AP-711

APPLICATION NOTE

EMI Design Techniques for Microcontrollers in Automotive Applications

CHRIS BANYAI
Technical Marketing Engineer
Intel Automotive Operations

DARYL GERKE
EMC Consulting Engineer
Kimmel Gerke Associates, Ltd.

February 1996

Order Number: 272673-001

Contact your local Intel sales office or your distributor to obtain the latest specifications before placing your product order.

Copies of documents which have an ordering number and are referenced in this document, or other Intel literature, may be obtained from:

Intel Corporation
P.O. Box 7641
Mt. Prospect, IL 60056-7641

or call 1-800-879-4683

CONTENTS PAGE

CONTENTS PAGE

Electronics content of automobiles and other vehicles has grown rapidly in recent years. Embedded microcontrollers are used in a wide range of vehicle applications for control, convenience, and comfort. Examples range from sophisticated engine and braking controls to automated radios and individual passenger temperature controls.

As the electronics content of vehicles have increased, so have the *electromagnetic interference* (EMI) problems. These range from annoyances (jamming an on-board AM of FM radio) to upset or damage (blowing out an engine control module due to power transients.) The problems are expected to get worse as system clock speeds and logic edge rates increase, due to increased EMI emissions and decreased EMI immunity.

This application note describes the automotive EMI environments, and then discusses how to identify and prevent many common EMI problems at the design stage. Although a range of solutions will be addressed, emphasis is on printed circuit board design methods.

This application note also describes some recent Intel sponsored research efforts that investigate EMI to on-board FM radio receivers. Several different design approaches were tested, using both two layer and multilayer circuit boards. The test program was based on an ABS (anti-lock braking system) control module that uses the Intel 80C196KR microcontroller. The results and recommended "low noise" design concepts, however, apply to any microcontroller design used in vehicular applications.

AUTOMOTIVE EMI PROBLEMS

Vehicle electronics are affected by several factors, including *harsh environments, high reliability,* and *extreme cost sensitivity.* Fortunately, these problems can be overcome through good EMI design techniques.

The automotive environment contains several threats, including power transients, radio frequency interference (both to and from nearby or onboard radio transmitters and receivers), electrostatic discharge, and power line electric and magnetic fields. Since vehicles can go almost anywhere, the worst case situations must be assumed.

Vehicular electronics must be designed for extremely high reliability. Even a single failure over millions of vehicles may not be tolerated. Furthermore, any system that affects vehicle safety must be "fail-safe." Systems must also be easy to install, test, and repair. And of course, all of this must be done at the lowest possible cost.

Here some comments on common EMI threats that are faced by the designers of vehicle electronics. They are divided into two broad classes—susceptibility (also referred to as immunity) and emissions. In the first case, the automotive electronics are the *victim* of EMI, and in the second case, the automotive electronics are the *source* of EMI.

Automotive EMI Susceptibility

Since the automotive environment is so severe, many automotive electronics designers are already well versed in dealing with the following EMI problems. Nevertheless, we'll review them here, since understanding the problems is the first step toward solving them.

Power Transients—Vehicle electrical systems are a rich source of power transients. Seven of the most severe have been characterized and have become a suite of standard EMI test pulses, as described in SAE J1113, "*Electromagnetic Susceptibility Procedures for Vehicle Components*". These transients include pulses that simulate both normal and abnormal conditions, including inductive load switching, ignition interruption or turn-off, voltage sag during engine starting, and the alternator "load dump" transient. The last is particularly harsh, and can destroy unprotected electronic devices. For more information on these transients, see SAE J1113.

All of these transients can damage or upset electronic systems. Digital circuits are particularly susceptible to transients, which can result in false triggering or "flipped bits" in memory. As electronic devices become faster and smaller, they become even more vulnerable to these transient spikes.

Radio Frequency Immunity—Since vehicles are often a platform for land mobile radio transmitters, the on-board electronics systems may be exposed to very high radio frequency (RF) electromagnetic field levels. The vehicles can also be exposed to high levels from external threats, like high powered radio stations or airport radar systems.

The "electric field" levels from these threats can easily reach 50–100 volts/meter. In order to protect against these threats, test levels of up to 200 volts/meter are specified for automotive applications. Since typical failure levels for unprotected electronic systems are in the 1–10 volt/meter range, substantial RF protection must be provided for electronic systems operating in the automotive environment.

Although both digital and analog circuits are vulnerable to the RF threat, low level analog circuits, such as sensors, are the most vulnerable. A common failure mode is "RF rectification", which occurs when an analog circuit is driven non-linear by a large signal induced by large RF fields. Voltage regulators can also be affected if the RF energy gets into the feedback loop of the regulator. Due to their higher operating margins, digital circuits are not as vulnerable to this threat, but even they can be affected at high RF levels when using two layer boards.

It's easy to predict electric field levels if you know the transmitter power and distance, using the following formula:

$$E = 5.5\sqrt{AP}/d$$

where E is the electric field level in Volts/meter
P is the transmitter output power in watts
A is the gain of the antenna (the product of AP is effective radiated power)
d is the distance from the antenna in meters

This formula assumes an isotropic or "uniform point" source, and assumes no intervening shielding between the transmitter and the vulnerable electronics. Nevertheless, it's quite accurate, particularly at the citizens band, land mobile, and television frequencies of 25 MHz and higher. Most radio frequency interference susceptibility problems occur at these higher frequencies, where the cables and even circuit traces can act as efficient antennas.

Table 1. Electric Field Levels vs Distance and Power (Free Space—Isotropic Source)

	1 Watt	10 Watts	100 Watts	100,000 Watts
1 meter	5.5 V/m	17.3 V/m	55 V/m	1730 V/m
10 meters	0.55 V/m	1.73 V/m	5.5 V/m	173 V/m
100 meters	55 mV/m	0.17 V/m	0.55 V/m	17.3 V/m
1 kilometer	5.5 mV/m	17 mV/m	55 mV/m	1.73 V/m

Table 1 gives some examples. Note distance is critical, and even a low power hand held (walkie-talkie) radio is capable of high field levels when it is close to the victim electronics. A 1 watt hand held radio at 1 meter results in about 5 volts/meter, while a 100 watt transmitter (typical of many fixed mobile transmitters) at 1 meter results in over 50 volts/meter. Both levels are much higher than from a 100,000 watt radio broadcast station located 1000 meters away. In vehicles, the local on-board transmitter is usually the biggest RF threat. Even low powered cellular phones can cause interference problems, if their antenna is close to victim electronics.

Electrostatic Discharge (ESD)—Since almost every vehicle has humans on board, and since humans are a ready source of ESD, this is another major EMI threat to vehicular electronics.

Most ESD requirements are based on the "human body model", which characterizes typical voltages, currents, and risetimes associated with a human ESD event. Although ESD discharges are usually specified in "pre-discharge" voltage levels, it's actually the current pulse that causes most of the problems. Like water running down a river bed after the dam breaks, the ESD current can upset or destroy any vulnerable electronics in its path. Furthermore, the electromagnetic field associated with the ESD event can also radiate into nearby electronics systems, causing even more upsets. This is known as the "indirect effect" of ESD.

Figure 1 shows the ESD current pulse resulting from a human ESD event. Note that this ESD current has a very rapid risetime in the 1–2 nanosecond range, with peak currents of 10 amps or more. A 1 nanosecond edge rate has an equivalent "bandwidth" of over 300 MHz, so ESD is very much a high frequency issue and requires high frequency design solutions.

Because of this high frequency content, a "direct hit" is not necessary. The intense electromagnetic fields can easily upset a nearby system from an "indirect hit" of ESD. This effect has been observed up to 20 feet (6 meters) away, and is a reason that "indirect" testing is now included in recent international ESD test specifications.

As mentioned above, the test specifications are usually given in the "pre-discharge" voltage levels. Most automotive electronics are designed to withstand at least 15 KV, a severe level that actually exceeds most human ESD levels.

Power Line Fields—Since vehicles can go almost anywhere, and since power lines (and transformers) can be almost anywhere, this threat must also be addressed in vehicular designs.

Normally, this is not a problem for microcontroller based systems, since at 50 or 60 Hz, electromagnetic field coupling is not very efficient. Nevertheless, very low analog level circuits can be affected by "stray" magnetic and electric fields. Thus, power line field requirements are usually imposed on vehicular systems to make sure no upsets occur due to this threat.

