

Simplify low EMI design with power modules



Michael Daimer
Analog Field Application Engineer
Texas Instruments

When designing a switching power supply, you may have heard of electromagnetic interference (EMI).

More and more applications must pass EMI standards in order for their manufacturers to receive approval for commercial resale. A switching power supply implies that there are electrical switchers inside the device, through which EMI radiates.

In this paper, I will explain the sources of EMI in a switching power supply and methods or technologies for mitigating EMI. I will also show you how power modules (controller, high side and low side FET and inductor in one package) help reduce EMI.

Sources of EMI in switching power supplies

First of all, you can't beat physics. According to Maxwell's equations, an alternating current produces an electromagnetic field. This occurs in every electrical conductor, which has by nature some electrical capacitance and inductance that forms an oscillating circuit. This oscillating circuit radiates electromagnetic energy into space with a specific frequency ($f=1/(2*\pi*\text{sqrt}(LC))$). The circuit acts as a transmitter of electromagnetic energy, but can also receive electromagnetic energy and act as a receiver. Antennas are designed in such a way as to maximize transmitted or received energy.

But not every application should act like an antenna, and negative side effects can occur.

For example, switching-buck power supplies are designed to convert a higher electrical voltage into a lower voltage, but they also act as an (unwanted) transmitter for electromagnetic waves and can disturb other applications, such as interfering with the AM band. This effect is called EMI.

To maintain functionality, it is important to minimize sources of EMI. The Comité International Spécial des Perturbations Radioélectriques (CISPR) defines standards such as CISPR 25, which is a benchmark

for automotive electrical applications, and CISPR 22, for information technology equipment.

How can you lower the radiated EMI of a power design? One method is to completely shield the switching power supply with metal. But in most applications, this is not an option due to cost and space. A better approach is to reduce and optimize the sources of EMI. Much literature already deals with this topic in detail; I recommend two at the end of this article.

Let's review the main sources of EMI in switching power supplies and why power modules can help easily reduce EMI.

Minimizing current loops in the layout

As the name implies, switching power supplies are switching. They are switching the input voltage on and off with frequencies from hundreds of kilohertz up to several megahertz. This causes fast current transitions (di/dt) and fast voltage transitions (dV/dt). Alternating currents and voltages produce alternating electromagnetic fields according to Maxwell's equations. The fields spread radially from their origin and their magnitude lowers with distance.

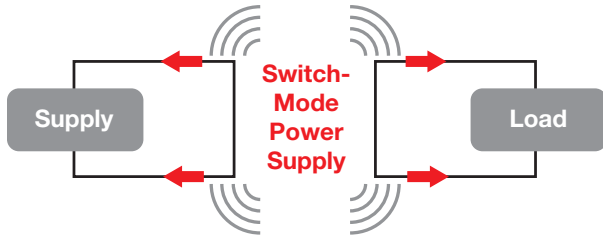


Figure 1. EMI coming from the switch-mode power supply has effects on the load and the primary power supply.

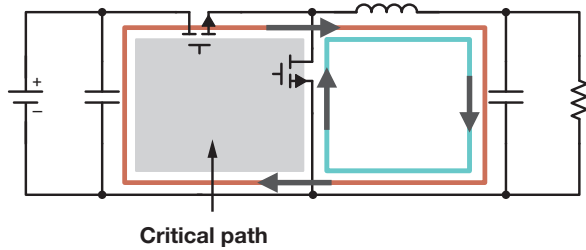


Figure 2. A critical current loop forms between the input, switches and input capacitor.

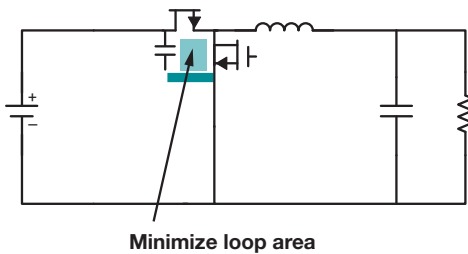


Figure 3. Minimizing the loop area helps mitigate EMI

Magnetic and electric fields interfere with the conducting parts of an application (for example, copper traces on a printed circuit board [PCB] that act like antennas) and cause additional noise on the lines, which again cause EMI (see **Figure 1**). The fact that several watts of power are translated

increases the range of radiated EMI. Radiated electromagnetic energy is directly proportional to the magnitude of current (I) and the loop area (A) through which it flows. Minimizing the area of alternating current and voltage loops helps reduce EMI (see **Figures 2 and 3**).

