DESIGN FOR EMI & ESD COMPLIANCE

All of we know the causes & impacts of EMI & ESD on our boards & also on our final product. In this article, we will discuss some useful design procedures that can be followed to reduce EMI, ESD related issues which in turn helps to get EMI & ESD compliances. Before going in detail first let us refresh our knowledge on EMI & ESD definitions

DEFINITIONS:

Electromagnetic Interference (EMI) is caused by undesirable radiated electromagnetic fields or conducted voltages and currents. The interference is radiated by a source emitter and is detected by a susceptible victim via a coupling path. The coupling path may involve one or more of the following coupling mechanisms:
1.Conduction - electric current
2.Radiation - electromagnetic field
3.Capacitive Coupling - electric field
4. Inductive Coupling - magnetic field

Static electricity is defined as an electrical charge caused by an imbalance of electrons on the surface of a material. This imbalance of electrons produces an electric field that can be measured and that can influence other objects at a distance. Electrostatic discharge is defined as the transfer of charge between bodies at different electrical potentials.

Static electricity is often generated through turbocharged, the separation of electric charges that occurs when two materials are brought into contact and then separated. Examples of tribocharging include walking on a rug, rubbing a plastic comb against dry hair.

Electrostatic discharge can change the electrical characteristics of a semiconductor device, degrading or even destroying it (Gate voltage of transistor is very sensitive—can be permanently damaged by high voltage).

REDUCING EMI:

The following discussion outlines the most common EMI problems and how to reduce them.

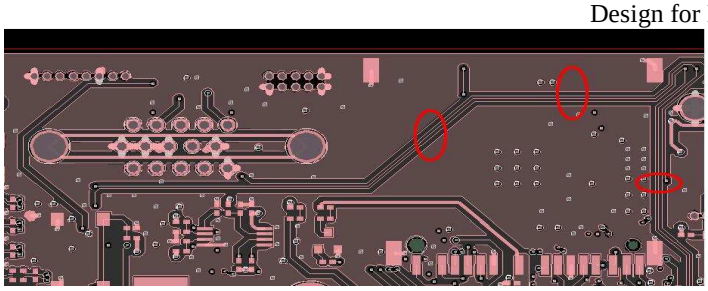
Ground Impedance: At the board level, most problems involve ground impedance. And even if ground impedance is not at the root of the problem, it will help to brought under control before the problem can be solved.

Consider some signal integrity problem, that of shared ground. It is also known as common impedance coupling. In this example, the ground path is shared by two circuits, the noise source and the receptor is sensitive low-level analog circuit. The voltage drop across the ground path looks like a signal to the analog circuit. How much noise can be tolerated is dependent on the application, but if the receptor is a sensitive analog circuit, the allowed voltage may be quite low. The mathematics of this problem is: **E** = **IR** (Ohm's law). The sensitivity of the recipient circuit will be dependent resistance & current across the input of the circuit. The solution is, either reduce the resistance to near zero or reduce the current to near zero.

To reduce the resistance to near zero, use a ground plane. At higher frequencies, the impedance of even a short trace is three orders of magnitude higher than that of a ground plane. Therefore, while the impedance of a short trace at 100 MHz may be 10 Ω or more, the impedance of a ground plane will be 10 mΩ. So, 1 A of current will bounce the signal ground trace about 10 V, which is clearly unacceptable in any digital signal case, because voltage drop across trace will acts as a antenna & radiate the EMI, but the same current will bounce the ground plane only 10 mV, a level that will almost always be acceptable.