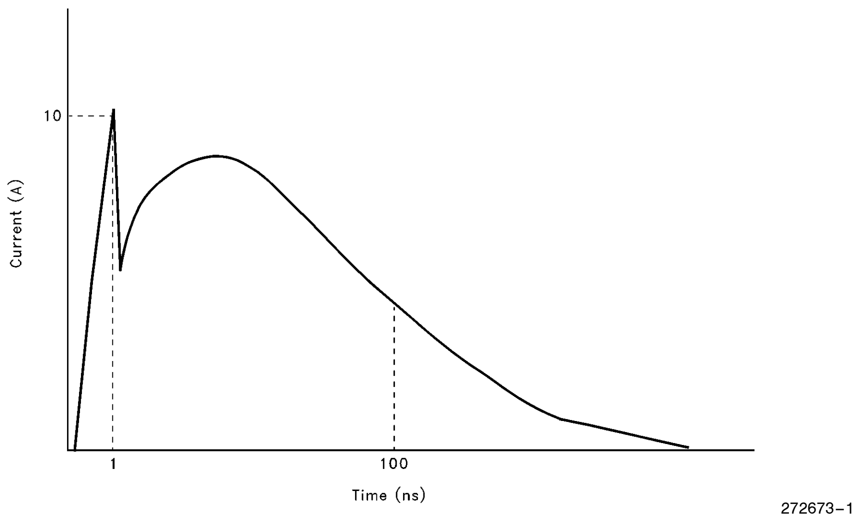


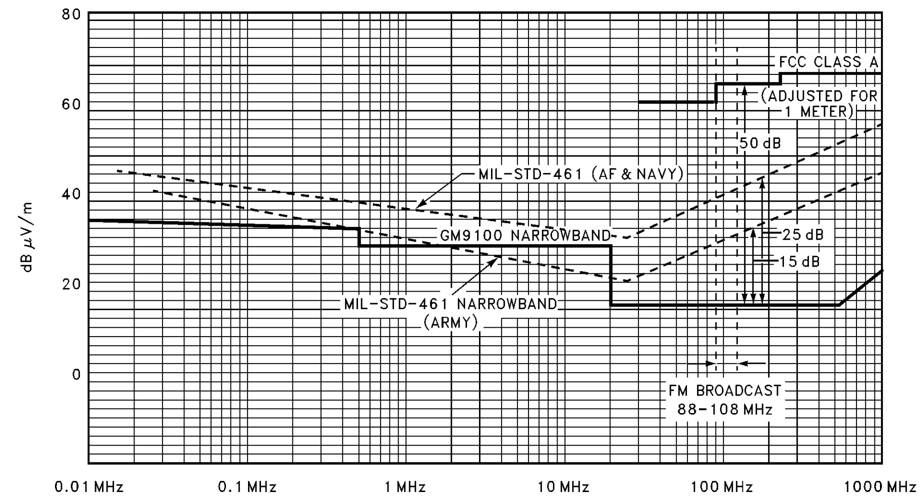
Figure 1. Typical Waveshape of ESD from Human Body

Automotive EMI Emissions

Almost all automobiles today have sensitive AM/FM radio receivers (or perhaps even a land mobile VHF radio), *so emissions from digital circuits are one of the biggest EMI problems facing today's designer of vehicular electronics.* Most of the time the problem is annoying, but in the case of emergency vehicles (police, fire, ambulance), jamming a radio receiver could be life threatening. As a result, most vehicle manufacturers now require suppressing the offending emissions to extremely low levels.

Why Commercial EMI Limits Don't Work—This problem is similar to the radiated emissions from personal computers, which results in the well known FCC (United States) or CISPR (European) EMI limits for computers. These commercial limits are aimed at protecting nearby television receivers (3–10 meters away) from interference. The military has similar limits with lower levels, designed to protect their radio communications and navigation systems from EMI.

Radiated emissions in the vehicular environment is much more severe than in the commercial or military environments. First, automotive radio receivers are much more sensitive than television receivers, and second, the offending circuits are much closer to the radio receivers (commercial or military), typically within 1 meter of the antenna.



272673-2

Comparison of Radiated Emission Specifications Commercial (FCC), Military and Automotive.

Figure 2

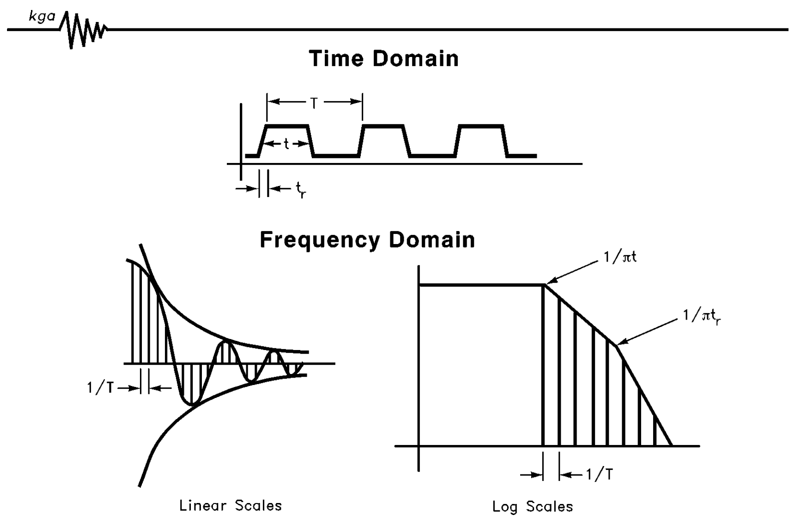
Several vehicle manufacturers have responded with their own radiated emission limits that are well below the commercial or military limits. Figure 2 compares the commercial FCC limits with both military (MIL-STD-461) with the General Motors vehicle "module" limits (GM9100) for radiated emissions. (All three sets of limits have been normalized to a 1 meter measurement distance.) In the FM broadcast range (88–108 MHz), the vehicle limits are about 6 μV/m (15 dB μV/m), which is about 300 times (50 dB) more stringent than corresponding commercial emission limits, and about 6 to 20 times (15–25 dB) more stringent than corresponding military limits.

Most vehicle problems occur in the FM broadcast band (88-108 MHz) rather than the AM broadcast band (550 kHz–1600 kHz). At FM frequencies, the cables act as efficient antennas due to the shorter wavelengths (3 meters for 100 MHz vs 300 meters for 1 MHz), so radiated emissions are much more efficient at these higher frequencies. On board VHF land mobile receivers operating in the 140–170 MHz (amateur, police, fire, ambulances) can also be similarly affected. On board UHF land mobile receivers in the 450 MHz range are not usually affected by today's microcontrollers, but they will be as the clock speeds and edge rates continue to increase.

Microcontrollers as Unintended Transmitters—The primary sources of emissions from microcontroller based systems are the clocks and other highly repetitive signals. The harmonics of these signals result in discrete narrowband signals that can often be received well into the VHF and UHF radio ranges. Although digital systems are often classified as "unintentional radiators", these harmonics are easily radiated by cables, wiring, and printed circuit board traces.

Fourier analysis is a useful tool to understand these harmonic emissions. The Fourier series shows that any non-sinusoidal periodic waveform is composed of a fundamental frequency plus harmonic frequencies. Fortunately, as the frequency increases, the amplitudes of the harmonics decrease. Unfortunately, square waves used in digital systems decrease at the slowest rate (20 dB/decade), and therefore are rich sources of high frequency harmonics. In a sense, a digital system is like a series of small radio transmitters that broadcast simultaneously on a wide range of frequencies.

Fourier Predictions



272673–3

Digital circuits are rich in harmonics, and act like multiple transmitters.

Figure 3

Figure 3 shows the relationship between the time and frequency domains for a repetitive digital signal such as a clock. Although in the time domain it is customary to express data in a linear fashion, in the frequency domain it is customary to express data in a logarithmic fashion. This "Bode-plot" approach provides additional insights, as slopes and break points become readily apparent.

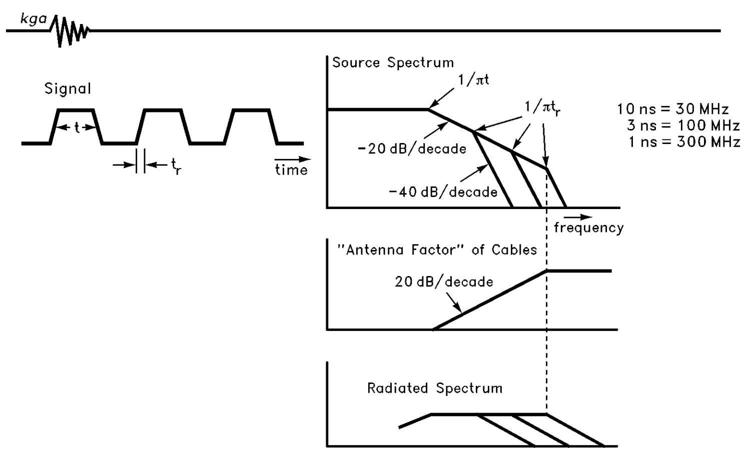
A widely used EMI frequency is the transition from −20 dB/decade to −40 dB/decade, which occurs at $1/(\pi t_r)$. This is often referred to as the "logic bandwidth", because this is the bandwidth necessary to pass the signal without degrading the edge rate. For example, the "logic bandwidth" of a 10 nanosecond edge rate is 32 MHz, while the "logic bandwidth" of a 1 nsec edge rate is 320 MHz. Since many systems today have edge rates in the 1–3 nsec range, these systems are "hot" with VHF and UHF energy. Note that this "bandwidth" is only effected by the "edge rate" and not the clock rate. Decreasing the edge rates will decrease this bandwidth, and thus decrease the high frequency content of the digital signal.

Figure 4 shows the effect of increasing a clock rate, but keeping the edge rate constant. In this case, the specific amplitude at a given frequency increases with the clock rate. *Note that BOTH edge rates and clock rates contribute to the radiated emissions that interfere with radio receivers*. Furthermore, the harmonics—not the fundamental clock frequency—cause the problems.

Cables and PCB Traces as Unintended Antennas—For emissions to be a problem, we need both a "transmitter" and an "antenna." We've already seen that repetitive digital signals act like multiple hidden transmitters. Now we'll look at how cables and board traces act as multiple "hidden antennas."

Figure 5 shows the effect of coupling the harmonics described above to a hidden antenna such as a cable. Although the harmonic frequency content decreases with frequency, the antenna efficiency increases with frequency. The net result is a pretty good system for transmitting VHF and UHF energy, even if it is unintended.

