that oscillator tank loops be located away from analog circuits, low-speed signals, and connectors. This applies both to the board, and the space inside the box containing the board. Do not design in cable assemblies that fold over the oscillator or the microcomputer after final assembly, because they can pick up noise and carry it elsewhere.

In prioritizing component placement, the most important things to do in PCB design are:

- Locate the **microcomputer next to the voltage regulator**, and the **voltage regulator next to where V_{Batt} enters** the board.
- Built a gridded or solid ground between the three (forming a single-point ground), and tie the shield at that point.



Chassis harness, microcomputer regulator are all in the same area. Wide ground fields interconnect all three, forming single-point ground reference.

Figure 10. Board Zoning

2.4 Signal Traces

2.4.1 Capacitive and Inductive Crosstalk

Capacitive and inductive crosstalk occur between traces that run parallel for even a short distance. In capacitive coupling, a rising edge on the source causes a rising edge on the victim. In inductive coupling, the voltage change on the victim is in the opposite direction as the changing edge on the source. **Most instances of crosstalk are capacitive.** The amount of noise on the victim is proportional to the parallel distance, the frequency, the amplitude of the voltage swing on the source, and the impedance of the victim, and inversely proportional to the separation distance. Measures that reduce crosstalk are:

- Keeping RF-noise-carrying traces that are connected to the microcomputer away from other signals so they do not pick up noise.
- Signals that may become victims of noise should have their return ground run underneath them, which serves to reduce their impedance, thus reducing the noise voltage and any radiating area.
- Never run noisy traces on the outside edge of the board.
- If possible, group a number of noisy traces together surrounded by ground traces.
- Keep non-noisy traces away from areas on the board where they could pick up noise, such as connectors, oscillator circuits, relays, and relay drivers.

Most EMI-related crosstalk problems center around the crystal, when the victim is located too close. No unrelated components should be closer than 1 inch to the crystal.

2.4.2 Antenna Factor Length Rules

Normally, for Federal Communication Commission (FCC) limits, trace length becomes important when it is greater than 1/10 of the wavelength. For military standard limits, that number becomes 1/20 to 1/30 of the wavelength. For automotive and consumer two-layer boards, 1/50 of the wavelength begins to be critical, particularly in unshielded applications. That says traces longer than 4 inches can be a problem for FM-band noise. In these cases, some form of termination is recommended to prevent ringing.

2.4.3 Series Termination, Transmission Lines

The main purpose of termination is to provide critical damping to achieve the highest possible data transmission rates with the least-possible overshoot. When applied to most microcontroller systems, however, the focus changes to taking out as much total differential-mode noise as possible while allowing system functionality. Below is a table of different methods of termination and the main characteristics of each method.

Table 1. Termination Characteristics

	Parallel	Thevenin	Series	AC	Active
Power dissipation	high	high	low	medium	medium
Number of components	1	2	1	2	1
Adds delay	no	no	yes	no	no

Note that CMOS is an under-damped technology, which means that you always have to be on guard for ringing and overshoot. Use some form of termination if any of the following conditions are present:

- A signal trace is more than 1 foot long.
- A signal goes to a cable that leaves the shielded enclosure.
- Any ringing is present.

Series resistance is an inexpensive solution to termination and ringing problems, and is the preferred method for microcomputer-based systems where minimizing the differential-mode noise is also a concern.

2.4.4 Impedance Matching at Inputs

The input to a CMOS device looks like a series inductance of about $5 - 40\text{ nH}$, which leads to about 5 pF in parallel, with about $5\text{ M}\Omega$ to the grounded substrate (see Figure 11). This is a very high impedance, and can lead to lots of ringing and other noise if the device driving the input is not matched in some fashion to the higher impedance. This is the complement of the situation of section 2.4.3, where attention is paid to the microcomputer's output because of the under-damped nature of the load it drives. Here, the microcomputer is the under-damped load, and ringing and overshoot are real possibilities. More than likely, some form of termination will be required, and again, the series resistance is the most likely solution. Resistance placed at the driver increases the output impedance, as seen by the trace and the input pin, thus matching the high impedance of the input.

If the input is connected to an open trace, such as the open line to a switch, a pullup or pulldown resistor is recommended. While this increases the amount of current switched when the input is activated, it reduces the impedance at all other times. This reduces the trace's chances of being a victim of coupled noise.