A look at the pinout (see **Figure 4**) helps you see the possibility of creating a good layout by reducing the area of high dI/dt loops. The switch node, for example, is a source of both: high current variation (dI) and high voltage transition (dV). A good pinout takes care of separating noise sensitive pins and noisy pins. Switch node and boot pin should be positioned as far as possible from the noise sensitive feedback pin. Additionally input pins and ground pins should be neighboring. This eases routing on the PCB and the placement of the input caps.

Figure 5 shows the modified evaluation module (EVM) of the [LMR23630](#) SIMPLE SWITCHER® converter. The two input capacitors are about 2.5cm away from the input pin. This was done to simulate a bad layout, since the current loop area (red rectangular shape in **Figure 5**) is made bigger than required and suggested by the datasheet. The oval red shape in **Figure 5** shows the switch node between converter and inductor. The loop area between both IC and inductor is as small as possible.

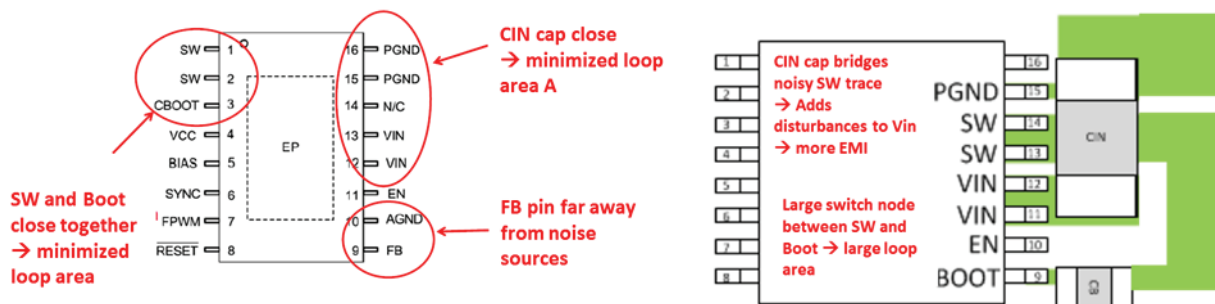


Figure 4. The pinout can help minimize loop areas. Left: Optimized pinout; Right: Non-optimized layout which makes a good layout almost impossible.

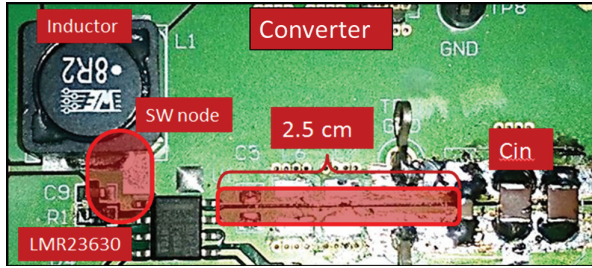


Figure 5. Example of a bad layout with a large loop area (red rectangular shape) between input pin and input capacitors. The second loop area (oval red shape) is formed between IC and inductor.

The graph in **Figure 6** shows the radiated EMI of the LMR23630 converter where only the loop area formed between V_{IN} , GND and the input capacitor is different. A good layout places the capacitors as close as possible to input and ground pins (the loop area is as small as possible). A bad layout places the input capacitors 2.5cm away from the input pin, forming a large loop area.

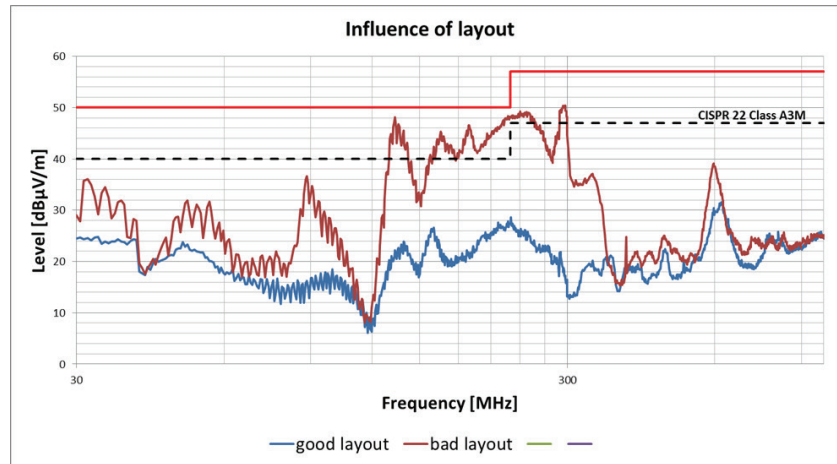


Figure 6. Influence of the input capacitance placement of the LMR23630 converter on radiated EMI.