How Fast Risetimes Cause Radiated Interference

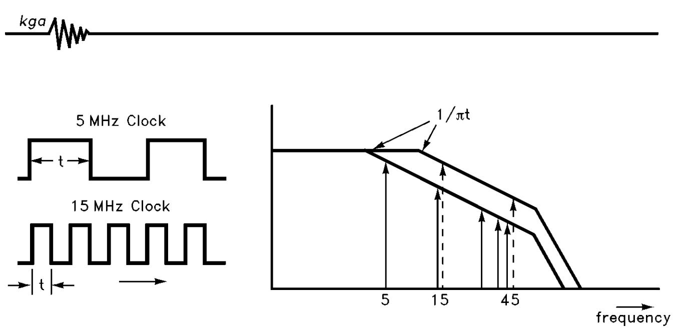


272673–4

High speed sources plus long cables equals emissions.

Figure 4

Clock Rate Affects Emissions, Too



272673–5

3x clock rate = 10 dB higher emissions.
The faster the clock frequency . . . the higher the emissions.

Figure 5

Any conductor will act as an efficient antenna when it's physical dimensions exceed a fraction of a wavelength. Since cables, by their very nature, are among the longest conductors in a system, they are usually the most significant "antennas." At higher frequencies, however, even the traces on the circuit boards become long enough to radiate. The secret to success for radiated emissions is to prevent high frequency energy from ever reaching the hidden antennas.

As a rule of thumb, any wire over 1/20 wavelength is considered an antenna for EMI purposes. Most communications antennas are 1/4 wavelength or longer, but even "short" antennas are significant radiators. Table 2 shows some typical frequencies and lengths, which are related by the formula $\lambda = 300/f$, where λ is the wavelength in meters, and f is the frequency in Megahertz. Also shows are "equivalent risetimes" based on the "logic bandwidth" frequencies of digital signals, using the formula $t_r = 1000^*f/\pi$, where t_r is the risetime in nanoseconds and f is the frequency in MHz.

Table 2. Frequency, Wavelength, Rise Time

Frequency	Approx. t_r	Wavelength	1/20 Wavelength
300 kHz	1 μsec	1000 meters	150 meters
1 MHz	300 nsec	300 meters	15 meters
3 MHz	100 nsec	100 meters	5 meters
10 MHz	30 nsec	30 meters	1.5 meters
30 MHz	10 nsec	10 meters	50 cm
100 MHz	3 nsec	3 meters	15 cm
300 MHz	1 nsec	1 meter	5 cm

Based on the 1/20 wavelength criteria, a cable that is 1.5 meters long is a good antenna for any frequency above about 10 MHz. Even a 15 centimeter meter cable is an effective radiator at 100 MHz, which is right in the middle of the FM broadcast band. At 300 MHz, the critical length drops to about 5 centimeters, so that even the board traces themselves become significant radiators at frequencies above 300 MHz.

Both the board traces and the cables connected to the circuit board can radiate, as shown in Figure 6. The cable contribution is normally much more severe, due to the large physical size of the cable. One can use antenna theory to predict the electric field levels radiated by these unintended antennas. For board radiation, assume a small loop antenna, and for cable radiation, assume a larger dipole antenna. The formulas are as follows:

$E = 265 \times 10^{-16}\ IAf^2/r$ (Differential mode currents on the board)

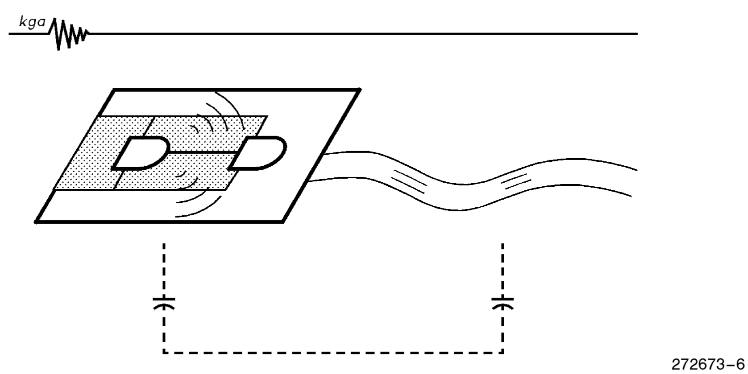
$E = 4 \times 10^{-7}\ IfL/r$ (Common mode currents on the cable)

Where E is the electric field intensity in Volts/meter, I is the current is amps, A is the "loop area" on the circuit board, L is the cable length, and r is the distance from the antenna.

For example, 1 milliamp of *differential mode* current at 100 MHz in a 1 square centimeter loop on the circuit board results in an electric field intensity of about 26 μV/m at a distance of 1 meter. For the same electric field intensity, only 200 picoamps of *common mode* current at 100 MHz would be needed on a 1 meter long cable for the same field intensity.

For this example, both of these levels are still well above typical automotive limits of 6 μV/m. First, it's obvious that if even a fraction of a percent of the PCB currents end up on the cables as common mode currents, we've got a serious emissions problem. Second, it's obvious that all high frequency currents (common mode and differential mode) must be controlled at the circuit board. There should be no doubt that the "hidden antennas" can cause serious EMI problems.

Physical Dimensions ***(Antenna Effects)***



Loop Radiation (Differential Mode)—On the board
—Signal Loops
—Power Loops
Cable Radiation (Common Mode)—Off the board

Figure 6

EMI DESIGN STRATEGIES

With all these potential EMI problems, it's apparent EMI protection must be designed in from the beginning. Unfortunately, there is no "magic solution" for EMI. Rather, it must be addressed throughout the design.

Different Approaches for Different Threats

Each of the EMI threats discussed earlier have preferred strategies. In many cases, one strategy may address several threats at the same time. In all cases, multiple levels of protection are needed.

Power Transients—The design strategies for this threat are to provide primary transient protection on all module input power lines, plus secondary protection such as filtering at the circuit level. The "load dump" transient is usually the biggest concern. Designing "noise tolerant" software is also very effective in controlling susceptibility to power line transients.

Radio Frequency Immunity—The design strategies for this threat are to keep the unwanted energy from reaching vulnerable circuits. This requires high frequency filtering on cables (both power and I/O) which act as antennas, plus careful circuit layout and circuit decoupling. Cable and module shielding are also effective, but are not popular in vehicular designs due to the costs.

Electrostatic Discharge—The design strategies for ESD are to limit *damage* by transient suppression or high frequency filtering on I/O and power lines, and to limit *upsets* by local filtering and decoupling, careful circuit layouts, and perhaps even shielding. Many of the design techniques for RF emissions and immunity work equally well for the "indirect" ESD effects due to the transient electromagnetic fields.

Power Line Fields—The design strategies are usually instrumentation oriented, and include local shielding and filtering of the most critical circuits. The design techniques necessary for RF emissions and immunity also minimize this threat. This is normally not a serious threat.

Radiated Emissions—The design strategies for this problem are to *suppress* the emissions at the source by careful circuit layout, filtering, and grounding, or to *contain* the emissions by shielding. For automotive designs, the emphasis is usually on suppression and careful circuit layout, since shielding is costly and difficult for most high volume automotive products. Nevertheless, shielding is seeing increasing use in vehicular applications.

EMI at the IC Level

As an integrated circuits vendor, we are often asked to make our circuits "more quiet" or "more rugged" in hopes of solving the EMI problems. After all, everyone always says to "suppress it at the source" or "harden at the victim." So why not just incorporate all the EMI fixes at the IC level, and be done with it?

If only life were so simple. While many equipment designers would like to push all the EMI responsibility back to the chip vendors, this approach is not very practical, due to constraints of chip speed, cost, performance, and wide temperature range. Lower EMI usually means slower speeds, which in turn means lower performance. Yet the trend in automotive ICs is toward higher performance devices with increasing clock speeds and edge rates.

Nevertheless, as chip manufacturers we are working hard to help control EMI at the IC level. For example, at Intel we've designed in the capability to turn off certain high speed control lines when not needed. We've redesigned clock drivers, and we've incorporated high frequency power and grounding schemes right on the silicon. We are working with a Society of Automotive Engineers Task Force to develop methods to measure and control EMI at the chip level. Our research into these areas continues. But while these efforts help, we can't do it all, and the real battle against EMI must still be waged at the circuit and module stage.

EMI at the PCB Level

This is where the most effective automotive EMI results can be achieved today. It's also where the designer has the most control. Many solutions at this level are inexpensive or even free, particularly when dealing with board layout and routing. Later in this application note, we'll provide some specific PCB design guides, but here is some general EMI advice.

First, plan for EMI, right from the start. Identify the most critical circuits, and decide what to do about them. Consider multi-layer boards, and allow for plenty of filtering and decoupling in the initial designs. You can always remove components later if you don't need them.

Second, stay involved with the design. Most good EMI designs are the result of the design engineer and the PCB layout specialist working closely as a team. As clock speeds and edge rates increase, this becomes even more important.

Third, test early and often. Unfortunately, EMI is not an exact science. Better to identity potential problems early in the design, when the alternates are inexpensive and plentiful. Late in the design cycle, EMI solutions may be expensive and painful.

EMI is a necessary part of any vehicular electronic design, and building in EMI suppression and hardness at the PCB level is very sensible engineering. Not only will you end up with an "EMI-proof" design, but you'll likely have a more reliable design as well.

