Your guide to CE MARKING

PART 3 GETTING YOUR PRODUCT TO PASS.

by Gareth Bradley

You've got a new product and you want it to pass, preferably first time. What can you do? Firstly it is a lot easier quicker and cheaper to ensure that EMC is thought about during the design stage of the product.

PCB layout design is important, a PCB that has had no consideration to EMC can contribute to EMC problems.

Increasing the width of power supply tracks on the PCB can help reduce the common impedance values which will help reduce EMC problems.

Minimising the loop area formed by the power tracks and the ground tracks will also help. This is done by keeping supply and return tracks close together. This will reduce pick up from external magnetic fields and reduce radiation from the board itself. See fig 1 for example of good layout and bad layout.

If your product is multi layered then use of separate power supply planes and ground planes can significantly reduce the common impedance values. If your PCB is only single side or double sided then the use of power planes and even a single ground plane may be impossible. In these cases the use of low impedance buses to reduce supply line impedance. These are available from various suppliers. They consist of two metal strips separated by a thin insulating layer with PCB pins for mounting and power take off. These provide a low impedance supply and the use of a separate distribution bus for power supplies will allow the routing of other signals to be simplified.

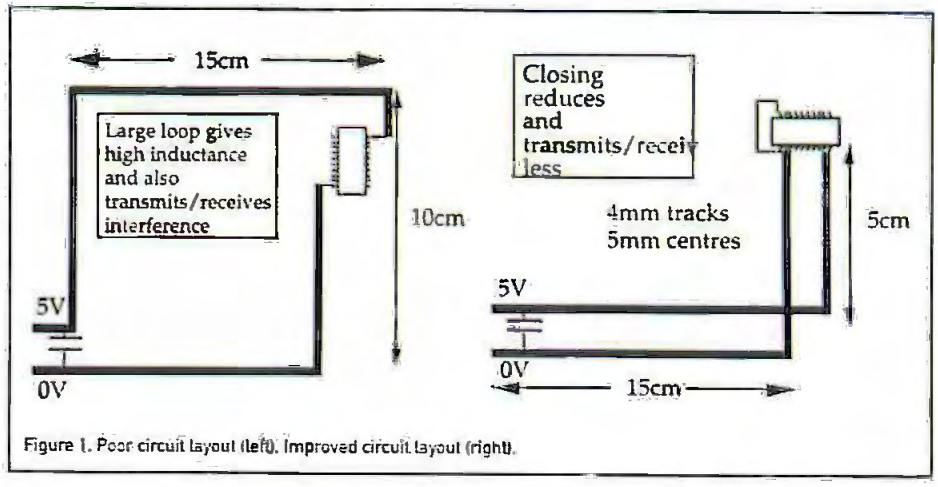


Figure 1. Poor circuit layout (left). Improved circuit layout (right).

Also having return paths for signal tracks and also minimising the loop area that can be formed by these signal tracks will help reduce EMC problems. See Figure 2 for examples.

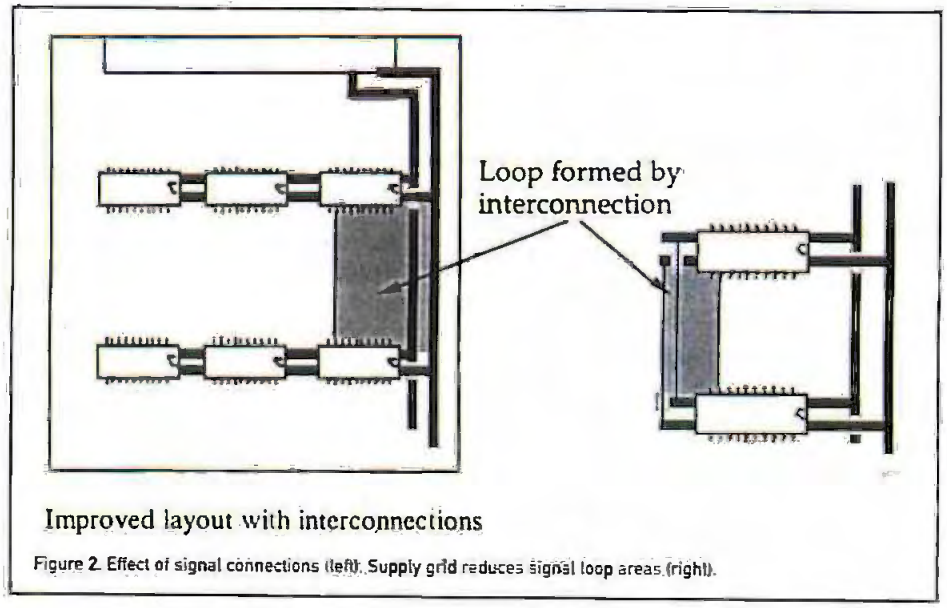


Figure 2. Effect of signal connections (left). Supply grid reduces signal loop areas (right).

