EMI Mitigation Techniques Using the LMZM23601

Jimmy Hua

ABSTRACT

Electromagnetic Interference (EMI) is an unwanted coupling of signals from one circuit or system to another. EMI is separated into two different categories: conducted and radiated. Conducted EMI is a form of conduction coupling caused by parasitic impedance, power and ground connections. Radiated EMI is the coupling of unwanted signals from radio transmission. This application note covers the EMI performance of several different techniques using both printed circuit board (PCB) layout changes and additional external circuitry using the LMZM23601 step-down power module. The PCB layouts in this experiment include the following: original LMZM23601 EVM, EVM with top layer shielding, perimeter fencing with vias, via stitching, and input/output via fencing. Two additional alternative techniques used to mitigate EMI noise are the high frequency(HF) bypass capacitor modification and an input filter design.

Contents

List of Figures

List of Tables

1 Introduction

Switch mode power supplies inherently generate noise emissions because of the high di/dt associated with their operation and parasitic inductance and capacitance in the circuit. EMI tests are often required at system level to ensure the complete system passes the relevant EMC standards. Some of the techniques available to help reduce EMI emissions in switch mode power supplies are input filtering, shielding, spread spectrum, and layout improvements.

This application note presents a study on the contribution of several different layout techniques to the overall radiated EMI performance of the LMZM23601 power module. The LMZM23601 is a 36-V, 1-A Step Down DC-DC integrated-inductor power module that has an output range of 2.5-V to 15-V. The tests in this report were done with 24-V input voltage 5-V output at 1-A load. Tests were performed on the default evaluation board, default evaluation board with an external input filter, default evaluation board with different high frequency(HF) bypass capacitors, and several other evaluation boards featuring different layer stack-up with ground shielding, perimeter via fencing, via stitching, and input/output via fencing. Below is a table for the different variations of EMI board techniques used for this experiment.

Table 1. PCB Layout Design Boards

Board Version	Shielding	Via Stitching (200mil)	Input/Output Fencing (50mil)	Perimeter Fencing (100mil)
1				
2	√			
3	√	√		
4	√		√	
5	√			√
6	√	√	√	√

NOTE: √ = layout technique implemented on the board

2 EMI Mitigation Techniques

The different EMI mitigation techniques explored in this experiment are: HF bypass capacitor modification, top layer ground shielding, via stitching, input/output fencing, perimeter fencing, all techniques, and input filter.

2.1 Default Layout With HF Bypass Capacitors Modification

This technique is a simple modification to the existing default LMZM23601 EVM and does not require any PCB layout stack-up change. By placing a low valued high frequency capacitor close to the input of the step-down converter, the EMI noise is significantly reduced at high frequency hence the HF bypass capacitor modification technique. With the simple addition of the HF bypass capacitors, the LMZM23601EVM was able to pass Class B limits.