EMI at the Cable and Interconnect Level

It is not usually cost effective to attack automotive EMI at this level. Cost and weight constraints generally preclude using shielded cables or expensive filtered connectors. Even external clamp on ferrites commonly used with personal computers are not practical. You can, however, provide cost effective filtering and transient protection at the I/O interface on the circuit board.

Fiber optics may change this in the future. Fiber optics can reduce and even eliminate many cable related EMI problems, but this approach is still too expensive for most automotive applications. Even when fiber becomes practical, however, the power wiring will still need EMI protection.

EMI at the Module Level

Like cable shielding, module shielding is often considered too expensive for automotive applications. Nevertheless, this approach should not be overlooked. It may be less expensive to shield and filter a module than to add a lot of components on the circuit board. This is particularly true if the module must be enclosed for environmental protection.

There are two simple secrets to success with EMI module shielding. First, seal all seams, and second, filter all penetrations. For most EMI problems, the material is not critical, and even thin conductive coatings provide very high levels of protection. For example, nickel paint on plastic typically provides 60 dB (a factor of 1000) or more of protection, and vacuum plating or electroless deposition on plastic typically provides 80 dB (factor of 10,000) of protection. A steel or aluminum box can provide well over 120 dB (factor of 1,000,0000) of high frequency shielding. These levels are easily attainable if the leaks (seams and penetrations) are sealed.

Good shielding need not be expensive. Consider the television-tuner, with its interlocking metal box and filtered input and output lines. This approach has been successfully used for almost fifty years, to provide cost effective EMI control in a highly cost sensitive industry. There is an increasing trend toward similar shielded modules in vehicles.

EMI at the Software Level

Although normally not effective against emissions (although there have been cases where it did make a difference), software can be very effective for EMI susceptibility. Like fault tolerance, you should build "noise tolerance" into your software.

There are two simple objectives—catch the errors before they upset the system, and then gracefully recover. It doesn't take much to provide this protection, and often times, just a few lines of code can work wonders.

To detect memory errors, add "checksums" to blocks of data in memory to tell when a memory state has changed. To detect program errors, add "tokens" to modules of code. Save the token on entering a module, and then check it on leaving the module. If they are not the same, an error has occurred, since the module was entered illegally. To detect I/O errors, do type and range checking on the data.

A "watchdog" can prevent becoming lost in an endless software loop. This is often a separate device, although many microcontrollers incorporate an internal watchdog. If the watchdog is not reinitiated in a predetermined time, it resets the entire system, bringing it back to a known state.

Many of these techniques are already used in vehicular applications with very good results. Often times they are incorporated for safety reasons, but they are also effective tools in the battle against EMI. For more details on software techniques, see the Bibliography.

EMI CIRCUIT BOARD GUIDELINES

Having examined problems and strategies, let's look at some specific solutions. We'll focus on circuit board issues, as this is where equipment designers have the most control. We'll share some details on how to attack potential EMI problems right at the root of almost all EMI problems—at the circuits and their interconnecting traces.

Guideline # 1—Identify Critical Circuits

The first step is to identify the most critical circuits on the circuit board. Experience shows that most EMI problems can be traced to a few key circuits. The good news is by identifying these circuits early, many EMI problems can be prevented with very little effort.

Emissions—The most critical circuits for *EMI emissions* are the highly repetitive circuits, such as clocks, address enables, and high speed data busses. Even signals with low repetition rates, such as address bit 0, can cause problems with sensitive automotive radio receivers. Consider adding a ferrite bead or small resistor (10–33 ohms) in series with any clock or other high speed output, right at the driving pin. This will help damp any ringing, and also helps provide an impedance match.

Both the signal traces, and the power traces are potential sources. In the latter case, remember a chip that is switching is consuming current in pulses, which can radiate just as well as signal current pulses. This means that any switched device is a potential source of emissions - not just the microcontroller. Figure 7 shows typical emission paths from critical circuits.

Since clock lines are critical, position the chips to minimize any clock runs. As previously mentioned, keep the clock traces and crystals away form any connectors. Route the high speed lines first, and keep those lines short and direct. Consider hand routing the critical lines, but if you use an autorouter, be sure to check to see where the lines have been routed.

Susceptibility—The most critical circuits for *EMI susceptibility* are the reset, interrupt, and control lines. The entire system can be brought to a halt if one of these lines are corrupted by EMI. Even though these circuits may have slow (or even nonexistent) repetition rates, they are still vulnerable to transients and spikes which can result in false triggering. Use high frequency filtering, such as small capacitors (0.001 μf typical) and ferrite beads (or 100 ohm resistors) to protect these lines. These components should be installed right at the input pins to the microcontroller.

Radiation and Coupling from Critical Circuits

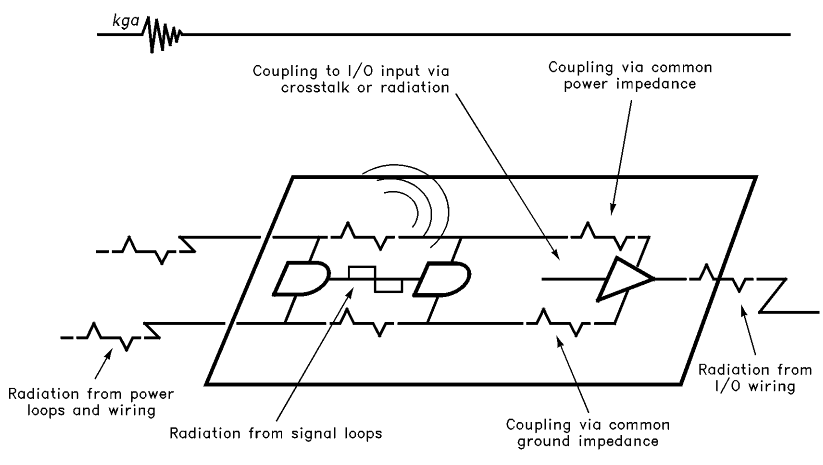


Figure 7

Be especially careful with the reset circuitry, particularly when using external devices for watchdogs or detecting power loss. Any false triggering of these external circuits can cause a false reset, so these external circuits must be protected against EMI as well. Once again, small capacitors and ferrite beads or resistors are very effective as filters against spikes and transients.

Guideline #2—Plan your board layout

After identifying the most critical circuits, the next concern is where to place the circuits on the board. Although this sounds trivial, EMI success hinges on how well this stage is implemented. Different circuits interact in subtle and unexpected ways, so you must plan your layout.

Group the circuits by speed of operation. While this seems obvious, this simple principle is often overlooked with devastating EMI results. Figure 8 shows an example of a well partitioned board. Note that in this example, the high speed digital circuits are separated from the lower speed digital and analog circuits. Furthermore, the high speed circuits are physically separated from the I/O connectors to minimize parasitic high frequency coupling.

Use special care with the input-output circuits, since they connect to the outside world. Even though most automotive I/O circuits operate at relatively low frequencies, the cables still act as antennas for high frequency energy. A key concern here is location. A common problem is placing a clock or crystal next to an I/O port, which results in parasitic coupling to the I/O wiring. A similar problem is routing reset or interrupt lines next to I/O lines, which allow them to pick up transients from the outside world. Keep these components and traces at least 25 mm (1 inch) away from any input/output or power connectors and wiring.

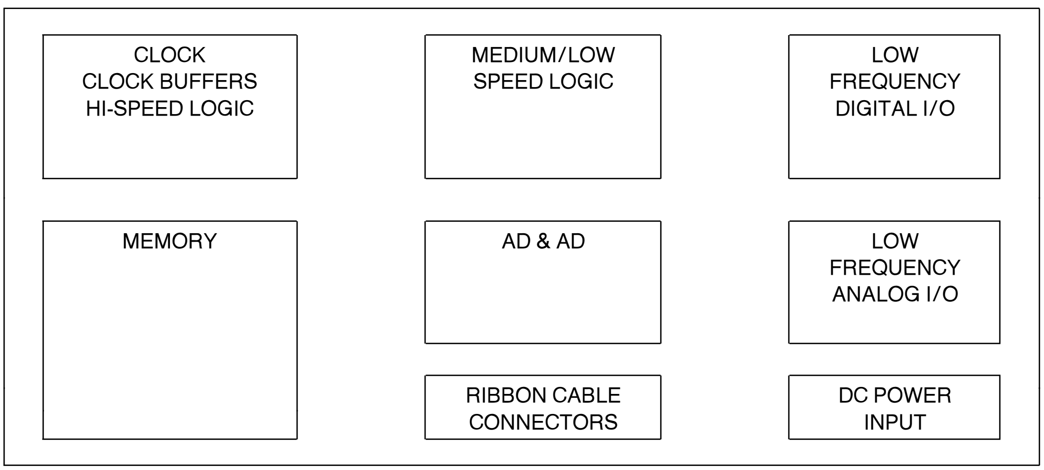


Figure 8. Board Partitioning

Guideline #3—Select an appropriate board type

The type of board has major impact on EMI issue, both emissions and susceptibility. For years, automotive designers have used two layer boards due to cost constraints. That trend is changing, and many new higher speed designs now use multi-layer boards. There is no doubt that multi-layer boards minimize and often eliminate EMI problems. Yet many simpler designs can be still implemented with two layer boards, if proper design techniques are used.