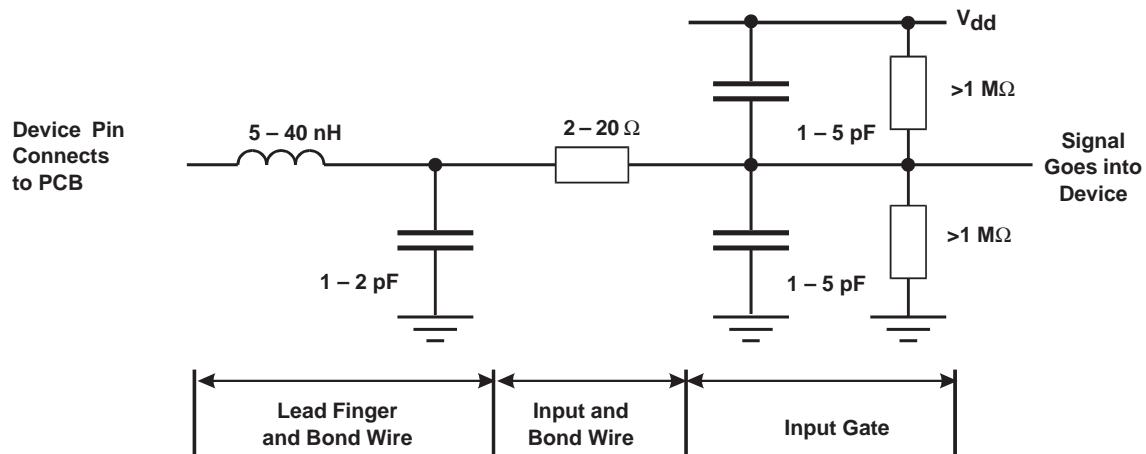


Figure 11. MOS Buffer Simplified Schematic

2.5 Cables and Connectors

A well-designed two-layer board, and most four-layer boards, have minimal radiation. The problem at the system level is the radiation due to cables interconnecting the PCB with any off-board support function, other processor, or display and keypad PCBs. Because usually there is only one ground wire between boards, that one inductive wire has to return all of the RF energy carried onto the second PCB by the other wires. If there is any impedance in the single ground wire, a portion of the RF energy does not return to the microcomputer's PCB via the ground wire, but rather through a radiated path. The energy radiates off the second board and couple back to the first, but, during the process, that radiation also can add noise in other locations in the system, as well as become direct radiation measured in the screen room. The key corrective action is to ensure the conducted path for the return has a very low RF impedance.

2.5.1 Differential-Mode and Common-Mode Noise

Common-mode noise is a big problem in cables, but the fault does not lie in the cable, it lies in the connections on the board that the signals and returns tie to that form the common impedance. Common-mode noise is corrected either at the source, by reducing the impedance of the common node, or reduced by placing a ferrite bead around the entire cable.

Differential-mode noise (the useful noise of an edge transition) should first be reduced to the maximum (slowest) allowable rise and fall times and should occur at only the minimum needed frequency. The noise radiated is due to the loop of the signal and its return. This loop is minimized by having as many returns as possible and by twisting each signal and return pair. The latter causes field cancellation at some distance away, in the same manner as routing power over ground.

2.5.2 Crosstalk Model

Crosstalk in a cable is the same as in the PCB. Noise is coupled from the source onto quiet victim signals. Therefore, run clocking or other high-speed wires twisted with their own separate return. Crosstalk is a problem in cables over 2-meters long, and can be a problem in cables as short as 6 inches.

2.5.3 Number of Returns

It is common practice in the computer industry to have at least one ground for every nine other signal lines in a cable or harness. With higher speeds, this ratio is moving toward 1:5. These higher speeds are not limited to data rates, but also to harmonic content. Use these guidelines in designing signal and return lines:

- The best practice is to have one ground return for each signal in the cable, as a twisted pair.
- Never run less than one ground return for every nine signal lines, even if it is just the jumper cable to the front-panel display in a completely sealed metal box.
- If the cable is over one foot long, it should have one ground return line for four signal lines.
- When possible, there should be a solid metal bracket, used as a mechanical brace, soldered between the two boards, to serve as a mounting bracket and as a robust RF ground return.

2.5.4 I/O Recommendations for Off-PCB Signals

The PCB should have a large ground area tied to the enclosure shield that serves as the ground for the bypassing capacitors on each of the wires entering or leaving the enclosure. These capacitors provide final filtering of microcomputer noise, but also are intended to filter to the shield any noise picked up on the cable outside the box. See section 3.3, *Cables and Bypassing to the Shield*.

