EMC DESIGN GUIDELINES

A little planning prior to your EMC testing - or even better during the design phase - can ensure better testing results. The following guidelines should help you eliminate some common EMC headaches:

Identify the Noise Sources
A very important general rule is that all types of noise should be handled as close to the source as possible, and as far away from the sensitive parts of a circuit as possible. This, of course, means that the task of identifying these sources is very important.

Transmitted Noise
In many microcontroller systems, the microcontroller is the only fast digital circuit. In such systems, the most important internal noise source is the microcontroller itself, and the resources used for preventing conducted and emitted RF are best used close to the microcontroller. This will reduce the amount of RF energy that reaches I/O cables and other parts of the system that may act as transmitting antennas.

Received Noise
The sources of received noise are usually outside the system, and therefore out of reach for the system designer. The environment is what it is, and the first possibility for the system designer to do something about the noise is on the system inputs and on the power cables. For a system delivered with dedicated cables, it is even possible to start on the cable itself. A good example here is a computer monitor, where you quite often see a filter put next to the VGA plug you connect to your computer. On other systems, the first chance comes with the I/O connectors. For a hand-held, battery-powered application without any cables, this is not applicable, but then this problem is similarly smaller. If external noise can be prevented from entering the system at all, there will be no immunity problems.

The Path to Ground
The best way to avoid noise problems is to generate no noise in the first place, but this is not always possible. Most kinds of noise are side effects of intended behavior of other parts of the system, and therefore cannot be avoided. All kinds of currents, AC or DC, high-power or low-power, signals or noise, are always trying to find the easiest path to ground. The basic idea behind many EMC design techniques is to control the path to ground for all signals, and make sure that this path is away from signals and circuits that may be disturbed. For transmitted noise, this means making sure that the noise will find a path to ground before it leaves the system. For received noise, it means making sure that the noise will find a path to ground before it reaches sensitive parts of the system.

System Zones
Handling every EMC problem at once is a very complex task. It is therefore a good idea to split the System into smaller subsystems or zones, and handle these individually. The zones may, in some cases, only be different areas of the same PCB. The important part is to have control of what happens inside one zone, and how the zones interact. For each zone, the designer should have some idea about what kind of noise the zone may emit, and what kind of noise it may have to endure. All lines going in and out of a zone may require some kind of filter. It is also very important to be aware about how noise may be radiated from one zone to another. Local shielding of very noisy and/or very sensitive circuits may be necessary. The split may be done in two ways or a combination of these:

- When possible, it is good to keep I/O and communication sections separate from the AC or DC section of the PCB.

- Keep power cables segregated from I/O and data/communication cables.

RF Immunity
Long I/O and power cables usually act as good antennas, picking up noise from the outside world and conducting this into the system. For unshielded systems, long PCB traces may also act as antennas. Once inside the system, the noise may be coupled into other, more sensitive signal lines. It is therefore vital that the amount of RF energy allowed into the system be kept as low as possible, even if the input lines themselves are not connected to any sensitive circuit. This can be done by adding one or more of the following:

- Series inductors or ferrite beads will reduce the amount of HF noise that reaches the microcontroller pins. They will have high impedance for HF, while having low impedance for low-frequency signals.

- Decoupling capacitors on the input lines will short the HF noise to ground. The capacitors should have low ESR (equivalent series resistance). This is more important than high capacitance values. In combination with resistors or inductors, the capacitors will form low-pass filters. If the system is shielded, the capacitors should be connected directly to the shield. This will prevent the noise from entering the system at all. Special feed-through capacitors are designed for this purpose, but these may be expensive.

- Special EMC filters combining inductors and capacitors in the same package are now delivered from many manufacturers in many different shapes and component values.