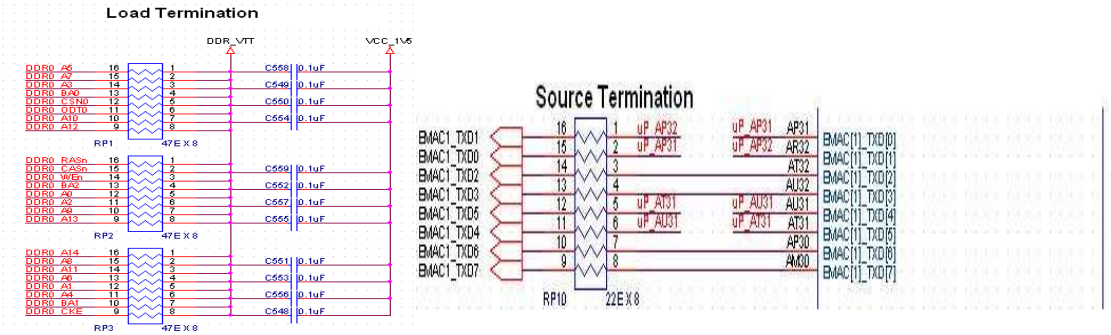
To reduce the current to near zero is not so easy. The current in the power path usually can't be reduced. Presumably, high current exists for a reason other than to just dissipate power. Rather, the alternative involves steering the currents along separate paths. This technique enables the current to avoid sharing return paths.

Example: Below figures shows one of the iWave layout snap, uses ground plane as reference for sensitive analog signals as well as digital high speed signals to reduce the resistance of the signal trace & indirectly EMI emission.



Terminate High-Speed Lines: This completes the impedance-controlled path. Impedance termination techniques include the load termination, ac termination, series termination, and several nonlinear load terminations. All of these prevent or at least minimize reflections, but the series impedance termination is the only technique that puts the termination at the source, rather than at the load. Therefore, it is the only termination that inherently restricts the high frequency from leaving the driver. Series termination (also known as source termination or back termination) has some downsides, including a slower switching time. However, if it can be used, it will be quieter than the load terminations. Although primarily an SI issue, it also results in reduced trace radiation.

Example: Below snaps shows the iWave schematics for source & load termination. All termination values are calculated & simulated for optimized EMI emission.

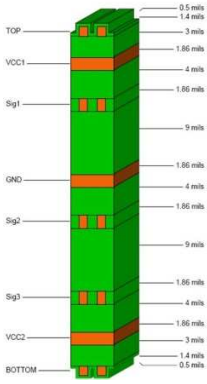


Slow the Edge Rates for Critical Signals: The general rule for both SI and EMI is to use no signal faster than needed for the desired function. Any high-frequency components not needed for the function only produce excess energy that can do nothing but interfere with neighboring signals. Excess energy means increased crosstalk to other adjacent signals and degraded signal quality. And, if the affected signal leaves the board, this excess results in unwanted energy piggybacking onto the signal lines.

The best place to slow the edge rate is to place a filter immediately at the driver, such as a series R and shunt C. Usually, the signal on the circuit board can tolerate the dc drop. Use as large a series resistance as possible. If the signal requires impedance termination, it will require series resistance of about 30 Ω. Some chips are available with built-in 25-Ω resistors, which are suitable for this problem. A shunt capacitor is usually not necessary, because the stray capacitance on the line and load completes the filter. This is similar to the series/source termination as we discussed in last section. Besides this, many of the controller's pins has the provision to adjust the slew rate through register setting. This could be made use of to provide optimum slew rate.

Control Path Impedance: Impedance control is becoming increasingly important in high-speed circuit design, especially in telecom and computer applications. It is important to remember that the signal path is only half the path. The return path is the other half. There are more ways to create a return-path discontinuity than there are in the signal path. For example, consider a signal path that passes through a via to another routing layer. The return-path impedance may be well controlled when the trace is immediately above a plane, whether a ground plane or a voltage plane. A discontinuity will appear every time reference planes are switched. The discontinuity also creates an emission problem due to long return path. A signal crossing a slot energizes the slot, launching an electromagnetic wave, and increases the voltage drop across the plane, creating a common-mode voltage. The fix is to avoid switching reference planes if at all possible and to unconditionally avoid crossing a slot with a high-speed signal line. If it is necessary to switch planes, place a decoupling capacitor alongside the via to give the shorter return path.