2.5.5 Keeping Noise and Electrostatic Discharge (ESD) Out

Noise and ESD incident on the cable are intended to pass through the bypass capacitor at the cable on the PCB and out to the shield (chassis). Therefore, the ground from the capacitor to shield should be wide (3:1) and bonded securely to the shield, preferably by two or more screws. The bypass capacitor value should be less than 1000 pF, so the effective series resistance (ESR) is minimum in the 50 – 500-MHz range. Lead length of axial devices would be a factor in the ESR, so surface-mount components are preferred.

2.6 Other Layout Issues

2.6.1 Front-Panel PCB with Keypad and Display in Automotive and Consumer Applications

In multi-PCB applications, a front-panel PCB, which carries the display and the keyboard, is part of the shielding enclosure. It also can be a source of emissions. The goal here is to make the ground return for the noise of the microcomputer, and to create an effective extension of the shield over the front of the box. This can be done by defining all lines on one side of the board to run one way, and the other side to run at 90 degrees (see Figure 12). Then, interspersed among the signals for keyboard and matrix, are the lines that make up two separate gridded ground planes: one that serves as the ground return and one that serves as part of the shield. The latter should securely contact the chassis at each corner and every two inches along the edge. The return ground plane should tie to the microcomputer's ground plane, preferably using something heavy, like soldered-in metal braces. This prevents the impedance between the controller and the front-panel PCB from being a major problem.