Multi-Layer Boards Preferred—Multi-layer boards offer several EMI benefits. First, the power and signal "loop areas" are minimized, both reducing emissions and increasing immunity at the board. Second, the power and ground impedance levels are lowered (often by several orders of magnitude) which reduces power and ground perturbations. Third, the presence of power and ground planes greatly minimize crosstalk between traces. As a result, multi-layer boards typically offer ten-fold to hundred-fold EMI improvements over two layer boards.

The multi-layer miracle occurs because of the "image-plane" effect. Place a current carrying wire close to a metal surface, and most of the high frequency current returns directly under the wire. (At high frequencies, this path has the minimal loop area, and thus minimal inductance.) A transmission line is formed by the wire's "mirror image" located over the metal surface. With equal and opposite currents, these transmission lines do not radiate well, nor do they pick up external energy. This is shown in Figure 9.

Note that both the power and ground planes act as "signal ground" planes at high frequencies. Since they are decoupled through bypass capacitors and their own capacitance between the planes, they are actually at the same potential at high frequencies. For signal purposes, both the power and ground planes are treated in an identical fashion.

The paths under the traces (in both the power and ground planes) must be continuous and unbroken. If a plane is cut, or if someone decides to "borrow" some of the plane area for trace routing, the return currents are forced to divert around the cut or break, creating a loop. Watch out for this problem in connector cutout areas, as currents may be forced to divert around the cutout. This is particularly troublesome, as this "hot spot" can result in increased coupling to or from the attached wiring.

Some Two Layer Techniques—Due to cost concerns, two layer boards are still widely used in automotive designs. In spite of their EMI disadvantages, some two layer boards can be designed to approach multi-layer EMI performance. For example, by routing critical lines with dedicated returns, and by routing all power traces as power/return pairs, "loop areas" can be minimized, just as with a multi-layer board. Filling in unused areas with "groundfill" helps lower the ground impedance.

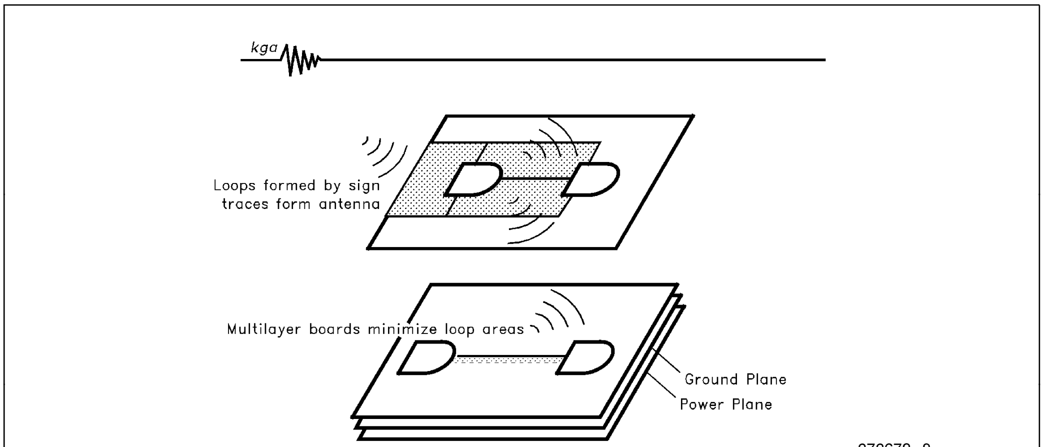


Figure 9. Image Plane Effects

One technique is to use one layer as a dedicated ground plane. You don't need multiple planes to benefit from the "image-plane" effect. In fact, on two layer boards, allocating one side as a dedicated ground plane produces benefits almost as good as a multi-layer board. This works best on boards with low routing densities.

As an alternate to a dedicated plane, ground grids can be used on two layer boards. One method accomplishes this by running horizontal ground traces on one side of the board, and vertical ground traces on the other side. They are connected together at the crossover points with vias. In this way, both surfaces may be used for routing, and yet a ground grid is provided for the high frequencies. Although not as good as a plane, the grid approach is still much more effective than random trace routing.

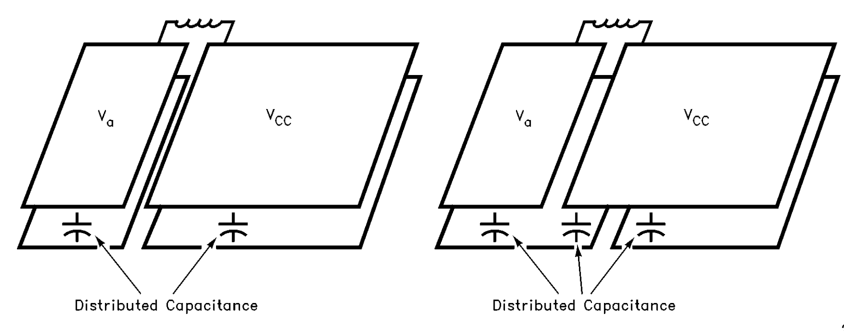
Even with care, however, it's difficult to achieve EMI success with two-layer technology on designs with clocks above about 15 MHz. As a result, we recommend multi-layer boards for new, high speed microcontroller designs.

Design Guideline #4—Isolate With Care

Isolated, or "split" planes have become very popular as a method of maintaining high frequency isolation on a circuit board. This technique has been used for years on boards with mixed analog and digital circuits, and we strongly recommend it for those applications. In other applications, Intel recommends using extreme care with this method. It is a useful technique, but if you do a poor job of isolating or segmenting the planes, you may end with more problems that if you stayed with solid planes.

Two errors must be avoided. First, make sure the corresponding planes are aligned, as shown in Figure 10. Failure to do so will allow high frequency energy to couple at the overlapped areas. Second, make sure the signal traces are continuously routed over the appropriate return plane. If the traces go "over and back", you create loops that couple energy between the isolated areas.

Use care at any signal or power interfaces between the isolated areas. If high frequencies must pass between the two areas, the power and ground points should be joined at a narrow "bridge", and the signal traces should be routed over this bridge. If high frequencies do not need to pass between these two areas, then the connecting traces can be isolated with ferrites, inductors, or common mode chokes. These are shown in Figure 11.



• Plane isolation is defeated by overlapping unrelated power and ground planes
• Layer capacitance couples all surfaces at high frequency

Figure 10. Align the Planes

Micro-Island—This term was coined by the EMI consulting firm of Kimmel Gerke Associates to describe an isolation design technique that has proven effective in controlling EMI in embedded control systems. Properly implemented, it provides many of the benefits of a multi-layer board even when using two layer boards. The technique typically yields ten fold reductions in radiated emissions over standard two layer boards. The technique also works with multi-layer designs, but they are generally more quiet in the first place.

For many automotive electronic systems, the embedded microcontroller is the only high speed source of EMI on the board. If one can confine that high frequency energy to a small area (Micro-Island) on the board, EMI emissions can be minimized. This is accomplished by providing a *local ground plane under the high speed circuitry* (typically the microcontroller and perhaps RAM or ROM memory devices), and then *filtering every trace* (signal, power, and ground) entering or leaving the isolated island. A small shield could be added to completely encapsulate the island for even higher levels of suppression, although this is rarely needed.

Here's how to create your own Micro-Island. See Figure 12 for details.

— First, define the boundaries of the island to encompass all high speed circuitry (microcontroller, crystal, RAM, ROM, etc.) Fill this area with a ground plane.

— Second, isolate every signal entering or leaving the island with a T-filter (ferrite-capacitor or resistor-capacitor). The capacitors are connected to the ground plane through a short lead.

— Third, isolate every power and ground trace with a series ferrite bead. Decouple the power and ground with a 0.01 μF capacitor at the capacitor energy point.

— Fourth, any signal not starting or ending on Micro-Island must be routed around the island. Later in this application note, we'll share some test results of this technique.

Note that this example actually shows two islands—one for analog and one for digital. If you are not using low level analog signals, one island is sufficient.