Figure 1. Board 1 PCB Layout

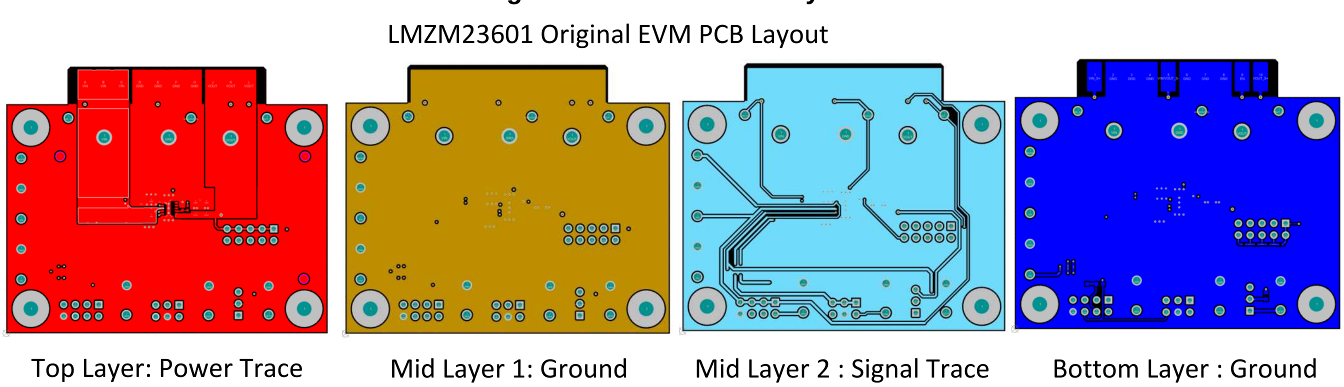
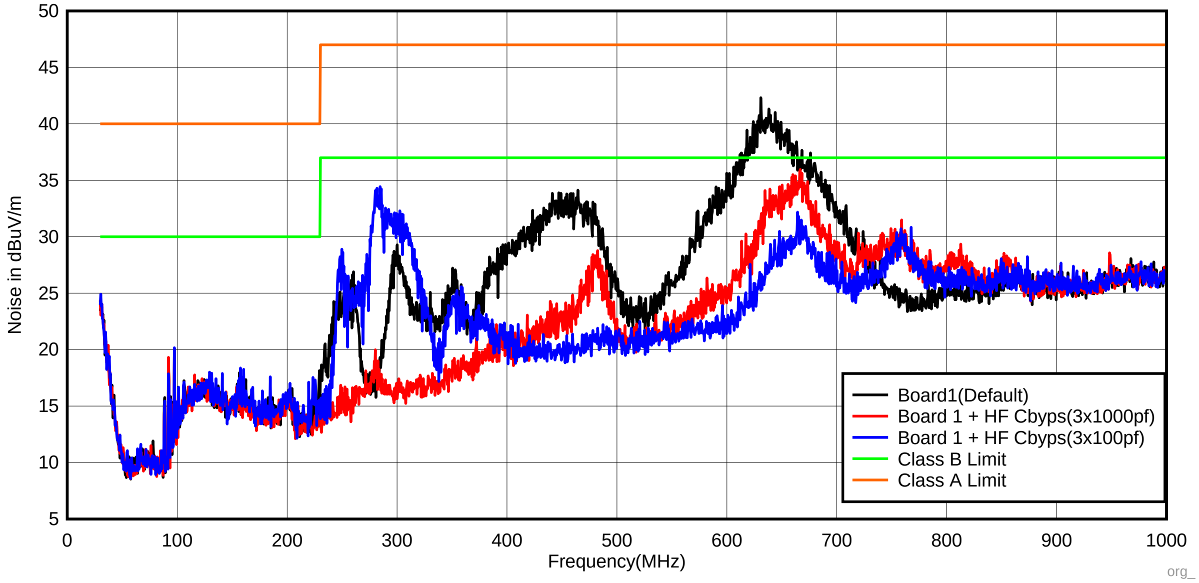


Figure 2. Default EVM vs. Default EVM With HF Bypass Capacitors



2.2 Shielding

The shielding technique implements dedicated ground planes above and below the signal traces to create a low impedance shield that protect against noise. This approach prevents unwanted interference from entering and exiting the sensitive areas of the circuit. Below is a comparison between a default EVM and an EVM with top layer ground plane shield.