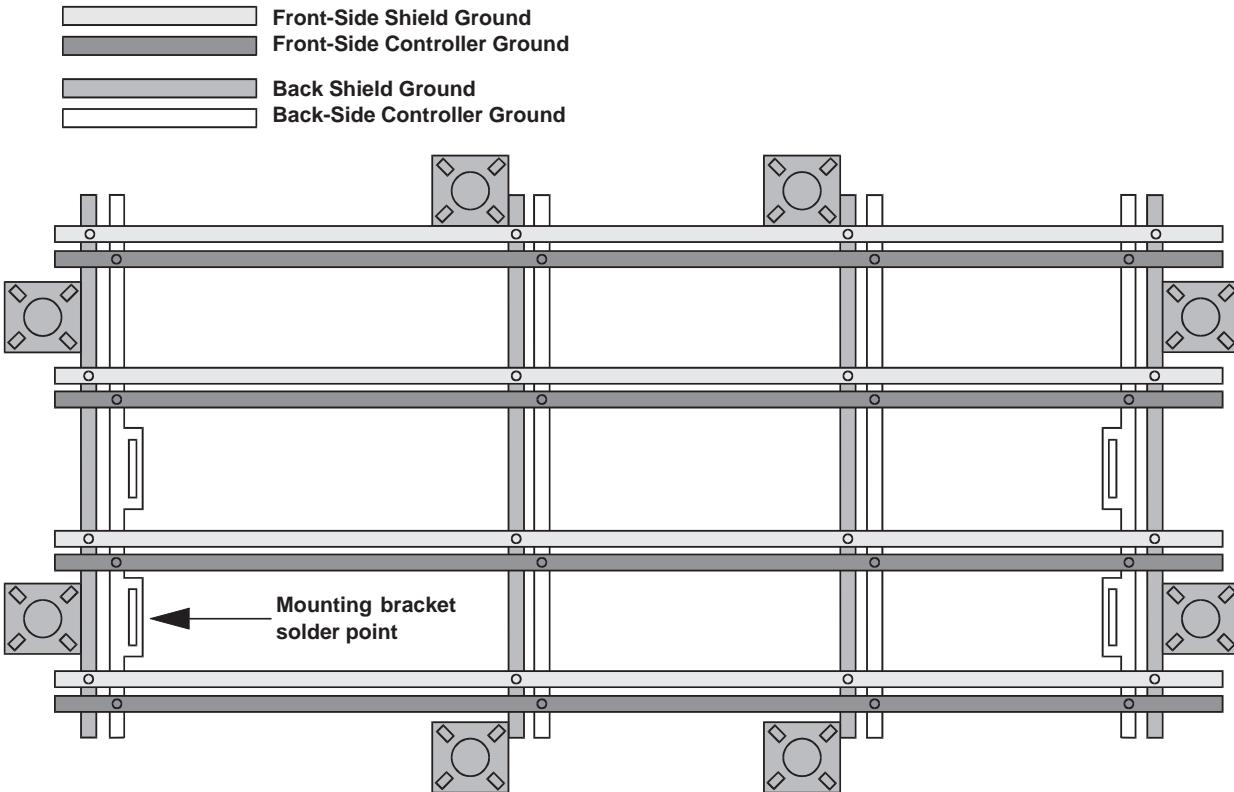


Figure 12. Front-Panel Gridding to Form Two Ground Planes

2.6.2 Layout for Susceptibility

Susceptibility (called immunity in Europe) occurs as incident electric or magnetic fields couple onto signal traces. Because the coupled signal is alternating current, the sine wave is superimposed on voltage already present on the trace. At the input to a microcomputer, that voltage is rectified, and causes a dc offset voltage on the pin. When this dc voltage gets large enough to shift the input away from the switching-point voltage, the intended switching function is no longer seen by the microcomputer. If the input is the oscillator, the device suddenly has no clocks. If the input is reset, the device may go into reset and stay there until the disturbing field is removed.

The physics of susceptibility are the same as for emissions, only applied in reverse. Large loop areas pick up more signal, just as they would radiate more signal. Therefore, the things you do to keep a signal immune from radiation are the same as those to keep it from radiating.

The most important pins for immunity are those that affect program control: the oscillator, reset, interrupts, and any input pins used for program branching. Apply the same rules as for reducing radiation from these pins. *By far, most susceptibility problems are associated with the loop defined by the oscillator pins, the crystal, the crystal bypass capacitors, and the path between the bypass-capacitor ground connection and the ground of the microcomputer.*

Also, be careful of ground bounce (common impedance coupling) of circuits that may generate these more critical signals. If the ground path has high impedance, it may cause the driving circuit's reference voltage to shift, causing that input to the microcomputer (RESET, OSC) to be outside the switching range of the microcomputer.

2.6.3 Autorouters

Autorouters for PCBs do not take any noise reduction actions; therefore, care should be taken in their use. Power and ground routing, as well as signals that impact susceptibility, should be laid out by hand. Any signal with clocked data, such as low address bits in a memory expansion bus, should be next. Only signals with switching rates below 50 kHz can be left safely to the autorouter. Even then, every signal should be checked for EMI issues. Routing near the crystal, and the crystal and tank circuit itself, should be checked. Finally, the ground traces should be gridded.

3 Shielding

3.1 How It Works

When an incident electric field traveling in the air hits a metal surface, the metal causes the penetrating field strength to decrease. The metal causes the field to be replaced with conduction currents that flow in the metal close to the surface. A very small (exponential decay) amount of the field does pass through, but for emissions, this is never a problem. The metal chassis serves as a shield. The fields from all the radiating surfaces inside it are blocked and kept inside the box, with the only noise coming from the cables or wires that enter or exit the box and from holes or slots made in the box.

If a metal enclosure is to be used, its shielding effect should be utilized. However, it is always better to reduce the noise inside the box than to rely on the shielding effectiveness.

3.2 Grounding the Shield

The shield has the difficult job of providing a terminating or conducting surface for direct ESD hits, ambient fields, and internal fields, as well as noise carried on the cables entering and exiting the chassis. To do this well, the shield should be thought of as an RF conducting plane, with the least number of breaks and impedance's between the source of the RF currents and the ground reference point. The ground reference point should be the single point, as mentioned earlier, that ties together the regulator ground, the microcomputer ground, and battery negative.

3.3 Cables and Bypassing to the Shield

The PCB should have a large separate ground area tied to the enclosure shield which serves as the ground for the I/O bypassing capacitors (see Figure 13). These capacitors provide final filtering of system noise, but also are intended to filter noise picked up on the cable outside the box. The value of the capacitor should be below 1000 pF, more likely about 470 pF. The connection to the chassis is an RF path requiring 3:1 length to width.