Example: For all high speed multilayer boards, iWave defines the pcb stack up & achieve the required impedance for the different signal paths (like analog, differential, single ended digital etc) before starting the layout. This helps to achieve the controlling the path impedance.



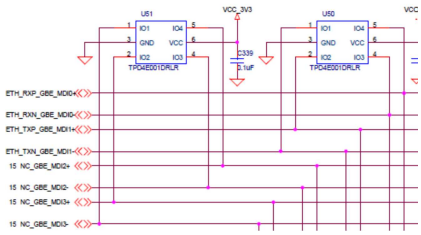
ESD PROTECTION:

The following guidelines can be implemented to protect our devices against **ESD**.

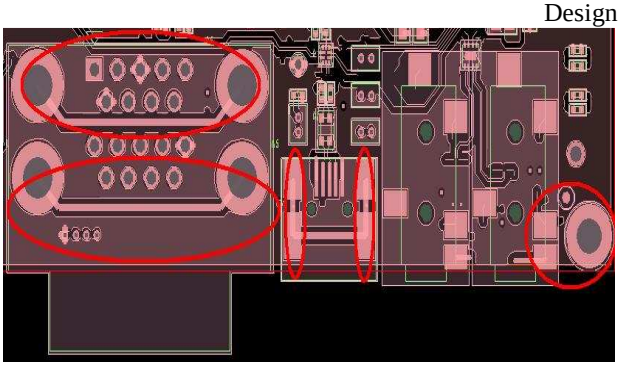
Input/Output lines to a circuit board can be subjected to extraneous voltage induced or directly connected - such as Electrostatic discharge (ESD). The circuitry itself should be designed to withstand the voltage, - special components are available to do this. It is also possible to supplement the protection with PCB spark gaps. Spark Gaps can also be fabricated as a part conductor pattern of a PCB, is show below. The contact area needs to be free of solder resist, in order to function as a sparkgap. Breakdown of small sparkgaps is approx V= (3000pd+1350) where p is pressure in atmospheres and d is distance in millimetres. This sparkgap can be expected to have a peak voltage of about 2000-2500V.

Earthing of connector grounds and such lines as sparkgaps requires special consideration. Usually the best physical arrangement for this type of earth is the Star - where the earth happens at one place, and the lines go back to that place separately. The most important issue is to carefully consider how the currents will flow to earth, especially under conditions of fault.

Example: ESD protection devices are available for all external peripheral interfaces such as ESD filter for analog video interface & high speed digital interfaces like Ethernet, USB etc. Below figure shows the iWave's schematics uses ESD filter for Ethernet interfaces.



Another example for ESD protection in layout stage is shown below. For all externally accessible connectors chassis ground (earthing) are separately routed and low resistance current path to earth is provided for high current /voltage spikes.



Shielding is one the techniques used to avoid the EMI emission & also it will block the radiation entering inside the enclosure from external noisy sources. ESD shielding components are also available, example antistatic papers. For all military grades electronic products should be shielded to avoid both EMI & ESD.

Example: iWave's Ruggedized PDA uses shielding for LCD window, and for all connector openings & also for external cables.

Finally, before implementing any hardware circuit, first thing to remember is that **NO component is perfect**, not even wire. Unexpected results happen when we forget this rule. The second most important rule is that **there is no such thing as ground**. This is especially true when either high currents or high frequencies are involved.

In this article, only few of the EMI / ESD related issues, solutions are discussed. It does not cover the full spectrum. iWave has built expertise in the board Design meeting the EMI & ESD compliance requirements. iWave has successfully implemented many of these techniques in number of boards. Among them one of the success story is iWave's Ruggedized PDA passing the military grade certification for EMI & ESD related test cases

Reference: http://www.radioing.com/eengineer/
http://www.seattlerobotics.org/encoder/feb97/powerup.html

From:

Name: Raghavendra C

Designation: Member Technical - HW

e-mail: raghavendrac@iwavesystems.com