Electronic Design

STEVE TARANOVICH, Contributing Editor

Popular Methods to Mitigate Electromagnetic Interference

From shielding to optimized vias, engineers can employ a number of commonly used approaches to reduce EMI in their designs.

This article presents methods that will help lower or even eliminate pesky EMI, leading to robust electronic designs.

Defining EMC

Electromagnetic compatibility (EMC) is defined as the capability of electrical equipment and systems to function competently in an electromagnetic environment.

In a system that requires EMC, components will act as electromagnetic sources that are designed to reduce their interference. And components that are typically susceptible to interference will be hardened to diminish that problem.

When end-equipment manufacturers integrate components from different suppliers, the best way to ensure that interferers and susceptible circuits will be able to harmoniously coexist is to form a common set of rules, whereby interference will be limited to a particular level and susceptible circuits can fully handle that level of interference.

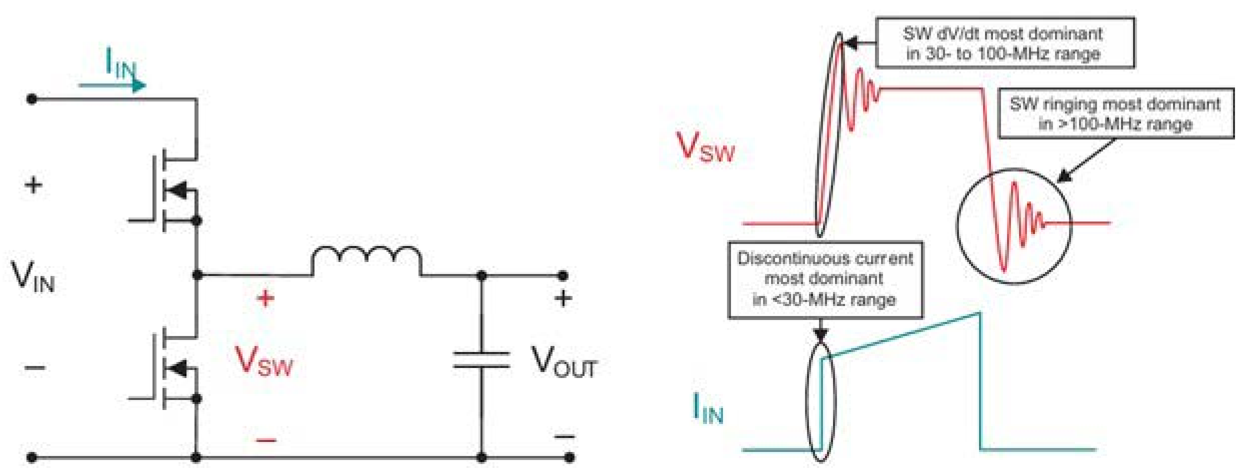
EMI Reduction Methods

A number of strategies can be employed to mitigate EMI, including shielding, grounding, filtering, component selection, and even software adjustments.

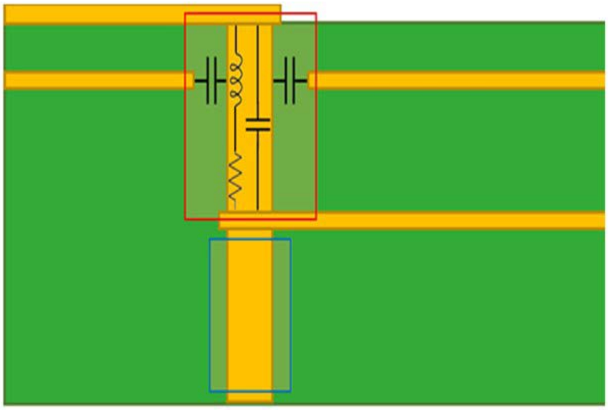
Specifically, metallic shielding enclosures could be added around circuits that generate EMI. In certain cases, this may be more easily accomplished by enclosing an entire circuit board. Shielding can also be applied to cables and connectors depending on the source of the EMI.

Most times, grounding and ground planes on printed circuit boards (PCBs) are effective in reducing EMI. Of course, proper design techniques need to be employed to avoid ground loops. Multilayer PCBs can utilize ground stitching, which ties together ground planes on different layers using vias.

Low-pass filters, decoupling capacitors, inductors, and ferrite beads are commonly used to attenuate unwanted high-



1. Here's an example of EMI sources within a switch-mode power supply (SMPS).7 (Texas Instruments)



2. Shown is a via stub resonance. (Intel)

frequency signals. Including these types of passives offers a way to eliminate the need for more expensive shielding.

Component selection and software adjustments may come into play for some applications. The former often comes down to choosing between a linear or switch-mode power supply (SMPS). The latter assumes software is actively involved in power control aspects of a design, such as motor-control systems or software-defined radio.

EMI: The Price of Using Switch-Mode Power Supplies

An SMPS can radically improve efficiency over and above linear regulators in most applications—but it comes at a price. The switching of power metal-oxide semiconductor field-effect transistors (MOSFETs) is typically a significant source of EMI that can negatively disturb reliability. This EMI generates from input currents that are discontinuous, fast rising slew rates on switching nodes, as well as increased ringing along switching edges due to parasitic inductances in the power loop.

Figure 1 depicts how each of these elements reveal themselves in different frequency bands, using a buck converter topology as an example.

Addressing PCB EMI

Various types of vias are available, such as the plated-through-hole (PTH) via. The PTH via is low cost and easily fabricated. However, it has a minor disadvantage in the form of "stubs" (these are parts of the via that aren't in the parts of the expected signal path). Stub resonance in a multilayer PCB design can be optimized to reduce loss.

Such stubs may poorly affect high-speed signal character, especially when the problem leads to unmistakably growing data rates along with the length of stubs in a thicker PCB. Stubs will perform like a resonant circuit that stores the highest energy at a particular resonance frequency. In the case where the signal has a powerful component at or close to that resonant frequency, it may culminate in significant signal degradation *(Fig. 2)*.

The resonance frequency of such a via stub has a pronounced effect on the insertion loss of the signal and thus leads to the EMI problems.

The solution to these issues is to adjust the stub length so that it influences the frequency points where the EMI occurs. By optimizing the stub length to fit within their desired frequency range, designers will be able to effectively control the offending EMI.