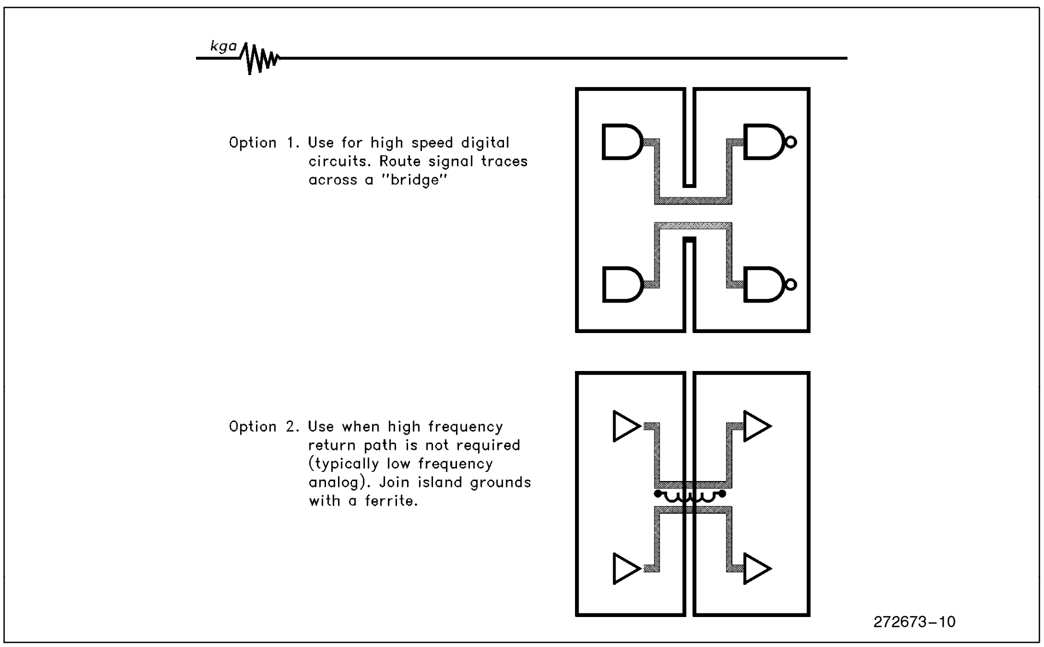
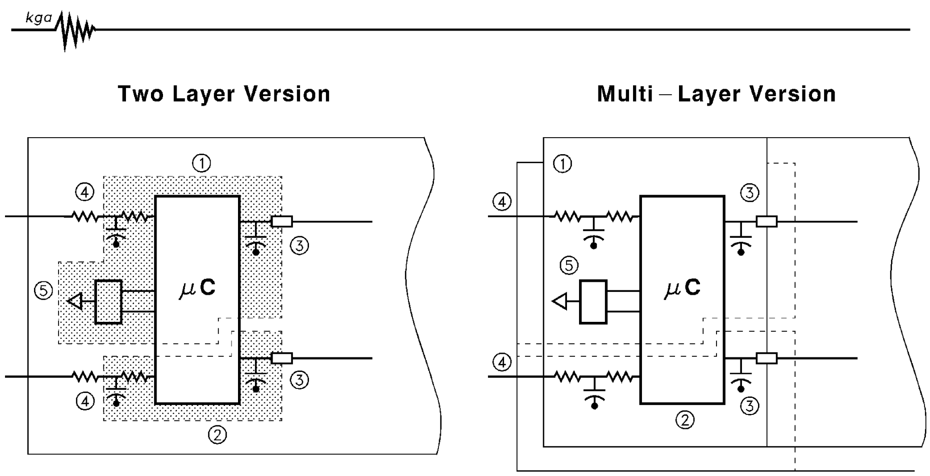


Figure 11. Routing Between Islands



1. Dedicated Digital Power/Ground Planes
2. Dedicated Analog Power/Ground Planes
3. Digital and Analog Power Decoupling
4. I/O Decoupled with Ferrite/Resistor and Cap
5. Grounded Crystal or Resonator

1. Ground Plane Under Micro-Controller
2. Separate Analog Ground Plane
3. Power Decoupled with Ferrite and Cap
4. I/O Decoupled with Ferrite/Resistor and Cap
5. Crystal or Resonator over Digital Ground Plane

Figure 12. Micro-Island

Guideline #5—Decouple the power

Many high frequency radiated emissions are caused by poor power decoupling. While everyone intuitively understands that high speed *signals* cause EMI, some forget about the power perturbations. Whenever a digital circuit switches, it also consumes current at the switching rate. These pulses of power current will radiate just as effectively as pulses of signal current. In fact, they often cause more radiation, since the power current levels are usually much higher than those on an individual signal line.

High speed CMOS devices are particularly vexing, since they exhibit high peak currents due to the momentary "short" across the power rails when the CMOS devices switch. In fact, CMOS peak currents are often higher than other technologies, so emissions may actually go up when a CMOS device (such as an 80C31) is used to replace an HMOS (8031) device, even though the average power is much lower with CMOS. It's the peak current, not average power, causing EMI emissions. The solution is to improve the power decoupling.

Circuit decoupling—We recommend local power decoupling of every integrated circuit on the board. For devices with multiple power and ground pins, each pair of pins should be decoupled. High frequency capacitors in the 0.01–0.1 μf range should be installed as close as possible to the device. For multi-layer boards, run a short trace from the power pin to the capacitor, and then drop the other lead into the ground plane. For two layer boards, "fat" traces (with a length to width ratio of 5:1 or less) should be used on both the power and ground sides of the capacitor to minimize inductance. In both cases, keep the leads as short as possible.

Additional protection can be provided by inserting a ferrite in series with the V_{CC} line to the microcontroller. This must be installed on the V_{CC} side of the capacitor, not on the IC side. This small LC filter further isolates the V_{CC} traces from current demands of the switched device. We strongly recommend this technique for two layer and Micro-Island designs; it's optional for multi-layer designs.

Power entry decoupling—We recommend high frequency decoupling at the power entry points. In addition to the standard 1–10 μf "bulk" capacitors, add a 0.01–0.1 μf high frequency capacitor in parallel at the power entry point. Due to internal resonances, the bulk capacitors are useless at frequencies above about 1 MHz. The high frequency capacitors are there to intercept any high frequency energy that tries to sneak out the power interface. For more protection, series ferrites can also be added.

Be sure to keep the leads short on the decoupling capacitors. The self-inductance of wires and traces is about 8 nh/cm (20 nh/inch), so even a few millimeters of wire length can defeat the decoupling at high frequencies due to the inductance. Figure 13 gives several examples of how lead inductance defeats the decoupling capacitor. Note that once you are above the resonant frequency, using a larger capacitor provides no additional benefits, as the inductive reactance prevails.

One more power decoupling hint. Add high frequency capacitors (0.001 μf typical) to the input and outputs of all on-board voltage regulators. This will protect these devices against high levels of RF energy (which can upset the feedback) and will also help suppress VHF parasitic oscillations from these devices. Keep the capacitors close to the devices, with very short leads.

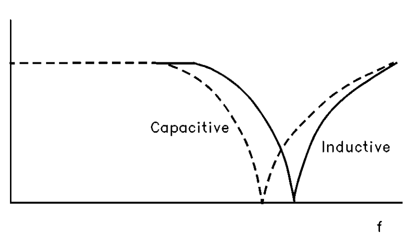
Guideline #6—Bulletproof the interfaces

We've already seen that cables and wiring can act as unintended antennas, and that even low levels on relatively short lengths can still cause EMI problems. Thus, we strongly recommend that you plan for filtering at your power and signal connections to the module. You may find that you don't need all of the filtering, but you can always remove components depending on EMI test results.

Power Interfaces—We already discussed high frequency protection at the power inputs. A few high frequency capacitors here is very cheap insurance, and they are often needed for automotive designs to meet the extremely low emission requirements. For immunity, additional low frequency filtering may also be needed, plus transient protection or energy storage for the automotive power transient requirements. Typically, meeting the "load dump" transient satisfies the other transient requirements as well.

Signal Interfaces—These need some protection, too. Don't assume that just because an interface is "slow" that high frequency energy won't try to enter or leave the system at that point.

kga



Capacitance (pF)	Reasonant Frequency (MHz) 1/2 inch leads	1/4 inch leads
10,000	16	23
1,000	51	72
500	72	100
100	160	230

272673–12

Combination capacitance plus internal and external lead length:
Larger capacitor moves resonant frequency down without providing any high frequency improvement

Figure 13. Capacitor Resonance

As a minimum, provide the space and pads for high frequency shunt capacitors and/or transient protection on every line. Better yet, provide for a series resistor as well. These can be "zero-ohm" resistors connected as traces between two pads. Cut and replace with actual resistors if necessary.

Don't overlook the ground leads in the signal interface, as these can provide sneak paths for common mode currents into and out of the system. We recommend adding a small ferrite bead in the ground lead, to complete the filtering of the interface.

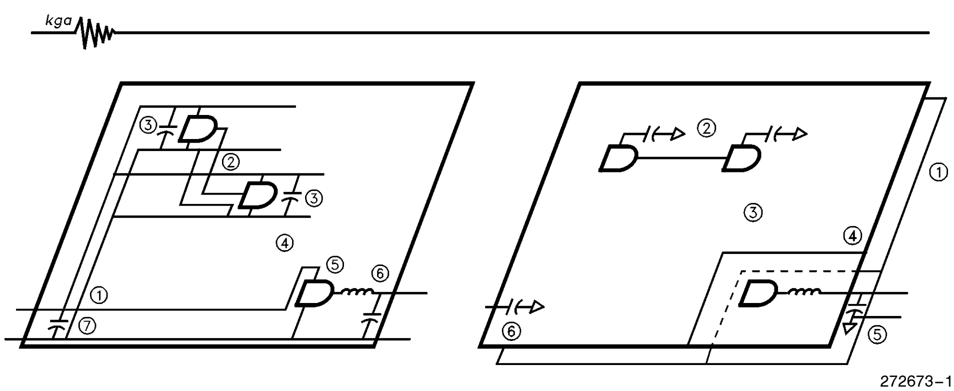
Here are some additional recommendations on how to ground shunt capacitors if filters. To contain emissions, connect the capacitor to the signal ground. The objective is to keep these currents on the board. To enhance immunity, connect the capacitor to a chassis ground (if available), not signal ground. The objective here is to keep the offending currents off the circuit board.

If there is no chassis ground, then a compromise is necessary for immunity. Connect the capacitor to the signal ground, PLUS add a series element (ferrite or resistor) to limit the EMI current shunted into the signal ground. This is mandatory for ESD—if not used, the ESD current will likely "bounce" the ground with upset or damage as the result.

Guidelines Summary

Figure 14 summarizes some EMI-quiet circuit board solutions. By using these simple techniques, many EMI problems can be minimized or eliminated.