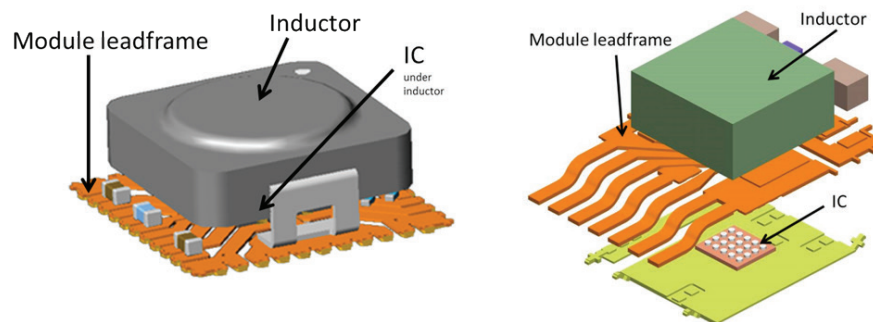


Figure 7. Internal composition of different types of power modules. In both cases, the inductor sits on top of the IC die.

The red line of the graph in **Figure 6** shows the radiated EMI for the bad layout. The blue line shows the radiated EMI of a good layout using the same EVM. The effect of modifying one loop area is tremendous. The radiated EMI level of the converter LMR23630 can be lowered by more than 20 dB μ V/m.

Therefore the placement of the input capacitors should be one of the first considerations, when designing either with a buck converter or a buck power module. Power modules also have the advantage that the critical loop area between inductor and IC is already optimized. The inductor is connected internally inside the package with the integrated circuit (see **Figure 7**). This placement creates a very small loop area inside the package. Therefore it is not necessary to route the noisy switch node on the printed circuit board.

Most inductors used in power modules are additionally shielded to prevent electromagnetic radiation coming from the coil. The high current and voltage transitions occur very close to the inductor and a part of the electromagnetic field from the switch node is shielded, with the inductor sitting on top of the lead frame (see **Figure 7**).

Fast voltage and current transients

Fast transients can cause ringing on the switch node, which will cause EMI. In some cases, a converter provides access to the boot pin. Placing a resistor in series with the boot capacitor will increase the rise time (dt), which can lower EMI at the expense of efficiency.

Figure 8 shows an EMI radiation scan of the [LMR23630 EVM](#). The layout was modified to place the input capacitors about 2.5cm away from the pins in order to simulate a bad layout and to show how placing a boot capacitor will influence EMI performance. It might be easier to place an additional boot capacitor in the design rather than change the layout completely. I recommend always planning for a boot capacitor in your design in case you need it. If not, you can short the space on the PCB with an 0Ω resistor.

Placing a boot resistor in series with the boot capacitor results in a lower EMI spectrum. The emission in some frequency regions drops as much as 6dB. **Figure 8** also shows the trade-off