Figure 3. Board 2 PCB Layout

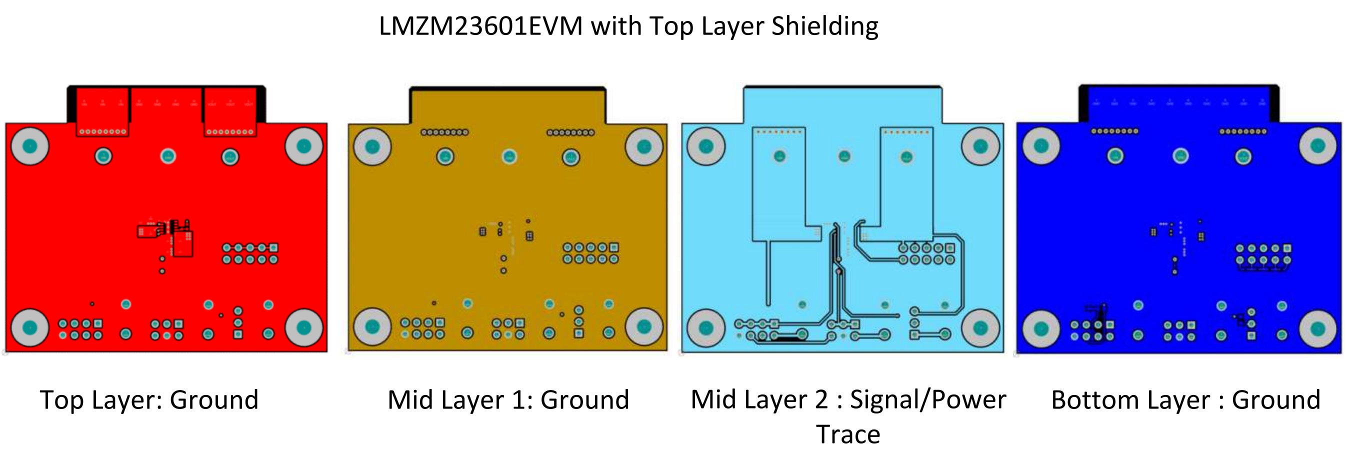
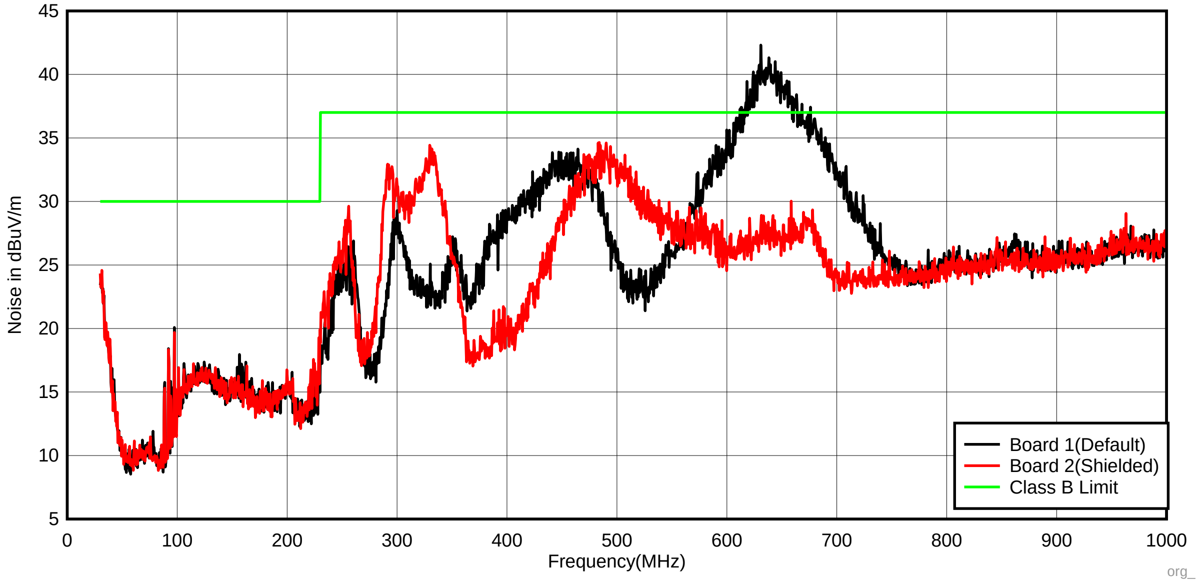


Figure 4. Default vs. Shielded



The radiated EMI test was performed in a 10-meter EMI chamber. The results show that the modified LMZM23601EVM with the top layer ground shielding complies with the CISPR22 Class B test while the original failed with the highest noise observed at around 650MHz. The remaining tests will use the shielded EVM as the controlled unit and Class B line as the limit for noise comparison.

2.2.1 Via Stitching

The stitching technique uses via connections between outer ground layers to mitigate EMI coupling. When using this technique, it is recommended to have the stitching vias evenly spaced around the entire PCB. The results below show that compared to the controlled unit, the addition of via stitching helped further reduce the high frequency noise.