The use of slower devices in the logic family can also help. The speed of the rise and fall times of devices contribute to the overall EMC performance of the product. The slower the rise and fall time of the devices the less they will contribute to the EMC performance. The functionality of the product may limit the choice on which logic family is used.

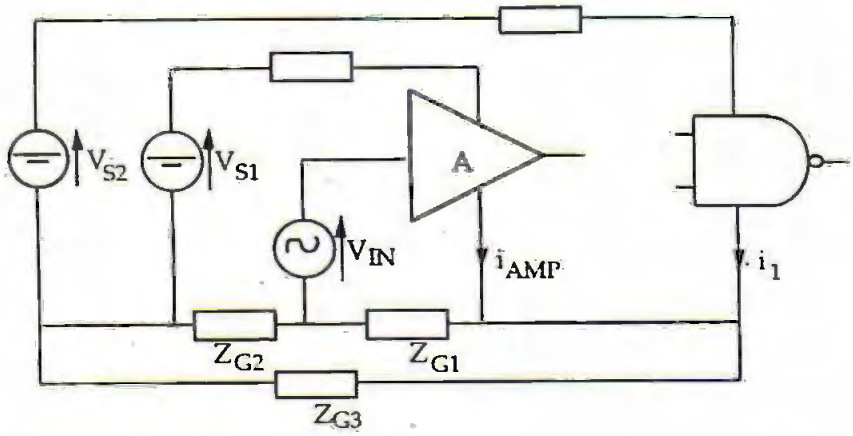
For particularly sensitive circuits which are used on the same PCB as power logic or power circuits such as signal conditioning circuits and a-d converters. The common of ground connections for each supply should be connected together at one point (if needed at all).If the common or ground connections are made at more than one point then this can cause problems. It can cause return current to flow along the small signal common track and create common impedance coupling which is not wanted. See fig 3 for examples.

The most common and basic way of helping towards reducing the EMC problems is the use of decoupling capacitors across Vcc and ground.

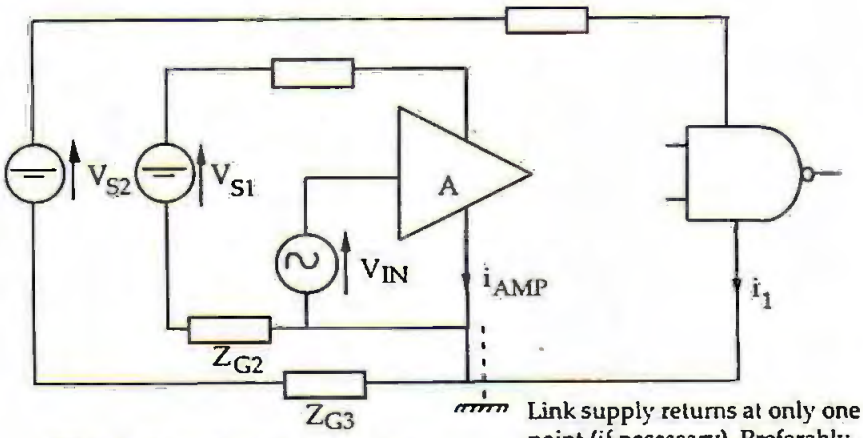
Capacitive coupling can be a big problem as far as EMC performance goes.

If you have any two conductors in proximity this will cause a capacitive coupling effect. This does not just apply to any PCB tracks in proximity but also applies to any components with metal cases that are nearby the source(heatsinks and mounting hardware etc).Any change applied to the potential of the source conductor or PCB track will cause current flow in the victim conductor which will result in a change in the potential of the victim conductor. Reduction of capacitive coupling can be achieved in several ways.