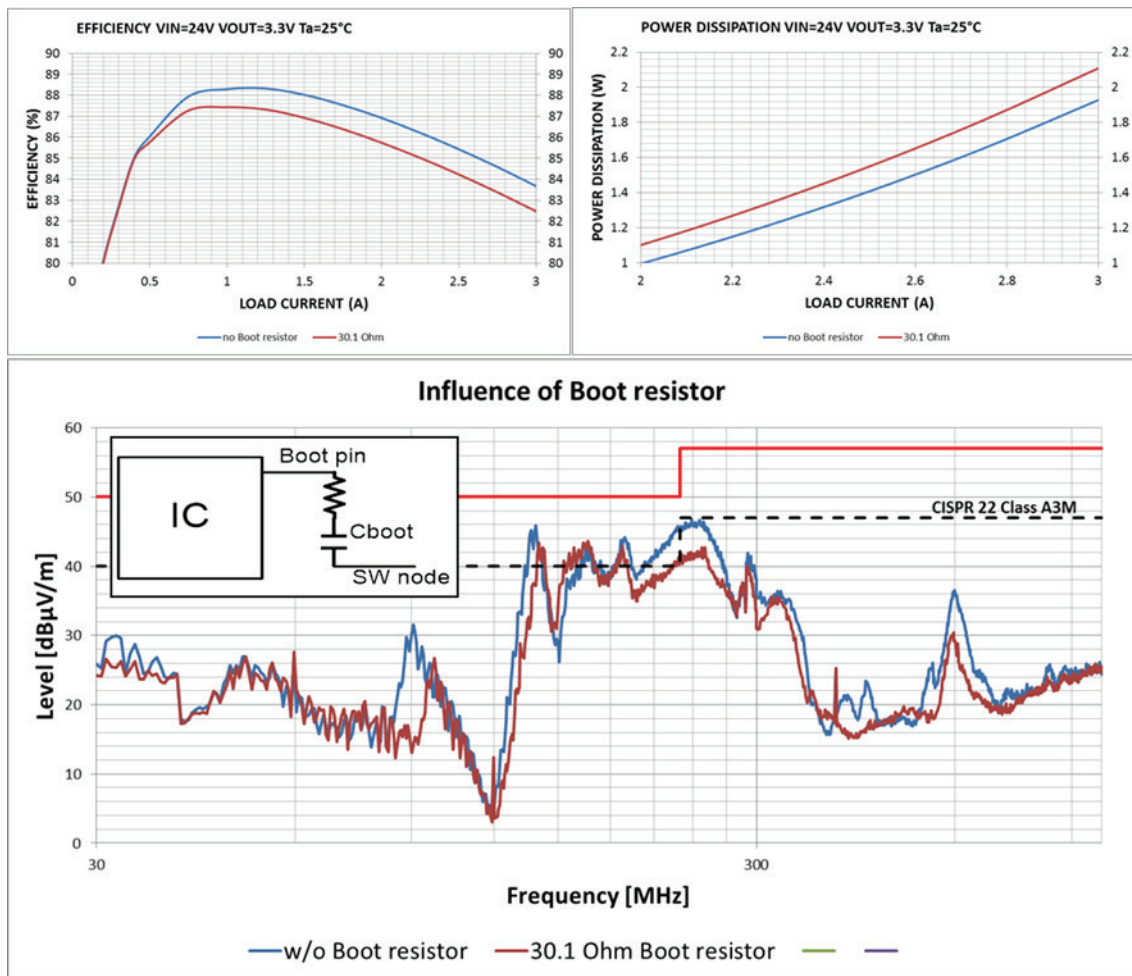


Figure 8. The influence of adding a boot resistor to the switch node of the converter LMR23630. The radiated EMI is lower, but the efficiency decreases due to higher switching losses.

in efficiency. Slowing down the rise time dt with a 30.1Ω resistor lowers the efficiency more than 1%. A look at the power loss illustrates this even more. The power loss increases for a full load (3A) from 1.9W up to 2.1W. This increase of more than 10% can be problematic and lead to thermal issues.

It's possible to lower the switch-node current ringing dI in synchronous converters by placing a small Schottky diode between switch-node pin and a ground pin to lower reverse-recovery current, but at the expense of a higher bill-of-materials (BOM) cost. Or you could add a snubber network containing an additional large package capacitance and resistance between switch node and ground. The snubber burns the energy of the switch-node ringing, but requires knowledge of the ringing frequency and proper calculations of the additional components. It also lowers the efficiency of the switching power supply.

Parasitic inductances and capacitances in current paths

For a synchronous buck converter, each IC architecture will contribute different amounts of noise appearing as radiated EMI. But it is difficult to find this out from a datasheet. Most datasheets

do not provide an EMI plot, since PCB layout, BOM components and other factors have an impact on EMI behavior. Sometimes, if you're lucky, the EVM user's guide provides a plot of the EMI behavior of this specific design. But if your design does not match the layout and BOM of the EVM, the EMI characteristic of your application may differ dramatically. Power modules simplify the layout and make fast-and-easy designs possible, since you only have to consider a few rules of thumb. For example, keep traces or cuttings in the ground plane to a minimum; in case they are necessary, design them in parallel to the current directions (Figure 9).

Protect noise-sensitive nodes from noisy nodes

Keep noise-sensitive nodes as short as possible and away from noisy nodes. For instance, a long trace from the resistor divider network to the feedback (FB) pin can act as an antenna and catch noise coming from radiated electromagnetic disturbances (Figure 10). This noise will be introduced into the FB pin, causing additional noise at the output and even making the device unstable. Taking this all into account is a challenge when designing the layout of a switching-buck regulator.

