TECHNIQUES TO REDUCE ELECTROMAGNETIC INTERFERENCE IN AUTOMOTIVE APPLICATIONS

By Dr. Kavul Tshiloz, Sr. Systems Engineer
Allegro MicroSystems

ABSTRACT

This application note discusses basic common causes of electromagnetic interference (EMI) and a number of solutions. EMI defines the conducted and radiated emissions of a particular system. Although this application note is targeted at automotive applications, the principles are equally applicable to commercial and industrial applications.

INTRODUCTION

EMI in a motor driver or a point-of-load power-management system is significantly more challenging to resolve in an automotive application than in a commercial application:

- EMI Limits: Commercial applications typically have less-onerous EMI limits.
- Ease of Measurement: Commercial applications are typically easier to measure because EMI filtering solutions can typically be implemented in the DC power supply.
- EMI Housing: In terms of conducted-emission measurements, the equipment under test for a commercial application is generally contained within some form of housing that is metal or otherwise facilitates EMI screening.
- Distance to EMI Source: In terms of radiated measurements, the commercial and industrial standards place the device under test farther from the EMI source (3 to 10 meters from the antenna) than the automotive standard (1 meter from the antenna).
- Range of Test: The standard frequency range tested for conducted emissions, for example, is significantly more narrow in commercial standards (150 kHz to 30 MHz) than in automotive applications (150 kHz to 108 MHz).
- Modifications to Standards: The commercial limit lines have remained largely constant for the past few decades, whereas automotive original equipment manufacturers (OEMs) often have unique standards that frequently change and impose additional limitations.

Automotive EMI standards span a wide frequency spectrum for both conducted emissions and radiated emissions, as presented in the IEC 61967-4 and CISPR 25 standards, respectively. Additionally, the IEC 62132-4 and ISO 11452-4 standards provide conditions and limit lines for immunity testing [i.e., electromagnetic compatibility (EMC) testing].

Many EMI solutions—such as filter capacitors, ferrite clamps, snubbers, shields, gate-driver slew-rate controls, etc.—are designed for specific frequency bands. As such, a single solution for the full-frequency spectrum of emissions does not exist. In addition, complex magnetic or electric field coupling problems that might exist can be difficult to simulate, such as a differential voltage bounce that results in pick-up of electric fields on the output or input power cabling as common-mode noise.

These complex interactions are extremely difficult to predict, and an iterative design or development process is often necessary. This process might involve, for example, changes to the routing of the input power signals and filtering with respect to the noisy power-switching sections of the system. Each system is unique and a solution that works for one system might not work for another system.

REDUCING EMI

EMI reduction can be considered in three parts: 1) Reduce the noise source; 2) Suppress the noise source; and 3) Contain the propagation of the noise. Many techniques can be implemented to reduce, suppress, and contain EMI. The type of solution that can be adopted is often constrained by the physical size of the system, the system environment, and the cost. Good practices, techniques, and solutions to mitigate the effects of EMI are:

- *Always employ best practices for PCB layout:*
  - □ ***Minimize switching-loop sizes:*** For example, in the layout of the buck regulator of an Allegro gate driver, the input, output, and hot loops are the most critical connections. Prioritize the layouts of these paths over all others:
    - ♦ Route the components that form the input, output, and hot loops with short, low-inductance traces.
    - ♦ Match the current path of the input loop to the current path of the output loop.
    - ♦ Minimize the areas of these loops as much as practicable to reduce any unwanted EMI.
  - □ ***Use ground planes to minimize differential noise:*** Differential nose can lead to noise radiation on the input power cables. Ensure that the ground plane is the bottom of a two-layer PCB with minimal traces; if traces are required, ensure that they do not affect any of the power nodes that connect to ground; i.e., ensure there is a direct connection between each power ground connection without the trace interfering and causing a longer route.
  - □ ***Maintain a solid ground plane, use top-to-bottom thermal vias:*** To assist with the heat-sinking of switching components, use thermal vias from the top side of the PCB to the bottom side. Ensure that a solid ground plane is maintained underneath the switching components.
  - □ ***(Recommended) Place PCB traces and planes far (at least 5 mm) from the edge of the PCB board:*** This minimizes noise fringing effects.
  - □ ***Place the EMI filtering components for the input and output stages far from the switching block:*** This helps to minimize possible noise crosstalk between the switching block and the filtering components. This does not negate the need to decouple the local capacitors, which continue to be required directly on the input and output ports of the switching block.
  - □ ***For more-detailed layout guidelines:*** See the Allegro application note Layout Guidelines for PCB Layout of Motor Gate Drivers (AN296278). [1]

