Fixing Electromagnetic Interference on Your PCB by Samuel Garcia

Article published on July 12th 2012 | Technology

As printed circuit boards continue to evolve and become more complex, there is a need to come up with ways to prevent the electromagnetic interference that can commonly become a concern with these items.

One way of doing this is to decouple the capacitors with the ground plane and reduction the number of emissions that cause this electromagnetic interference. For both digital and analytical items, you will need to reduce the number of emissions that are radiated and ensuring that signal nets have minimal levels of actual interference.

More passive devices will use a network analysis solution with their modeling techniques to help produce solutions that remain effective. The most common types of devices that can be found as part of this solution for electromagnetic interference will be those that are frequency independent, solutions that are equation based or algorithmic and those who handle this concern though measured data levels.

When there is a noticed level of concern with a signal, you will need to determine how much of the energy is actually being reduced as a result of the radiation that may be appearing in the fields. A solution for this will be to add in decoupling capacitors around the Vinput along with the ground planes and this can quickly improve the signal. While this solution can be important and temporary, you will want to ensure a full wave solution is done on the unit to determine what the root cause is and to come up with an accurate answer that will include the spatial dependence of the unit. This can help you come up with a more permanent solution for the signal concerns you have.

Looking at the actual power source on the PCB can be useful in this process as well. Much of the loss that stems on these boards can stem from the power supply that they use. It will be important that you spend the time to determine what alternatives are in place for these items and determine if the dc is resulting in the loss that you are experiencing.

Since the electromagnetic interference can also stem from the crystal oscillation unit. It is because of this that many PCB manufacturers are starting to produce built in solutions to offset some of these concerns and to help the hobbyist avoid a majority of the concerns that can come up with the EMI.

It is important that you take the time to understand the regular concerns with EMI on any PCB you are considering. With some research, you can determine which choices are going to be prone to more problems. Then take advantage of the simple solutions that are in place for helping to reduce these interferences.

Article Source:
http://www.articleside.com/technology-articles/fixing-electromagnetic-interference-on-your-pcb.htm - Article Side

Samuel Garcia - About Author:
a EMSCANâ€“ A professional Toronto based company that provides services for both Printed Circuit Board and Antenna Engineers. Complexity of printed circuit boards can be reduced with the application of a Electromagnetic Interference in effective manner.

Article Keywords:
Electromagnetic Interference, PCB Engineers, EMC problems, EMxpert, emi shielding, printed circuit board, spurious signals, radiating components, near-field emissions

You can find more free articles on Article Side. Sign up today and share your knowledge to the community! It is completely FREE!