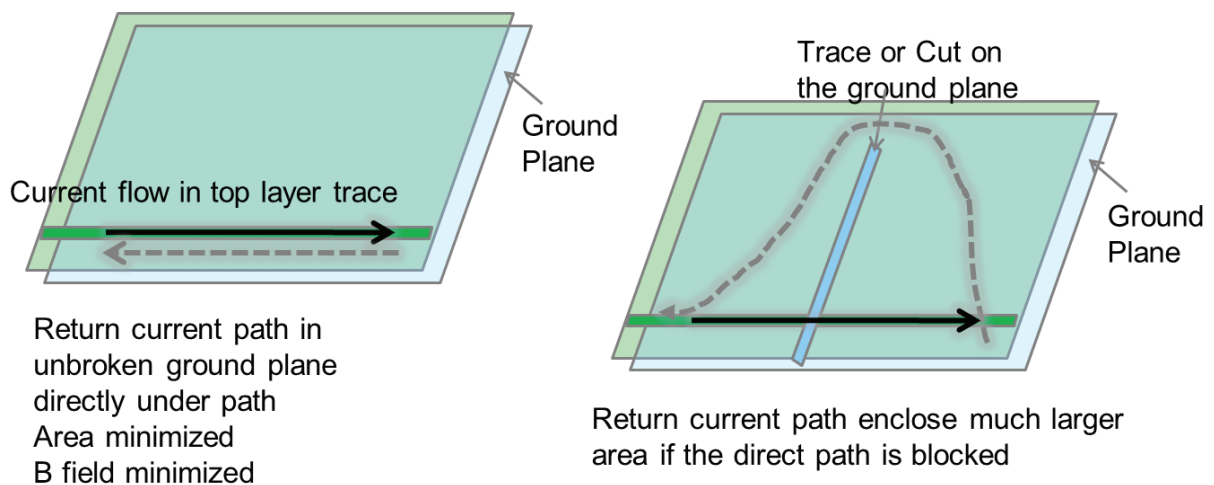


Figure 9. Cuts and traces in the PCB influence current flow, and therefore also influence radiated EMI.

| Noise-sensitive nodes | Noisy nodes |
|-----------------------|------------------|
| Feedback pin | Switch node |
| Frequency setting | Inductor |
| Compensation network | High di/dt caps |
| Sensing paths etc. | FETs, diodes etc |

Table 1. Examples of noise sensitive and noisy nodes in a buck converter.

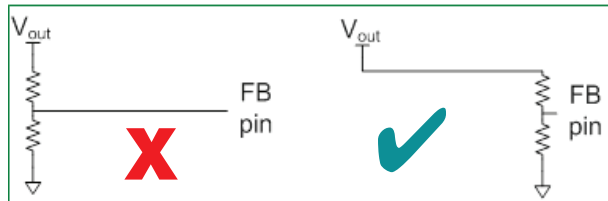


Figure 10. Always place the resistor divider on the FB pin as close as possible to the FB pin.

Modules have an advantage in that they keep both noise-sensitive and noisy nodes to a minimum, therefore minimizing the chance of choosing the wrong layout. The only thing is to keep the traces of the FB pin short.

Conclusion

There are many knobs to tune EMI in a switching buck converter, but following best practices may not be good enough. Finding the best configuration consumes a lot of precious design time. Power modules already include both FETs and the inductor, making it simple and fast to create and finish a power design with good EMI performance. The most critical point when designing with a buck module is the placement of a few external components, which can help improve EMI performance considerably.

EMI comparison of a converter and a power module

Previously, I described the sources of EMI in switching power supplies and how to reduce them. Now, I'll demonstrate how modules help

mitigate radiated EMI by comparing measurements between a converter and a power module that use the same integrated circuit (IC). Both from the SIMPLE SWITCHER product line at TI, the converter is the LMR23630 and the power module is the [LMZM33603](#), which uses the LMR23630 IC. I partially modified the EVMs of both devices to get the same BOM count so that the results only depend on the selected part (converter or power module) and the layout. Both EVMs have a good, optimized layout. Later on, I made the layout worse by placing capacitors farther away from the input pins.

Performance of the LMR23630 converter

Figure 11 shows four different EMI spectra for different design layouts. The design worsens stepwise (similar to **Figure 5**, only done stepwise). The first measurement (good layout/blue line) is with the unmodified layout of the EVM (a good layout with all input capacitors very close to the input pin). For the second measurement (small cap near/red line), the two 4.7 μ F capacitors are placed 2.5cm away from the input pin. The small 0.22 μ F capacitor stays very close to the input pin. In the third (small cap far/green line) and fourth (no small cap/purple line) measurements, respectively, the small capacitor is 2.5cm away from the input pin and then completely removed.