- ***Reduce the effects of voltage overshoot and ringing when parasitic components resonate:***
  - □ Consider the case of a buck regulator:
    - ♦ In a buck regulator, the main voltage and current switching waveforms are the main source of noise for both conducted and radiated emissions.
    - ♦ The current waveform through the inductor when the MOSFET in the buck regulator switches is typically trapezoidal or triangular with current spikes at turn-on when the buck regulator is in either continuous or discontinuous conduction mode, as shown in Figure 1.

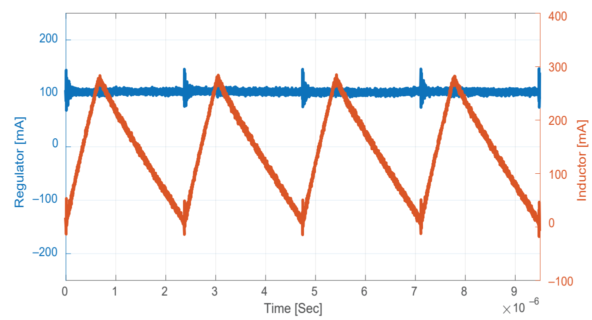

Figure 1: Inductor Current, Buck Regulator, AMT49100

    - ♦ The inductor voltage waveform of the buck regulator built into the AMT49100 Allegro gate driver, obtained at a switching frequency of approximately 410 kHz operating with a supply voltage set to 48 V, is shown in Figure 2.

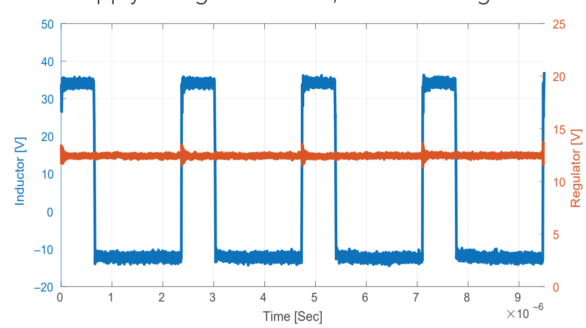

Figure 2: Inductor Voltage, Buck Regulator, AMT49100

    - ♦ The buck regulator as a step-down converter provides approximately 12.5 V and 100 mA to the regulator (the V_{REG} terminal of the AMT49100 gate driver).
    - ♦ The inductor voltage waveform is rectangular with some high-frequency harmonic ringing on the edges.
    - ♦ These waveforms all exhibit a rich harmonic content up to very high frequencies.
    - ♦ The fast-switching edges are necessary to minimize the power dissipation of the gate drivers.
    - ♦ This gives rise to high-frequency noise content due to parasitic leakage inductance and capacitance networks resonating with the fast edges.

[1] https://www.allegromicro.com/-/media/files/application-notes/an296278-pcb-layout-of-motor-gate-drivers.pdf?sc_lang=en

- ◻ Consider the use of one or more of the following techniques to reduce the high-frequency noise content due to parasitic leakage inductance and capacitance networks resonating with the fast edges:
  - ♦ Slow the speed of the gate driver with either: a series resistor in the gate circuit, as shown in Figure 3; or built-in slew-rate control of the gate driver, which in turn reduces both the dV/dt and the dI/dt transients. The tradeoff of this effective noise-source-reduction technique is increased power dissipation.
  - ♦ Apply a lossless snubber network (i.e., a resistor and capacitor network) formed by R1 and C1, as shown in Figure 3.
  - ♦ Apply an additional lossless snubber network (i.e., formed by R2 and C2) if needed.
  - ♦ For a more-detailed analysis of the snubber network design, refer to the Allegro application note Motor Drive Discrete MOSFET Bridge Circuit Design and Layout (AN296215). [2]