Figure 5. Board 3 PCB Layout

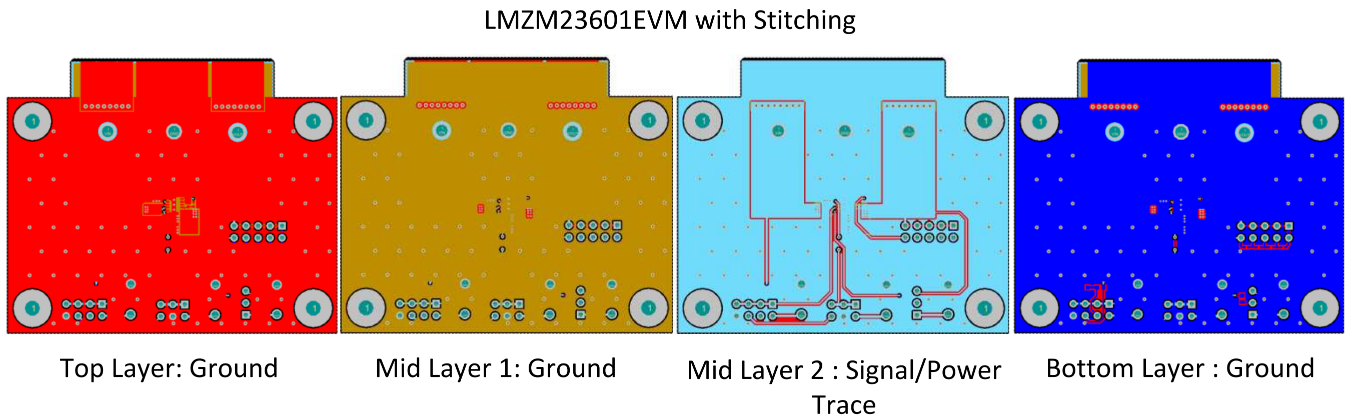
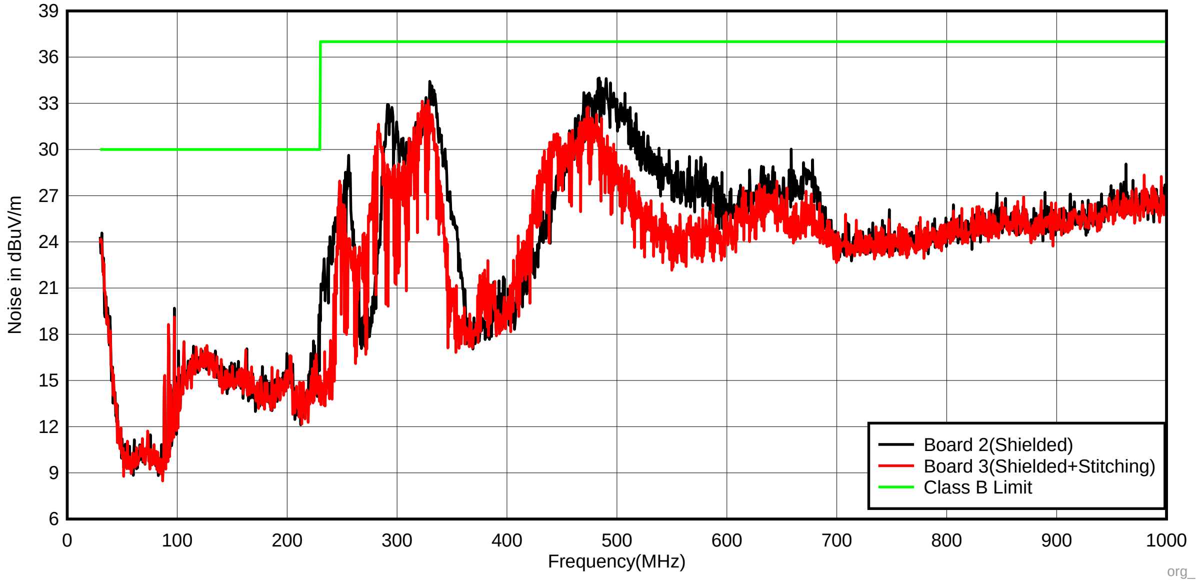


Figure 6. Shielded vs. Shielded + Via Stitching



2.2.2 Input/Output Fencing

Input/Output fencing technique centralizes the the stitching vias into a single array of vias around the input and output signal traces. Input/output fencing shows a slight reduction in noise around 200MHz to 350MHz.