You can see in **Figure 11** that the placement of the input capacitors is very critical. Placing the small input capacitor far away from the input pin or removing it completely violates the CISPR 22 Class A3M. Placing the small capacitor near the input pin minimizes the loop area for high frequencies. The small capacitor filters high frequencies, whereas larger-capacitance capacitors filter lower-frequency noise.

Power modules normally include a small input capacitor in their package. Let's look at the performance of a power module when I mess up the layout.

LMZM33603 power module performance

Figure 12, which shows the EVM layout of the power module, also worsens stepwise. The blue line

shows the radiated EMI for the unmodified EVM. The red and green line show the bad layout, one with two 4.7µF input capacitors underneath on the bottom side of the PCB (red line). The green line has the capacitors approximately 3.5cm away from the input pins (highlighted by the red oval shape in Figure 13). The red bulky line in Figure 13 additionally shows the modified EVM and the critical loop area formed between V_{IN} , input caps and ground. The EMI performance gets worse, but doesn't violate CISPR 22 Class A3M criteria.

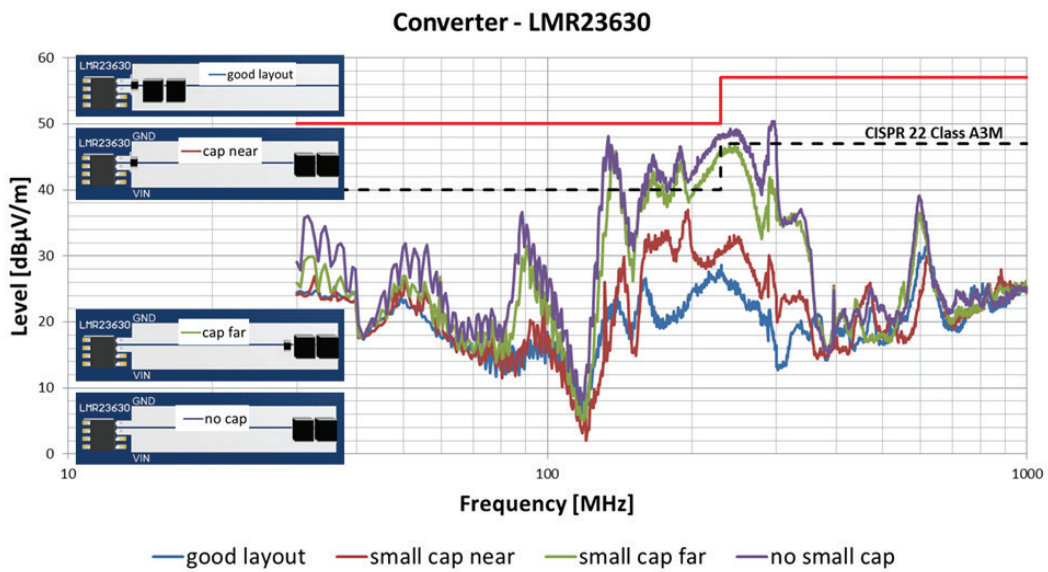


Figure 11. Radiated EMI of the converter LMR23630 with different input capacitance placements.