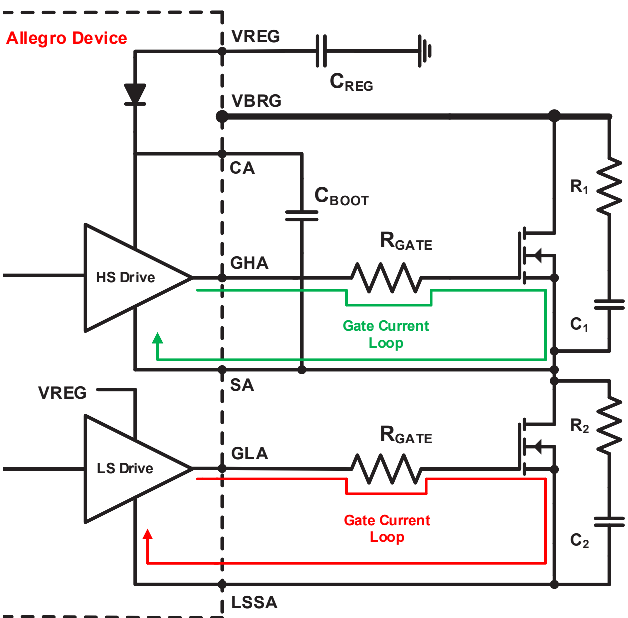

Figure 3: Snubber Network (R1, C1, R2, and C2)

- *Route the input and output cables to minimize mutual inductance:* The input, output, and motor phase cables can act as a transmitter or receiver, where noise on the output side couples onto the input side. The orientation and position of the input cable relative to the output cable significantly affects this mutual inductance. To minimize this mutual inductance, route the output cable parallel to the input cable such that current flow in the cables occurs in opposite directions.
- *Ensure that a supply decoupling capacitor on the input terminal is implemented correctly*: Because an Allegro gate driver is a switching circuit, current spikes occur from all supplies at the switching points. As with all such circuits, the power-supply connections should be decoupled with a ceramic capacitor between the supply terminal and ground.
  - ◻ Select a capacitor that:
    - ♦ Addresses the problem frequency band.
    - ♦ Assures good attenuation over a wider frequency band. This might require more than one capacitor.
  - ◻ Connect the capacitor(s) as close as practicable to the device supply terminals for V_{BB}, V_{DD}, and V_{IO} and the ground terminal, as shown in Figure 4, where the VBB supply decoupling capacitor (C16, highlighted in green) is placed on the bottom layer, as close as practicable to the Allegro U1 device.

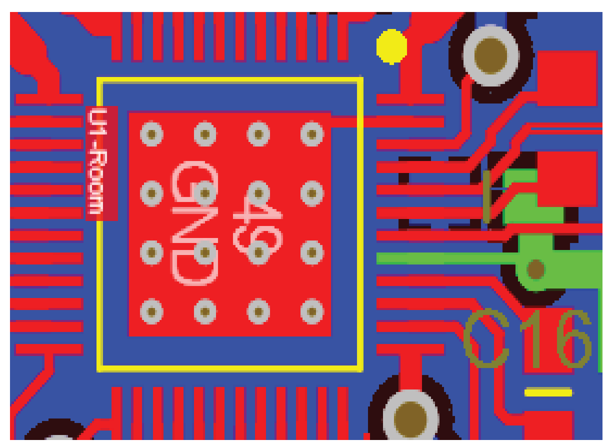

Figure 4: VBB Decoupling Capacitor, C16

[2] https://www.allegromicro.com/-/media/files/application-notes/an296215-motor-drive-discrete-mosfet-bridge-design.pdf?sc_lang=en

- ***Use a π filter if needed:*** If supply decoupling capacitors do not address the problem frequency band, consider the use of a capacitor-inductor-capacitor (CLC) filter, also known as a π filter, on the VBB supply terminal of the gate driver.

  A π filter is an advanced type of filter that has two capacitors (C1 and C2) connected in parallel and an inductor (L) placed in series, as shown in Figure 5:

  - When battery voltage (12 V or 48 V) is applied to the VBBIN input terminal of the π filter as shown in Figure 5, capacitor C1—which reacts minimally to the AC component and greatly to the DC component—passes to ground any AC component noises from the battery supply voltage.
  - Because the inductor offers low reactance to the DC component and high reactance to the AC component, the 12 V or 48 V DC component passes through the inductor.
  - Capacitor C2 performs the same function as capacitor C1: It passes any remaining AC component noises from the inductor to ground, and it provides a very steady DC voltage on the VBB terminal of the gate driver.
  - For better performance, place the components that form the π filter (i.e., the two capacitors and the inductor) as close as practicable to the VBB supply terminal of the employed gate driver.

