

HOW TO REDUCE NOISE IN PCB

STRATEGIES TO PREVENT NOISE IN PCB:

1.Filtering

When to use filters : Filters should only be used when we are dealing with analog signals not digital signals ,and filters can be used with low frequency signal and also high frequency signals, but this will require some special filtering techniques

How filters work : filters simply are designed to allow a specific frequency to pass through it and reach it's destinations and other frequencies will get heavily attenuated and doesn't reach the receiver by this way only desired signals will pass through, and any added noise is going to be filtered and usually the filter is placed somewhere near the receiver side so the signal is filtered right before it reaches the receiver.

Common filter types :

- **Band stop filter (notch) :** this type of filters is used to remove specific frequencies will allowing the rest of frequencies to pass
- **Band pass filter :** this type of filters is used to allow only specific frequency to pass, and other frequencies will be removed
- **Low-pass filter :** this type of filters is used to pass the high frequencies while removing low frequencies
- **High-pass filter :** this type of filters is used to pass the low frequencies while removing high frequencies

- **EMI filter** : this type of filters is used to block high-frequency electromagnetic noise using a combination of inductors and capacitors

Filter should be specifically selected for the type of noise to eliminate also not affecting other signals carrying the noise itself. And commonly in pcbs we shouldn't be using any type of filters as filters can attenuate the digital signal so much so it can't even reach the receiver ,instead other techniques are more common to be used like shielding and isolations.

2.shelding

Shielding is basically trying to make a cage for the circuit to eliminate any source of outside noise to penetrate into the pcb very common type of shielding is (FARADAY CAGE) this technique uses a metal box to shield the circuit and connecting this box to the ground through vias to prevent noise

Another type of shielding is coplanar (copper wave guide) is technique is done by placing the trace carrying signal between two copper pours and connect these copper pours to GND through vias so any outside noise will not reach the signal track and dissipate in the GND pour

3.isolation

Used in mixed signal designs where the design contains digital and analog signals. The general idea of isolation is preventing the return path of signals to interfere with return path of another signal which creates noise. This is usually done by trying to avoid return paths from crossing each other in pcb layout and this can be achieved by changing the layout and moving components with signal return path slightly away

from each other , splitting ground planes is not required to be done here for most of the designs unless we are working with highly precise DC measurements or audio systems in these cases splitting ground planes is required to eliminate noise effectively

WHAT TYPES OF NOISE IN PCB:

Cross talk: occurs when a signal in a trace induces a signal on other trace and that usually happen when the traces is too close to each other

Signal-to-signal noise : occurs when one signal interfere with another

Power-to-signal noise : occurs when fluctuations in power supply affect the signal

External radiated emissions : occurs when radiated emissions are electromagnetic waves generated by the PCB that interfere with other nearby electronics

Thermal effects: basically noise caused by heat generated by components have high effect in high-power circuits.

HOW TO PREVENT NOISE WHEN ROUTING ABOVE POWER PLANE:

When routing a signal trace above a power plane this can cause EMI problems in the signal this problem doesn't happened when routing over GND plane as the GND plane with will have a mutual capacitance between the signal and this provide a return path for current to pass through but in power planes it is different as there is no mutual capacitance between the power plane and the signal so capacitors

should be manually placed to provide this mutual capacitance and return path for the current .

To connect the capacitor properly one terminal should be connected to GND, and the other end should be connected to PWR and the capacitors should be placed one at the beginning of signal trace and one at the end .

Placing these capacitors this way will provide a decoupling / bypass for the signal and the current will have a low impedance route from the power plane to GND this way will eliminate the noise.