ESD and Transients

Handling ESD is usually quite simple: make sure that the user cannot touch the sensitive parts of the system. This is, in most cases, taken care of by the equipment enclosure and only I/O pins leaving the system need special attention. However, ESD discharges may induce currents in nearby paths, causing incorrect values of the signals. Keep in mind that both ESD pulses and other types of transients are very high-frequency phenomena, and that stray capacitance and inductance have a very important influence of their behavior. A transient on one line may also affect the behavior of other signals nearby. The important thing is to make sure that the most efficient path to ground is one that does not affect the system. If, for instance, the most efficient path to ground for an ESD pulse is along the I/O line, to the microcontroller pin, through the ESD protection diode and then to ground, a logic high input may be read as low. If the system software cannot be made to handle this (and that is usually the case), the system requires some kind of hardware that will create a more controlled path to ground. The RF filters listed above can also work on ESD and transients, and may, in some cases, be sufficient. But reducing a 4 kV spike to a 4V spike requires a very strong filter. It can be done by large series resistors, but that is not always an option. Overvoltage protectors are sometimes a better solution. There are many types of these, most of them acting as very fast zener diodes. They will have very high impedance to ground as long as the I/O line voltage is within the specified limits, but will switch to a very low impedance value when the voltage is too high. A transient is then very effectively shorted to ground. MOV's at the Mains input is a cheap and very successful solution to prevent failure or damage from Surge transients.

Power Supply, Power Routing and Decoupling Capacitors

One of the most common reasons for EMC problems with microcontroller products is that the power Supply is not good enough. Correct and sufficient decoupling of power lines is crucial for stable microcontroller behavior, and for minimizing the emitted noise from the device. Looking at the datasheet for a microcontroller, one can be fooled to believe that power supply is not critical. The device has a very wide voltage range, and draws only a few mA of supply current. But as with all digital circuits, the supply current is an average value. The current is drawn in very short spikes on the clock edges, and if I/O lines are switching, the spikes will be even higher. The current pulses on the power supply lines can be several hundred mA if all eight I/O lines of an I/O port changes value at the same time. If the I/O lines are not loaded, the pulse will only be a few ns. This kind of current spike cannot be delivered over long power supply lines; the main source is (or should be) the decoupling capacitor. For those products that use external power supplies, using an approved supply is critical for the Safety requirements and certainly helps with the chances of passing the EMC requirements. You must keep in mind that using an approved supply does not keep you from performing the AC tests on the supply. If you supply the power supply with your product or sell it as an option, then you are responsible for the tests to that supply while attached to your product.

Shielding

In some cases it is not possible to get the noise levels of a system low enough without adding a shield. In other applications a shield may be used because it is easier to use a shield than to achieve low noise levels by other means. Depending on the application, the shield may cover the whole system or only the parts of the system that need it most. If the zone system is used in the design, it is easy to determine which zone(s)need to be shielded. In either case, the shield must be completely closed. A shield is like a pressurized container: almost good enough is as bad as nothing at all. As described earlier, all lines entering or leaving a zone need to be filtered. A single line that is not filtered will act like a single hole in a bucket of water. It will cause a leak. A semi-closed shield, connected to ground, may still reduce noise. It will act as a ground plane, reducing the size of the loop antennas. A common rule of thumb says that the maximum dimension of any mechanical slit or hole in the shield should be less than 1/10th of the minimum wavelength of the noise. In a system where the maximum significant noise frequency is 200 MHz, this wavelength is 150 cm, and the slits should be less than 15 cm. But such a hole will still cause some reduction of the effectiveness of the shield. A hole that does not affect the effectiveness of the shield has to be less than 1% of the minimum wavelength, in this case 1.5 cm. It may turn out that a 100%-effective shield is not required, though. The filters on the I/O and power lines are usually more important. In many applications, where high-frequency noise (>30 MHz) is dominant, it may not even be necessary to use a metal shield. A conductive layer on the inside of a plastic housing will, in some cases, be sufficient.

Multi-layer Boards

When three or more layers are used, it is essential that one plane is used as a ground plane. It is also recommended to use one layer as a power plane if four or more layers are used. These two planes should then be placed next to each other in the middle of the board, to reduce power supply impedance and loop area. It is not a good idea to place the power and ground planes as the outer layers to act as shields. It does not work as intended, as high currents are running in the ground plane. A shield layer would have to be a second pair of ground layers.

F2 Labs can also provide you with Pre-compliance testing to ensure that there are no major EMC issues with the product prior to the full EMC Evaluation. For more information please contact our EMC Technical Manager, Ken Littell, at 440-426-6002 or via email at klittell@f2labs.com.