How to reduce noise in your PCB design

1. Filtering

Pairing up the right filter that is going to be used with the specific noise that needs to be eliminated is a critical stage as also to not affect the signal that is carrying the noise.

- Usage

Filters are used with analog signals unless they have very high roll off, they can be used with digital signals

They usually target low frequency analog except the specialized filters which are used for RF signals

- Types of filters

1.1 Notch filter

Removes specific frequencies

1.2 Band pass filter

Allows a band of the desired frequencies and removing everything else

1.3 Low pass filter

Attenuates a specific band of frequencies in a signal's frequency spectrum

1.4 High pass filter

Allows high-frequency signals to pass but blocks slowly changing signals.

2. Adding shielding

By putting a bunch of metal around the problem circuit and ground it, and that's going to eliminate the noise

- The idea behind shielding

By putting some vias around the components that have noise, and finding a big metal shielding can that basically encloses this entire component.

So, the idea is that if this component is emitting a lot of noise, the shield is then going to block this, and then by attaching this to some pads, that will be connected to ground, usually on the next layer, and that enhances a totally enclosed cage that prevents any noise from escaping this component and possibly reaching other circuits.

Vice versa, it also prevents any noise from entering inside the cage up to very high frequencies.

 people will go to various lengths to try and create a similar structure around different elements in a PCB.

3. Advanced noise reduction options

By reducing the coupling strength between the noisy circuit and another circuit or component, although spacing things out is always going to reduce noise

By bringing the ground closer to the trace that's being affected, and that will also reduce these parasitic capacitance and parasitic conductance values.

4. Isolation

 By making sure that different frequencies or digital and analog signals don't interfere with each other, when are on the same board

By practicing good layout practices, and learning how to track return paths, the PCB will be in a good shape as a lot of noise problems will be prevented.