1. You can increase the distance between the source and the victim. This will reduce the actual coupling capacitance.



The wrong way to do it



This is how it should be done

Link supply returns at only one point (if necessary). Preferably at system ground point

Figure 3. Use of separate power supplies to minimise coupling between circuits.

2: Use of a ground plane.
3: The use of an earthed screen between the source and the victim to prevent the electric field produced by the source from reaching the victim. The screen must be made of conductive material and must also be properly earthed.
4: Use of guard conducters to reduce the coupling effect.

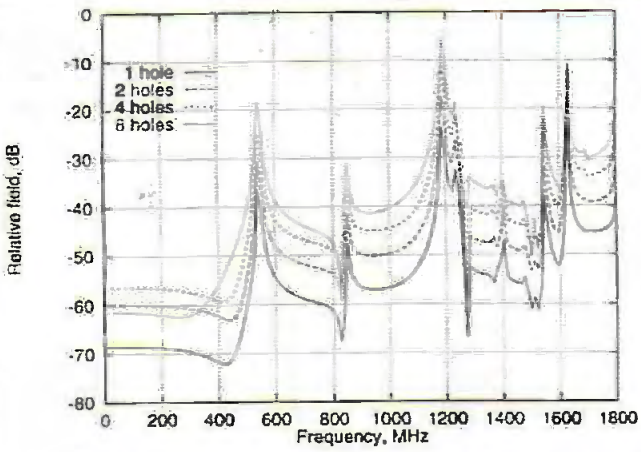


Figure 4. Effects of increasing number of holes.

When designing the PCB careful layout of the components on the PCB is a good measure against capacitive coupling. Do not run sensitive analogue circuits next to high power or switching circuits.

Inductive coupling is when a magnetic field exists around circuit carrying current, any change in the current flow produces a proportional change in the magnetic field and this change can induce currents in other nearby circuits. The most common source are ones that large are rapidly changing currents are present. Switching regulators, high speed logic circuits with fast rise and fall times, power switching circuits, and power amplifiers are all examples.

This inductive coupling on PCB is usually due to long parallel tracks or running them in closely coupled loops. This can be avoided with the following techniques.

1: Increasing the distance between the tracks to reduce mutual inductance.
2: The use of an earthed screen between the circuits.
3: Using a ground plane near the cause of the coupling. Eddy currents induced in the plane cancel out some of the magnetic field.

Any tracks that run near power tracks that cross the power track at right angles are not effected by this it is only parallel runs that are effected.

Overall emissions from the product can be reduced and even nullified by the use of metal cases.

A completely closed box will offer good protection against emissions. For every hole that is placed in the case this will reduce the effectiveness of the shielding. The shielding effectiveness will vary depending on the size and amount of the holes. See fig 4 for examples. It is better to have separate holes than combine any of the holes. The amount of fastenings and type of fastening can make a difference, the more secure they are the better the protection. Where any cables leave and enter the case, the position of these cables in relation of the Case can also make a difference to the overall EMC performance. See Figure 5.

There are companies that specialise in providing solutions to minimise EMC emissions. These products include caskets to go on any lid or any part of the case that detaches. There is also paints available to coat the inside of the case. These paints can offer quite good protection.

Useful links

1.www.spraytech.co.uk
2.www.applicoat.com
3.www.lairdtech.com

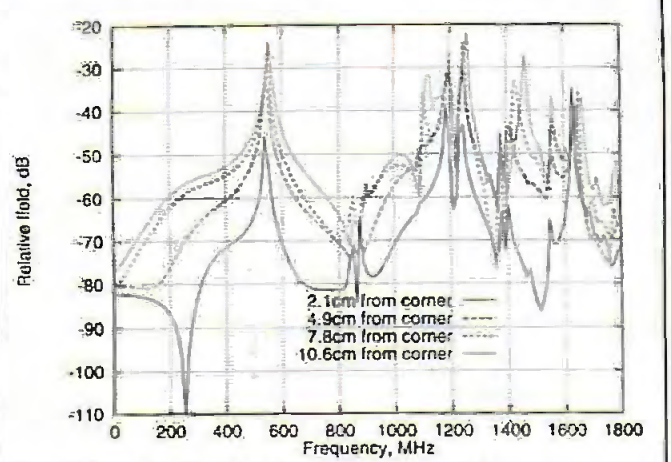


Figure 5. Comparison of varying entry point for cable penetration.