How to reduce noise in PCB design

By: Yahia Ahmed Saber

Noise in PCBs can come in different forms like crosstalks which could be signal to signal noise and can come in form of electromagnetic coupling which could be power to signal noise. Moreover, noise can be caused thermally. There are various ways to try to reduce noise such as:

1. Filtering

This method is most popular in low frequency analog circuits. Notch filters can be used to remove specific frequencies. Bandpass filters are used as well to allow the frequencies you want and remove all other frequencies and conversely the bandstop filters operate oppositely and are beneficial too.

2. Shielding

A big metal shielding can is used to enclose the component that is emitting noise or receiving noise from other components. This enclosure creates a cage with it being connected to the ground. There are various shielding cans used in this method as well as various shielding materials that can be used inside the enclosure to absorb the radiation.

3. Isolation and splitting up ground plane

Isolation prevent the return paths from the signals in the circuit from interfering with the return paths of other circuits. Tracing the return paths of each circuit and changing the layout of the PCB is essential for reducing noise. Nevertheless, there are some applications where doing this won’t help so splitting up the ground plane may be useful if necessary.

4. Other noise reduction options

Spacing components out more is a guaranteed method to reduce noise. In addition, decreasing distance between trace and ground plane reduces parasitic capacitance and inductance values which helps in noise reduction. Placing stitching vias around the circuit properly can supress resonances and prevent noise by creating resonant cavities.