Figure 7. Board 4 PCB Layout

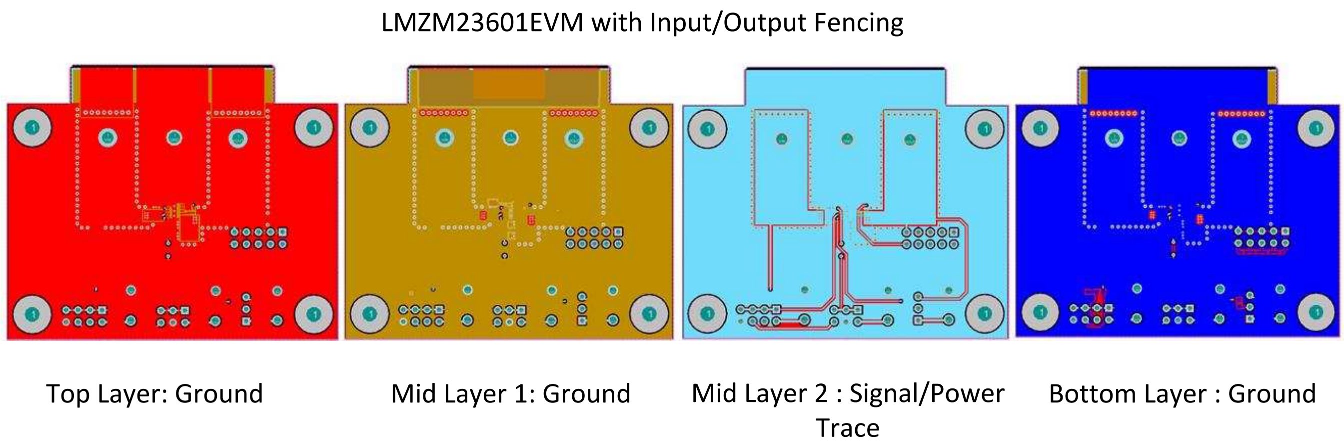
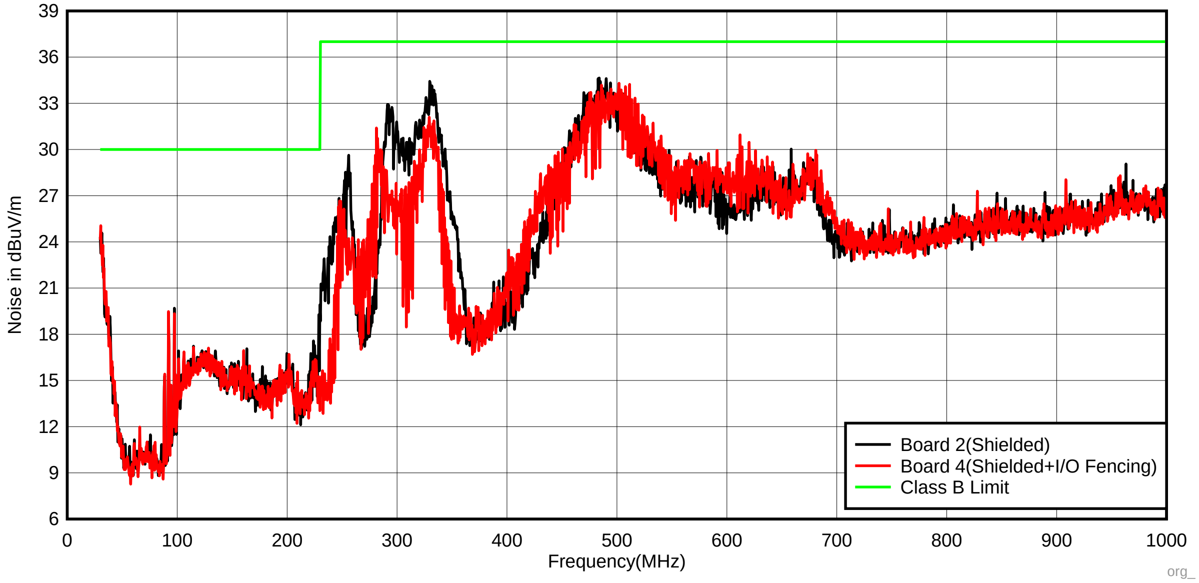


Figure 8. Shielded vs. Shielded + I/O Fencing



2.2.3 Perimeter Fencing

Perimeter fencing uses the same idea as input/output fencing except placed around the perimeter of the PCB. Perimeter fencing result is similar to input/output fencing but has slightly better noise reduction at high frequency range of 600MHz to 750MHz.