Some EMI-Quiet PCB Solutions



GOOD
- 1. Parallel Power/Ground Traces
- 2. Parallel Signal/Return Traces
- 3. Power Decoupling at Chip
- 4. Separation from I/O
- 5. Separate I/O Power
- 6. High Frequency Filter on I/O
- 7. High Frequency Capacitor on Power

BETTER
1. Multi-Layer Board
2. Power Decoupling at Chip
3. Separation from I/O
4. Isolated I/O Power/Ground Plane
5. High Frequency Filter on I/O
6. High Frequency Capacitor on Power

Figure 14. Summary

INTEL SPONSORED TEST PROJECT

In 1994, the Intel Automotive Operations in Chandler, Arizona, commissioned a research project to investigate the effects of different printed circuit board techniques on radiated electromagnetic interference emissions. The project involved laboratory testing of several design variations of a "typical" automotive electronics module using the Intel 80C196KR microcontroller. The emphasis was on practical, low cost solutions that could be used by Intel automotive customers.

Test Methods and Procedures

The primary objective was to test several different circuit board configurations for RF emissions in the 30–1000 MHz range. The test procedures were based on the "module level" radiated emissions tests of GM9100. These procedures are used by General Motors to qualify electronic modules supplied by GM vendors, and are aimed at minimizing interference to vehicular radio receivers when the modules are installed in the vehicle. The test levels are very stringent, with levels of 15 dB μV/m (6 μV/m) at 1 meter over most of the frequency range of interest. These levels are 50 to 100 times tougher to meet than those for personal computers.

Six board configurations were tested. All were based on an actual ABS module design. The representative boards were populated with an 8XC196JT microcontroller, regulator, load dump diode, and hex buffers. The fail-safe ASIC and other circuitry was simulated by placing appropriate capacitor and resistor loads on the test board. The component placement on the test boards was approximately the same as a fully populated ABS module. This was done since it imposed realistic PCB routing constraints on the test boards, and provided realistic coupling paths on the test boards. Actual production connectors and cables were also installed during the tests.

The six test configurations were as follows:

1. Standard Two Layer Layout
2. Two Layers with Micro-Island Isolation—"Poor Implementation"
3. Two Layers with Micro-Island Isolation—"Better Implementation"
4. Standard Four Layer Layout
5. Standard Four Layer Layout with Micro-Island Isolation
6. Customer Supplied ABS Module—Partially Populated

Note that a key layout technique studies was the "Micro-Island" technique discussed earlier. While this is not a new concept, this method is a practical technique to isolate high frequency sources to one portion of the circuit board. The intended effect is to eliminate high frequency escape paths to the rest of the board and the interconnecting cables. This high frequency isolation was achieved by placing the control (80C196JT) and fail-safe microcontrollers on an island, and then filtering all traces bridging the island with ferrites and capacitors.

Micro Island Anomalies—Note also that two versions of the "Two-Layer Micro-Island" configuration were tested. Due to layout routing difficulties, the PCB designer compromised the micro-island technique by "cutting up" the ground plane to route needed traces. The PCB designer believed, erroneously, that taking a few traces from the ground plane area would have minimal impact. As is often the case in real designs, there was no easy way to route all the traces without "borrowing" from the ground plane.

Rather than give up on the Two Layer Micro Island approach, the cuts were bridged with high frequency capacitors. The objective was to create a high frequency "grid" to minimize the effects of the cuts for the traces. While simple wire jumpers could have been used, capacitors were chosen since this technique is often used to "stitch" planes of different voltages together at high frequencies. As it turns out, these two configurations yielded some very significant test results.

Test Results & Conclusions

Figures 15–19 show the emissions from the various test module configurations. The tests showed primary sources of emissions were clock harmonics, as expected. At frequencies below 150 MHz, the interconnecting cable was the primary antenna. At frequencies above 150 MHz, both the cable and the board contributed to the radiation. This is also consistent with expectations, given the wavelength and dimensions of the cables and board traces.

Here are some conclusions based on the test results from the different board configurations:

— The standard two layer board had the poorest performance, and would likely fail GM9100.

— The four layer micro-island board was the best, and would likely pass GM9100.

— The four layer standard board would also likely pass GM9100. However, it had higher emissions above 150 MHz than the four layer micro island board.

— The two layer micro island board without the capacitors would likely fail GM9100, but would likely pass with the addition of the auxiliary capacitors and jumpers.

The experiments showed some other significant results, as follows:

— The four layer standard board was 10-25 dB more quiet than the two layer standard board.

— The properly executed two layer micro island board approached the results of the four layer micro-island board.

— The poorly executed two layer micro island board was almost no better than the standard two layer board. Cutting the plane essentially destroyed the micro-island protection.

— A local shield positioned over the microcontroller yielded 6 dB reductions. This was believed to be due to reduced capacitive pickup of energy by the external cable

Additional near field tests were done on these test boards using strip lines, loop probes, and the EMSCAN® board measurement system. While more frequency components were noted in the near field tests, it was difficult to correlate these results with the far field data.

In conclusion, the multi-layer board performed much better than the same layout on a two layer board. The micro-island approach is a viable solution, but it must be properly implemented. Finally, achieving the low emission levels necessary for automotive applications is possible, but difficult, with little room for design errors.

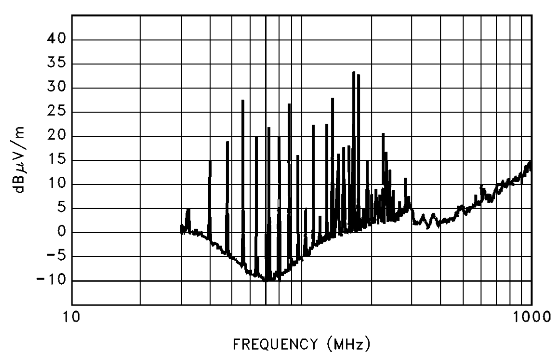


Figure 15. Two Layer Standard

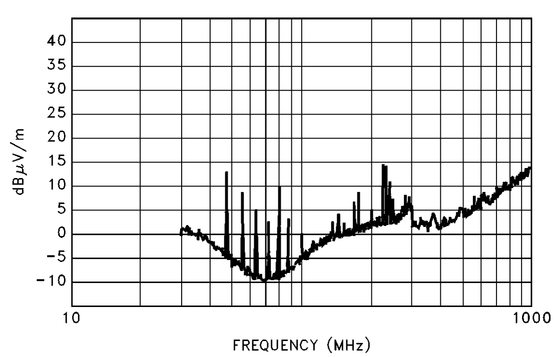


Figure 16. Four Layer Standard

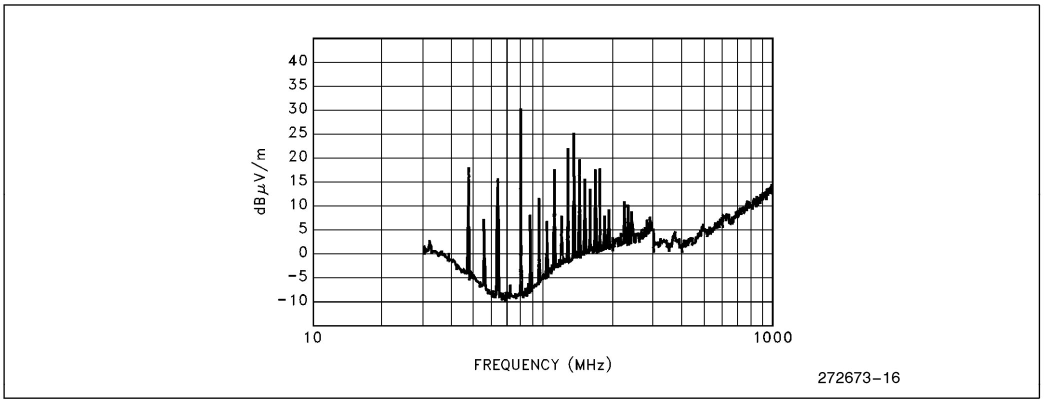


Figure 17. Two Layer Micro-Island (Cuts in Ground Plane)

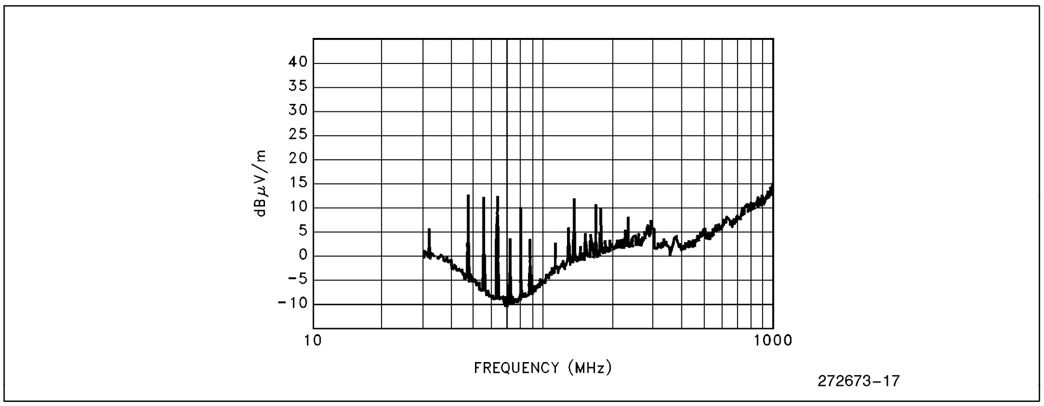


Figure 18. Two Layer Micro-Island (Capacitors to Repair Cuts)