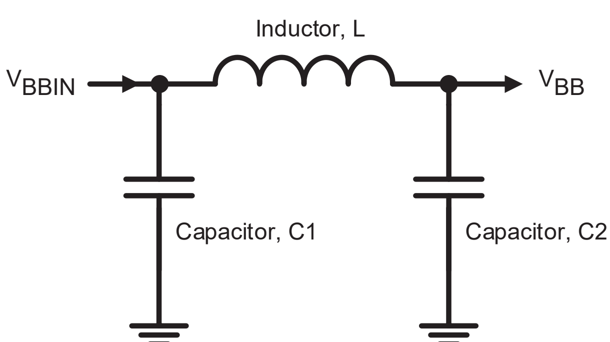

Figure 5: CLC or π Filter

- ***Use a ferrite clamp to attenuate common-mode noise:*** Any common-mode noise that propagates onto the input or output connections can be attenuated with a ferrite clamp applied on the input and output cables.

  - Consider the example: A ferrite clamp can be modeled as a simplified circuit consisting of a capacitor (C_{PAR}), an inductor (L_{CLAMP}), and a resistor (R_{AC}) in parallel and wired with a series resistor (R_{DC}), as shown in Figure 6, where:
    - The series resistor (R_{DC}) quantifies the resistance of the ferrite clamp to DC currents.
    - The inductor (L_{CLAMP}) in the circuit represents the main function of the ferrite clamp: It provides inductive impedance to attenuate high-frequency signals.
    - The parallel resistor (R_{AC}) in the equivalent circuit accounts for losses in eddy currents that are induced within the ferrite clamp at high frequencies.
    - The capacitor (C_{PAR}) accounts for the natural parasitic capacitance of the ferrite clamp.
  - Select a ferrite clamp with consideration for the following:
    - Generally, at low frequency, the frequency response of a ferrite clamp is dominated by its inductive component; and, at higher frequencies near its self-resonant frequency, the frequency response of a ferrite clamp is dominated by its resistive component. The capacitive element becomes dominant at frequencies higher than the self-resonant frequency.
    - In practice, a ferrite clamp allows signals within a certain frequency range to pass with minimal effect but dissipates other signals at higher frequencies as heat.
    - Selecting the right ferrite clamp for the input and output cables requires careful consideration for the impedance at the frequencies of interest, as well as the impedance characteristics of the ferrite clamp with respect to DC bias current: As the DC bias current increases, the core material of the ferrite clamp starts to saturate, which can significantly reduce the inductance of the ferrite clamp and can degrade its EMI filtering capability.
  - Consider the position of the ferrite clamp: Placement as far as practicable from the system generally achieves the best results.

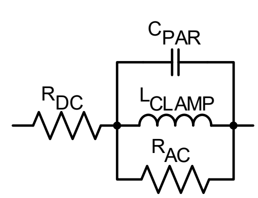

Figure 6: Ferrite Clamp Equivalent Circuit

EXPERIMENTAL RESULTS

Experimental tests were performed in the Allegro EMC test laboratory to investigate the performance of the conducted emissions (CE) and radiated emissions (RE) tests of the AMT49105 gate driver. In the tests, the placement of the supply decoupling capacitor connection was varied. It was placed: 1) as close as practicable to the VBB terminal of the AMT49105; and 2) as far as practicable from the VBB terminal of the AMT49105 gate driver.

In these tests, A 10 cm by 10 cm four-layer PCB was designed and populated to ensure good radio-frequency parameters of the coupling and decoupling networks. Ground via stitching was used to tie together the larger copper areas in layers. This approach leverages the board structure to create a strong vertical connection that maintains low impedance and short return loops for the AMT49105 test board.

A via hole was made through the PCB laminates, across four adjacent layers of the 10 cm by 10 cm designed board, as shown in Figure 7. This via hole provides an electrical connection of the VBB terminal of the AMT49105 gate driver from the top layer to the bottom layer.