Figure 9. Board 5 PCB Layout

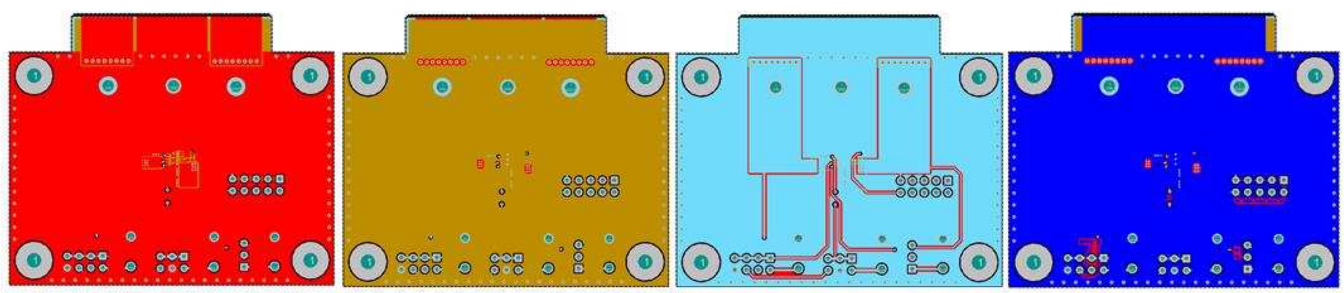
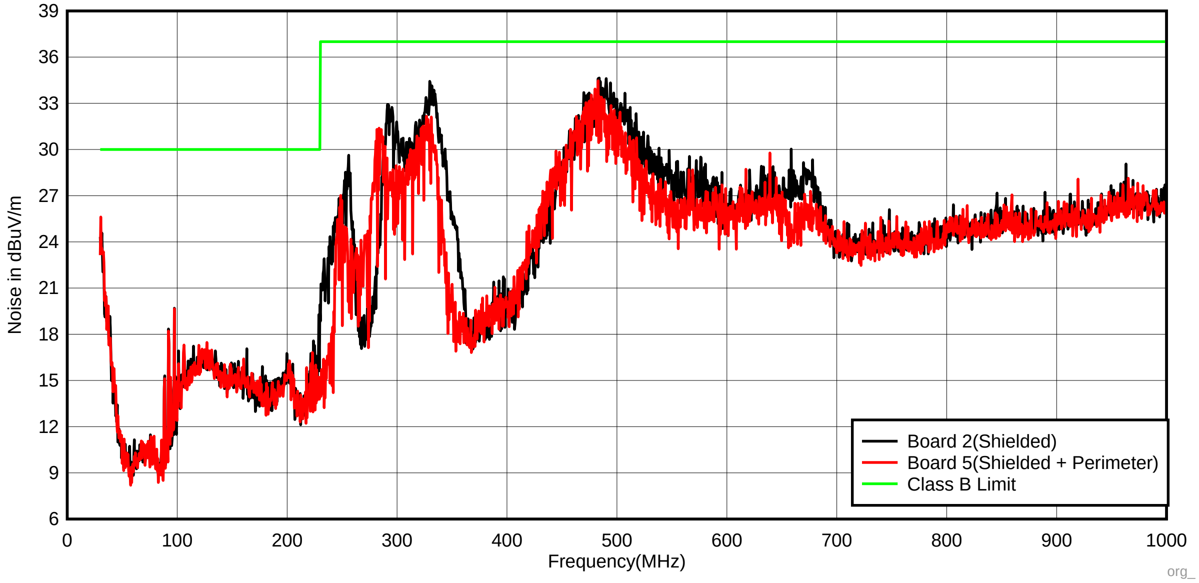


Figure 10. Shielded vs. Shielded + Perimeter



2.2.4 All PCB Layout Techniques

This PCB variation implements all of the PCB layout techniques previously discussed: perimeter, shielding, input/output fencing, and stitching. With the benefits at different frequency ranges using the "All Techniques" method, the result below agrees with the assumption that noise mitigation will improved compared to the controlled unit. Significant reduction in noise can be seen around the 270MHz to 320MHz range.