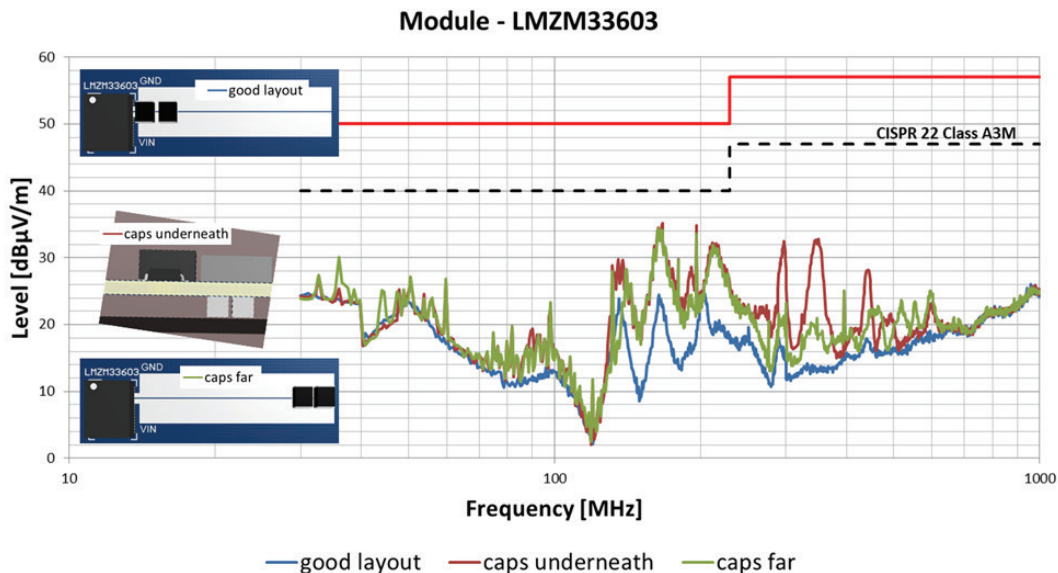


Figure 12. Radiated EMI performance of the TI's LMZM33603 power module

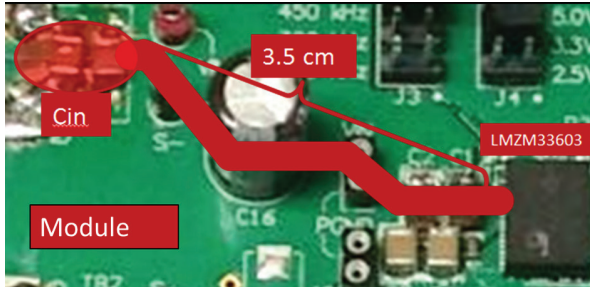


Figure 13. A bad layout example for the TI's LMZM33603 power module.

Power modules forgive layout design errors

Figure 14 compares the converter LMR23630 (red line) and the power module LMZM33603 (blue line) in a single graph. Both have a bad but comparable layout, with all of their external input capacitors far away from the input pins.

It is obvious that the power module LMZM33603 has better radiated EMI performance than the

converter LMR23630. Both layouts are not perfect, but the power module would pass a CISPR test, whereas the converter would fail.

Conclusion

As I mentioned early on, creating a good layout design for a switching power supply is challenging. Even experienced engineers can make errors really quickly, like a non-perfect placement of input capacitors.

Power modules are more forgiving of design layout errors. They are a good choice for your switching power supply when meeting EMI performance and being efficient with your design time are critical.

For additional reading on creating a good layout for reducing EMI, I recommend the application reports, "[AN-2155 Layout Tips for EMI Reduction in DC/DC Converters](#)" and "[AN-643 EMI/RFI Board Design.](#)"

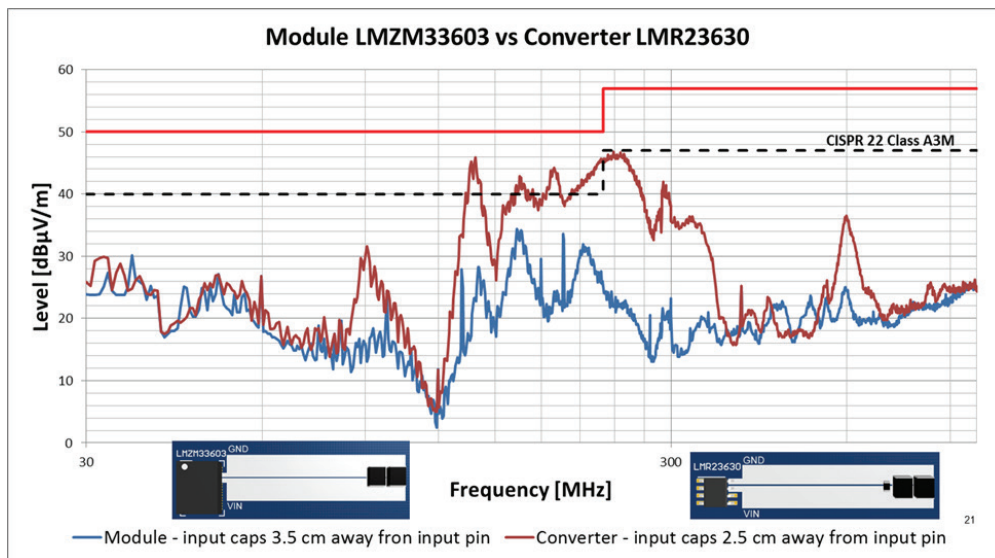


Figure 14. Comparing the EMI performance of the converter LMR23630 and the power module LMZM33603, both from TI.