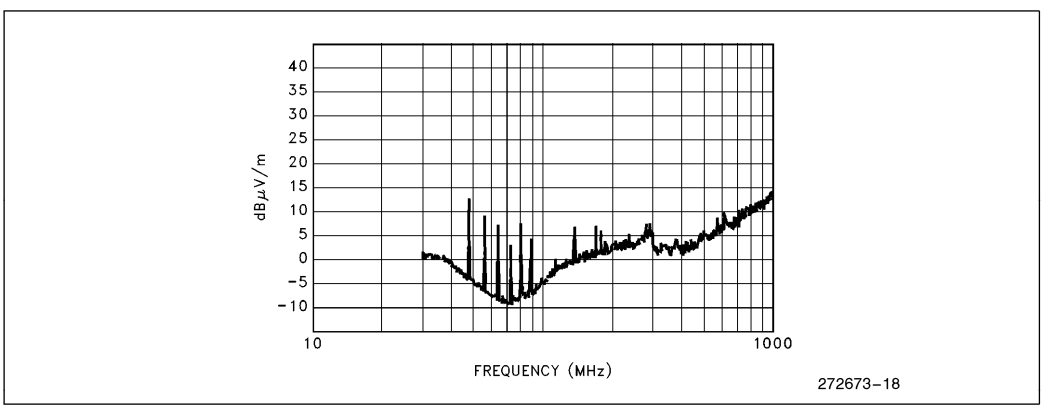


Figure 19. Four Layer Micro-Island

SUMMARY

Automotive EMI problems are harsh. The susceptibility levels are often high, and the radiated emission levels are extremely low. The emission levels are particularly grueling, as they require suppression to levels up to 1000 times below commercial designs, and up to 100 times below those for personal computers. Unfortunately, there are no simple solutions to these problems.

We at Intel are committed to helping you solve these problems. We'll continue our research and development at the chip level, doing what we can to control the EMI problems at that level. It should be apparent, however, we can't do it all. EMI control must also be addressed at the circuit board and module levels. We'll continue our efforts at these levels, too.

For additional help with these problems, we invite you to contact your Intel or Distributor Applications Engineers. Many have received introductory EMI training, and may be able to help you with basic questions. For more involved problems, they can refer you to EMI design experts.

Acknowledgments—The main sources of the information for this application note are listed in the Reference section. Daryl Gerke, PE., of Kimmel Gerke Associates Ltd., was responsible for supplying much of this information, and conducting the Intel sponsored test program. His firm specializes in EMI design, troubleshooting, and training courses. Figures 1–14 are from the firm's EMI training courses, and are used here courtesy of Kimmel Gerke Associates Ltd. Mr. Gerke can be reached in St. Paul, Minnesota, at 612-330-3728.

REFERENCES:

1. Gerke, D.D. and Kimmel, W.D., *EDN's Designer's Guide to Electromagnetic Compatibility,* Cahners Publishing, Newton, MA, 1994
2. Williamson, T.W., *Designing Microcontroller Systems For Electrically Noisy Environments,* Intel Application Note AP-125, 1982
3. Gerke, D.D. and Kimmel, W.D., "Focus on Automotive EMI", *Kimmel Gerke Bullets,* (EMI Newsletter) Vol. 2, No. 4, Kimmel Gerke Associates Ltd., 1991
4. Kimmel, W.D. and Gerke, D.D., "Microprocessors and VHF Radios—Mutual Antagonists, EMI *EXPO 1989 Symposium Record,* pp. C6.9-C6.22, August 1989
5. Gerke, D.D., "Designing Microcomputer Systems to Tolerate Noise", *Society of Automotive Engineers Transactions 870787,* April 1987
6. Yarkoni, B. and Wharton, J. "Designing Reliable Software for Automotive Applications," *Society of Automotive Engineers Transactions 790237,* February 1979
7. SAE J1113, *Electromagnetic Susceptibility Measurement Procedures for Vehicle Components,* Society of Automotive Engineers, August 1987
8. SAE J1752 (Draft), *Electromagnetic Compatibility Measurement Procedures for Integrated Circuits,* Society of Automotive Engineers, August 1994
9. GM9100P, *Automotive Component EMC Specification,* General Motors
10. Gullett, C. "The Hidden Schematic", *Common Ground,* (Intel Automotive Newsletter) Volume 1, Number 5, Intel Corporation, February 1993.

APPENDIX A
Automotive EMI Test Techniques For The Design Engineer

EMI testing can be complex and expensive. Furthermore, it takes many years of experience to develop a high level of EMI test expertise. As a result, most designers send their products to an EMI test laboratory to demonstrate compliance to the appropriate EMI test requirements.

This testing is often done near the end of a design project, so if problems occur, they can be painful and expensive to fix. Fortunately, there are a number of tests that can be done during the design phase that can identify potential problems when they are still easy to fix.

The tests we'll discuss here are *engineering* tests, and not *compliance* tests. As such, high degrees of quantitative accuracy are not necessary. The objective of these *engineering* tests is to improve the probability for success of the eventual *compliance* tests. The goals are to uncover problems early, and to demonstrate design improvements. Here are some comments on EMI tests that you should consider during the design phase.

Power Transients—Several test equipment manufacturers offer test systems that generate the automotive transients described in SAE J1113 *(Electromagnetic Susceptibility Procedures for Vehicle Components)*. Be sure the test system includes the "load dump", which is a very severe transient. Most design engineers can run this test in their engineering lab with the appropriate equipment.

Power Line Electric and Magnetic Field Immunity—Since this is rarely a problem, it is probably not worth engineering tests. You can do these tests in an engineering lab if you really insist on it, using Helmholtz coils for the magnetic fields, and parallel plates for the electric fields.

Electrostatic Discharge—Several test equipment manufacturers offer ESD test systems. A good guideline for this test is IEC 801.2 *(Electromagnetic Compatibility for Industrial Process Control Measurement and Control Equipment—Electrostatic Discharge Requirements)*. This widely used test method is based on the "human model" for ESD, and is easy to do in the engineering lab. Be sure and do both the "direct" and the "indirect" ESD tests as described in the 1991 version of IEC 801.2

Radio Frequency Immunity—Full comprehensive RF immunity tests can be difficult, since they often require antennas, amplifiers, and shielded rooms. Small modules, however, can be tested in a "TEM" cell, which is a special test fixture that is a piece of expanded transmission line. The cell is completely shielded, so you don't need an antenna or shielded room. You still need a signal generator and a power amplifier to develop the appropriate test levels. A modified version of the TEM cell, known as a GTEM!* cell has proved popular for these applications.

For frequencies below 100 MHz, special probes may be used to inject RF energy directly on the cables. This can be useful, since at frequencies below 100 MHz, the cables are the most likely antennas for picking up the RF energy.

Crude RF immunity tests can be done by keying small VHF/UHF hand held radios near the equipment under test. At 1 meter, a 1 watt hand held radio generates an electric field of about 5 volts/meter. At 1 foot, that increases to about 15 volts/meter. At less than one foot, the levels are not as meaningful, since the unit under test is the "near field". Keep in mind these radios only transmit on select frequencies. Nevertheless, if a failure occurs, you know you have problems.

Radio Frequency Emissions—This can also be difficult in an engineering lab, since full tests often require a shielded room, antennas, and sensitive spectrum analyzers or EMI receivers. The automotive test thresholds are several orders of magnitude below commercial limits, so a shielded room is almost mandatory. In some cases, a TEM cell or "strip line antenna" can be used, but the correlation with final levels can be difficult. These methods are useful, however, for making relative measurements, such as assessing design changes.

Two useful troubleshooting tools for emissions are current probes and "sniffer" probes, which are connected to a spectrum analyzer. The former are single turn current transformers that are clamped over a cable to measure the high frequency current on the cable. The latter are magnetic loop antennas that show the presence of high frequency magnetic fields (and thus currents) in a circuit or cable. While useful, you can not correlate these measurements with actual emission measurements, since they only measure the current, and do no account for the antenna effects.

A relatively new system for testing circuit boards for emissions is the EMSCAN* system. The board under test is scanned for both frequency and location. The results can be plotted to show RF "hot spots", similar to thermal hot spots. Like the current probes and sniffer probes, however, the measurements do not correlate with final emission levels, since antenna effects are not included. Nevertheless, many manufacturers have found this system useful in designing RF quiet boards.

Finally, crude emission tests can be done by placing the antenna of an FM or VHF radio receiver near the unit under test. A typical test distance would be 1 meter away, using a representative antenna such as a vertical "whip." If you can hear emissions on the radios, you are probably above the limits.

Summary—Engineering level tests will never replace final EMI compliance tests, but they can still prove very useful. Here are two final notes of advice. For immunity testing, you'll likely need to add light emitting diodes or other devices to indicate failures, as external equipment may mask the test results. For emissions testing, pay attention to software and be sure to exercise all peripherals to assure that maximum noise levels are generated.

INTEL CORPORATION, 2200 Mission College Blvd., Santa Clara, CA 95052; Tel. (408) 765-8080

INTEL CORPORATION (U.K.) Ltd., Swindon, United Kingdom; Tel. (0793) 696 000

INTEL JAPAN k.k., Ibaraki-ken; Tel. 029747-8511

Printed in U.S.A./xxxx/0296/B10M/xx xx