Figure 11. Board 6 PCB Layout

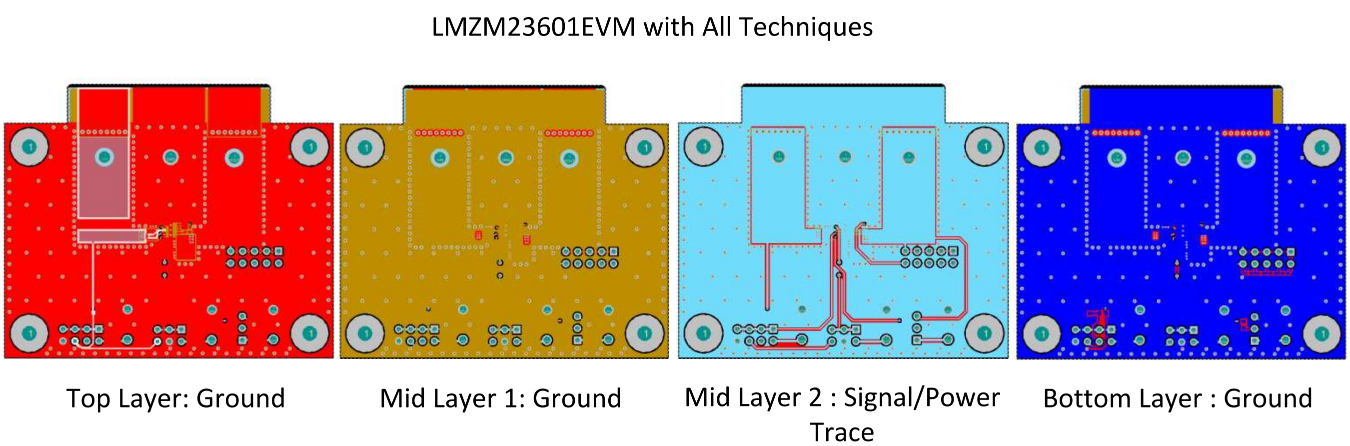
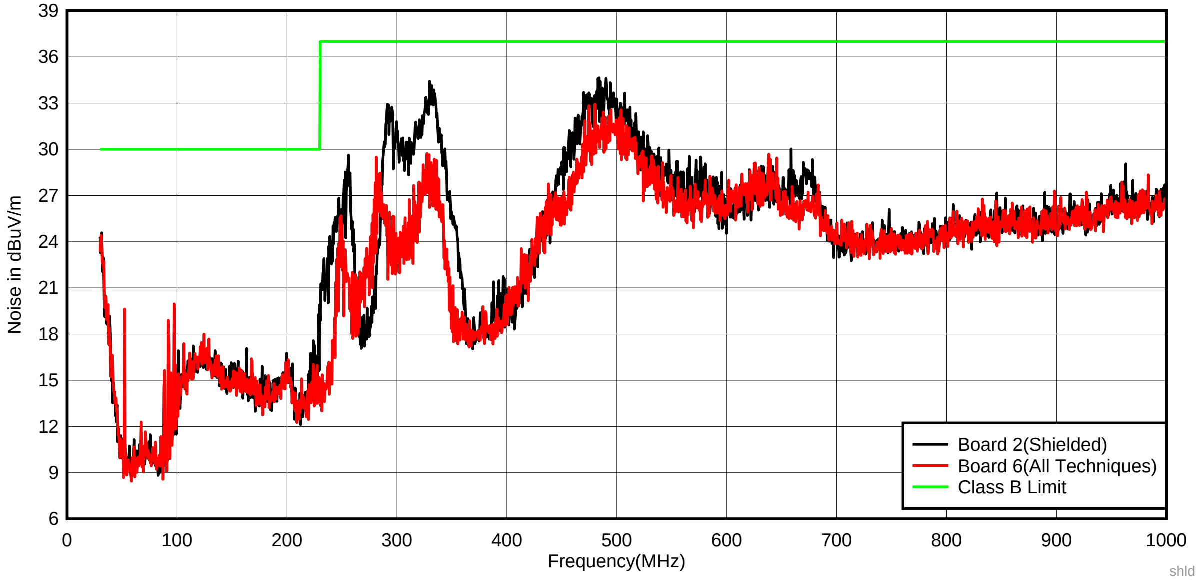


Figure 12. Shielded vs. All Techniques



2.2.5 Input Filter

The most effective way to reduce EMI noise for both conducted and radiated test is through an input filter. Refer to the SNVA489C application note for further information on designing an input filter.

