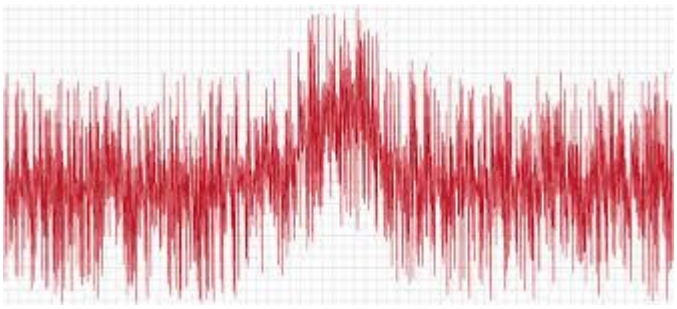
Noise

1. ## Introduction:

With all the different circuits in operation on a PCB, there are many opportunities for undesirable noise to be created. Let's look at some of the more common noise problems that can occur.

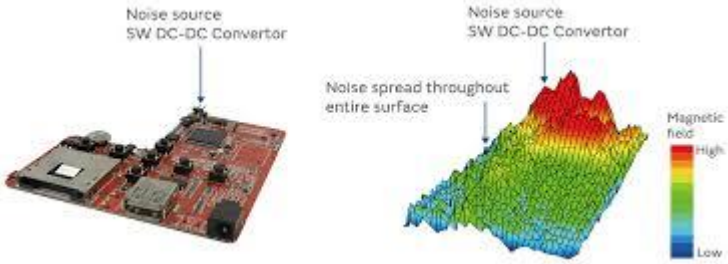
Ground bounce: As the switching speeds of high-speed digital circuits continue to increase, there is less time for the signals to return back to their reference ground level. This can cause the signal to "bounce" above that ground level resulting in higher than expected amounts of current and create noise in the output signal. With a lot of switching happening at the same time, the amount of noise from the ground bounce can create false or double switching. This effect can cause failures in the normal operations of the circuits.



Crosstalk: With multiple circuits operating in a PCB, the more active circuits can influence the less active circuits if their traces are running side by side. This noisy unintentional electromagnetic coupling between the two is known as crosstalk. It can happen with traces that are too close to each other horizontally (side by side) or vertically on another layer.

EMI: Electromagnetic interference can come from many sources, with one of the chief problems being poorly designed return signal paths. Ideally, the signal layers of PCB will be sandwiched between power and ground planes allowing for a clear return path on the ground plane. If the plane is blocked by holes or split planes then the signal

return will wander around trying to find its way back creating noisy interference as it goes.



Power supply: To work effectively, the onboard power supplies need to be laid out with the shortest paths possible for the current. Without care in how the parts are placed, the power supply could create a ringing signal that will add to the noise level of the board.

How to avoid Noise:

1. Filtering

Filtering circuits and components are best used when you only want a specific frequency or range of frequencies to reach the load end of an interconnect. Flip this around; filters can also be used to remove one specific problem frequency from an interconnect (e.g., a notch filter). Filtering is best left to the analog or RF designer; you should not use filter circuits to try and remvoe noise from an interconnect carrying digital signals. For RF and analog systems, bandpass filtering is typically used to pass signals into a receiver and prevent noise.

Power systems will also make use of filters, primarily on the output stage of a regulator or the input stage where the device is connected to the main (unregulated) power source. In this usage, the filter is applied to try and remove conducted common-mode currents. Note that switching regulator topologies have a built-in filter from the inductance. If you have too much noise on the output from your regulator stage, you might need to increase the regulator's inductance (assuming you've already laid out the board correctly).

2. Isolation

This term is a bit ambiguous and is probably overused when we talk about noise. It often comes up in the context of mixed-signal PCB layouts. From what I can tell, this refers to two possible courses of action:

1. Splitting the system so that it sits on top of two different ground planes
2. Separating nets from each other so that there is less crosstalk

For mixed-signal systems, if you follow some basic layout principles surrounding placement and routing, you won't need to do something undesirable like splitting a ground plane. Separating nets is always a good way to prevent crosstalk or interference that might appear as noise on the system IOs.

3. Shielding

This should be a last resort to try and remove noise. There are many components in a PCB layout that can create a noise problem, and so it is tempting to just target those components with shielding and ignore everything else. Before using shielding, check the list of recommendations below first, then consider playing with some shielding options.

In the case you do find that you need shielding to overcome your noise problem and pass EMC testing, there are some commercially available shielding options:

- Custom shielding cans that will cover components and connect to ground
- Shielding gaskets, compounds, and meshes
- Ferrite plates that can be placed in an enclosure

The simplest form of shielding probably already exists in your PCB: your ground plane. Simply reworking the stackup to put the ground plane closer to your components, or adding a ground plane when there was none, will provide additional shielding.