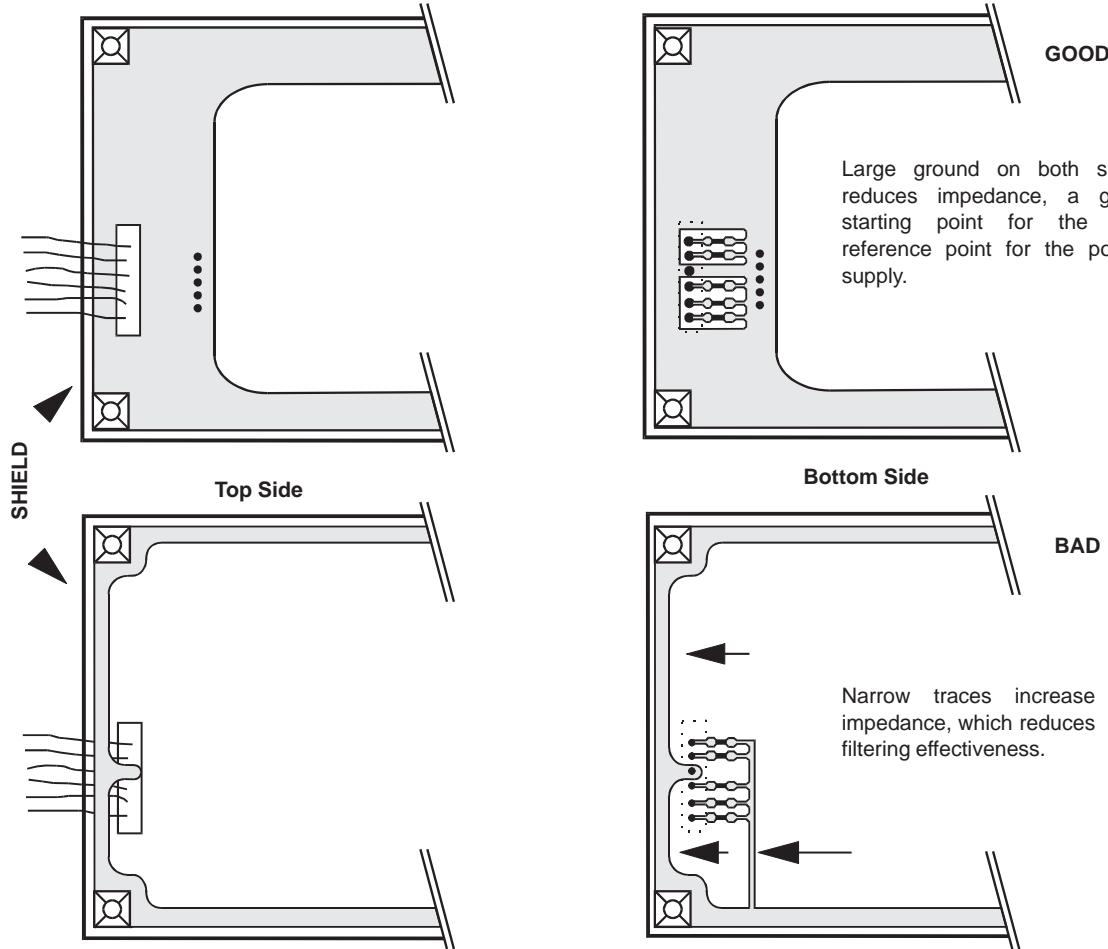


Figure 13. Mounting Filter Capacitors for External I/Os

3.4 Slot Antennas: Cooling Slots and Seams

Slots antennas are formed by long thin gaps in the shielding material, such as at the seams between the two pieces of the box, and at the front-panel interface. These slots are very effective radiators. It is important that some form of contact assurance be used, such as dimpling, or using alternating fingers, to insure contact between the two surfaces. No slot should be more than 4 inches long. Cooling slots should not be used. For emissions reasons, only small round holes should be used if ventilation is needed.

4 Summary

The design of systems that generate low electromagnetic interference is not a mystery, but requires application of well-known engineering techniques. The design begins with the selection of semiconductor components that produce low electromagnetic radiation. However, in many cases, other criteria, such as the required performance of the semiconductor component, may be in contradiction with low interference. The main task is the design of a PCB that eliminates antennas that can radiate electromagnetic energy. Even if this can be achieved sometimes, large loops of signal and corresponding ground-return lines that carry high frequencies must be avoided. Therefore, a careful positioning of the integrated circuits is essential to achieve short interconnect lines.

In the next step, a close ground grid is placed over the printed circuit board. This grid ensures that return lines are close to the signal lines, thus keeping the effective antenna area small. A ground plane in a multilayer board provides this feature. By using this technique, low electromagnetic emission can be achieved with low design effort. However, some cost-sensitive applications allow two-layer PCBs only. Nevertheless, in this case, careful layout provides nearly the same performance as a multilayer board.

Finally, filtering of critical lines, such as the supply line, ensures that high-frequency currents do not leave the PCB.

By applying the rules presented in this report, shielding of the total system is not required. Experience and careful work by the design engineer are much more effective than sophisticated computer-aided design tools.

5 References

1. *Printed-Circuit-Board Layout for Improved Electromagnetic Compatibility*, October 1996, Application Report, literature number SDYA011.
2. *Electromagnetic Emission from Logic Circuits*, November 1998, Application Report, literature number SZZA007.
3. Texas Instruments Internet Web Page at <http://www.ti.com>