Figure 13. Input Filter Schematic Using the LMZM23601

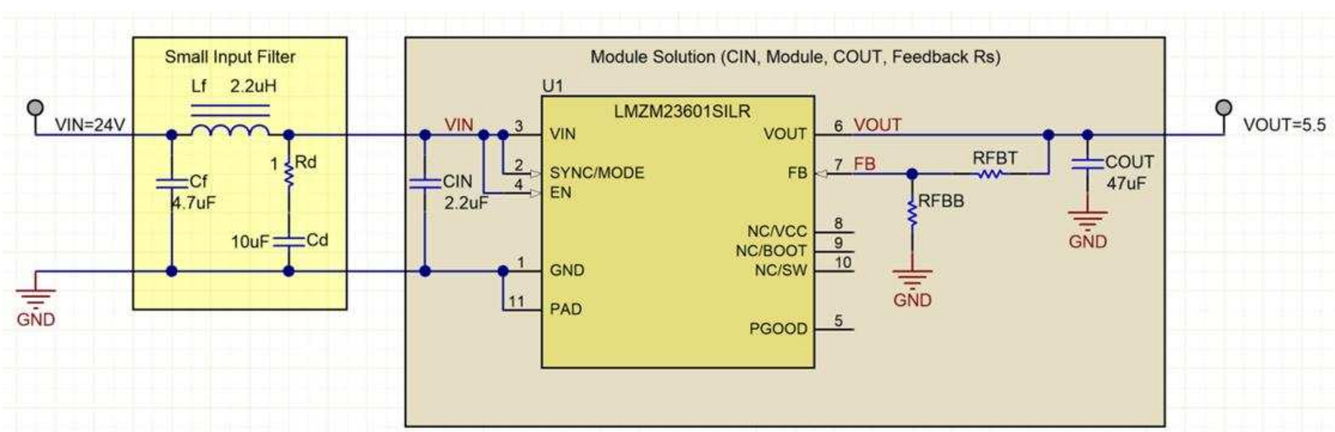
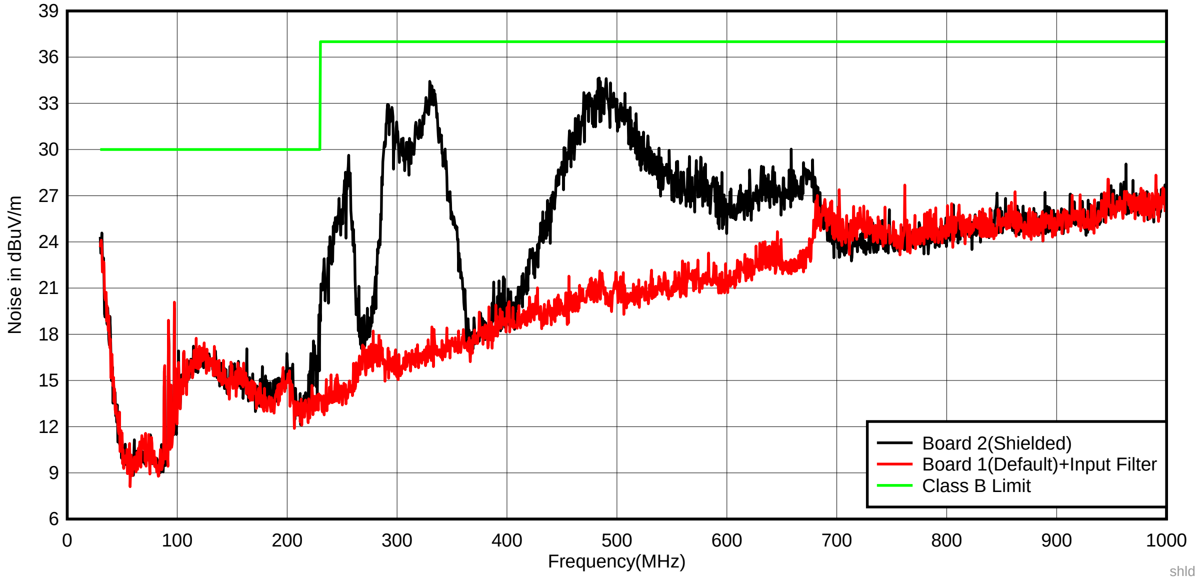


Figure 14. Shielded vs. Default With Input Filter



3 Conclusion

There are several EMI reduction techniques that can be implemented for DC-DC switching converters with each technique presenting different trade-offs. The experiments tested in this application show that adding a few high frequency bypass capacitors will allow the default LMZM23601EVM to pass Class B limits. The PCB layers can be utilized for effective shielding, via stitching, perimeter fencing and input/output fencing to further improve EMI performance of the system. Input EMI filter can be used to significantly improve radiated EMI for step-down converters at the cost of increased BOM.

4 References

1. A. Suntives, A. Khajooeizadeh, R. Abhari, “Using Via Fencing for Crosstalk Reduction in PCB Circuits”
2. Hartley, Richard, “Controlling Radiated EMI Through PCB Stack-Up”
3. X. Ye, D. M. Hockanson, M. Lin, W. Cui, S. Radu, J. L. Drewniak, T. P. VanDoren, T. H. Hubing, R. E. DuBroff, "The EMI benefits of ground plane stitching in multi-layer power bus stacks"
4. N. Moonen, F. Buesink, F. Leferink, "Optimizing Capacitor Placement in EMI-Filter using Back Annotation of 3D Field Coupling Parameters in Circuit Models"
5. F. Gisin, Z. P. Tanner, "Radiation From Printed Circuit Board Edge Structure"