Important Notice: The products and services of Texas Instruments Incorporated and its subsidiaries described herein are sold subject to TI's standard terms and conditions of sale. Customers are advised to obtain the most current and complete information about TI products and services before placing orders. TI assumes no liability for applications assistance, customer's applications or product designs, software performance, or infringement of patents. The publication of information regarding any other company's products or services does not constitute TI's approval, warranty or endorsement thereof.

The platform bar is a trademark and SIMPLE SWITCHER is a registered trademark of Texas Instruments. All other trademarks are the property of their respective owners.

IMPORTANT NOTICE FOR TI DESIGN INFORMATION AND RESOURCES

Texas Instruments Incorporated ("TI") technical, application or other design advice, services or information, including, but not limited to, reference designs and materials relating to evaluation modules, (collectively, "TI Resources") are intended to assist designers who are developing applications that incorporate TI products; by downloading, accessing or using any particular TI Resource in any way, you (individually or, if you are acting on behalf of a company, your company) agree to use it solely for this purpose and subject to the terms of this Notice.

TI's provision of TI Resources does not expand or otherwise alter TI's applicable published warranties or warranty disclaimers for TI products, and no additional obligations or liabilities arise from TI providing such TI Resources. TI reserves the right to make corrections, enhancements, improvements and other changes to its TI Resources.

You understand and agree that you remain responsible for using your independent analysis, evaluation and judgment in designing your applications and that you have full and exclusive responsibility to assure the safety of your applications and compliance of your applications (and of all TI products used in or for your applications) with all applicable regulations, laws and other applicable requirements. You represent that, with respect to your applications, you have all the necessary expertise to create and implement safeguards that (1) anticipate dangerous consequences of failures, (2) monitor failures and their consequences, and (3) lessen the likelihood of failures that might cause harm and take appropriate actions. You agree that prior to using or distributing any applications that include TI products, you will thoroughly test such applications and the functionality of such TI products as used in such applications. TI has not conducted any testing other than that specifically described in the published documentation for a particular TI Resource.

You are authorized to use, copy and modify any individual TI Resource only in connection with the development of applications that include the TI product(s) identified in such TI Resource. NO OTHER LICENSE, EXPRESS OR IMPLIED, BY ESTOPPEL OR OTHERWISE TO ANY OTHER TI INTELLECTUAL PROPERTY RIGHT, AND NO LICENSE TO ANY TECHNOLOGY OR INTELLECTUAL PROPERTY RIGHT OF TI OR ANY THIRD PARTY IS GRANTED HEREIN, including but not limited to any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI products or services are used. Information regarding or referencing third-party products or services does not constitute a license to use such products or services, or a warranty or endorsement thereof. Use of TI Resources may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

TI RESOURCES ARE PROVIDED "AS IS" AND WITH ALL FAULTS. TI DISCLAIMS ALL OTHER WARRANTIES OR REPRESENTATIONS, EXPRESS OR IMPLIED, REGARDING TI RESOURCES OR USE THEREOF, INCLUDING BUT NOT LIMITED TO ACCURACY OR COMPLETENESS, TITLE, ANY EPIDEMIC FAILURE WARRANTY AND ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF ANY THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

TI SHALL NOT BE LIABLE FOR AND SHALL NOT DEFEND OR INDEMNIFY YOU AGAINST ANY CLAIM, INCLUDING BUT NOT LIMITED TO ANY INFRINGEMENT CLAIM THAT RELATES TO OR IS BASED ON ANY COMBINATION OF PRODUCTS EVEN IF DESCRIBED IN TI RESOURCES OR OTHERWISE. IN NO EVENT SHALL TI BE LIABLE FOR ANY ACTUAL, DIRECT, SPECIAL, COLLATERAL, INDIRECT, PUNITIVE, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES IN CONNECTION WITH OR ARISING OUT OF TI RESOURCES OR USE THEREOF, AND REGARDLESS OF WHETHER TI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

You agree to fully indemnify TI and its representatives against any damages, costs, losses, and/or liabilities arising out of your non-compliance with the terms and provisions of this Notice.

This Notice applies to TI Resources. Additional terms apply to the use and purchase of certain types of materials, TI products and services. These include; without limitation, TI's standard terms for semiconductor products (<http://www.ti.com/sc/docs/stdterms.htm>), [evaluation modules](#), and [samples](http://www.ti.com/sc/docs/sampterm.htm) (<http://www.ti.com/sc/docs/sampterm.htm>).

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2017, Texas Instruments Incorporated