The first experimental test was performed with ceramic supply decoupling capacitors, C3 (100 nF, 50 V) and C4 (100 pF, 50 V) placed on the bottom layer as close as practicable to the VBB terminal of the Allegro AMT49105 device, as shown in Figure 8.

The second experimental test was performed with the ceramic supply decoupling capacitors, C_3 (100 nF, 50 V) and C_4 (100 pF, 50 V), superimposed and placed as far as practicable from the VBB terminal of the AMT49105 gate driver, as shown in Figure 9.

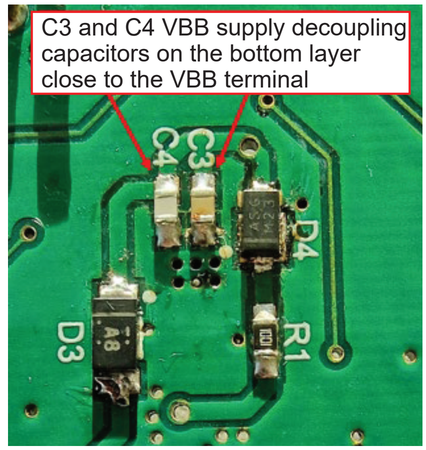

Figure 8: Supply Capacitors Close to VBB, Recommended

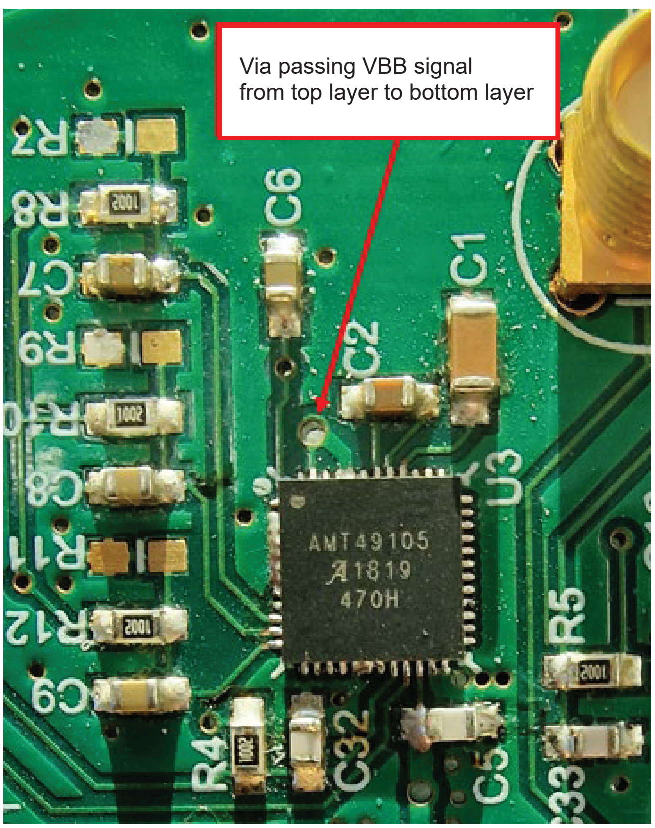

Figure 7: VBB Via Hole from Top to Bottom

Figure 9: Supply Capacitors Far From VBB, Not Recommended

CONDUCTED EMISSIONS

Conducted emissions are noise components that can be generated by any device under test and can be transferred to other devices through cables, board traces, power planes, ground planes, and parasitic capacitance. Tests of the conducted emissions for the VBB terminal were performed according to the IEC 61967-4 and CISPR 25 standards.

The test bench contains:

- A high-frequency Rohde and Schwartz ESPI7 receiver with a frequency band of 9 kHz to 7 GHz.
- A Rohde and Schwartz line-impedance stabilization network (LISN) V-NETWORK ESH3-Z6.
- A 12 V car battery to supply the board.
- An AMT49105 Allegro device, as shown in Figure 7, Figure 8, and Figure 9.

The conducted emissions tests consist of:

1. Base measurement of the AMT49105 device on the board in the powered-off state.
2. Powered-on measurement with the supply decoupling capacitors connected as close as practicable to the VBB terminal, as shown in Figure 8.
3. Powered-on measurement with the supply decoupling capacitors connected as far as practicable from the VBB terminal, as shown in Figure 9, with consideration for the conducted emissions test parameters in Table 1.

The results obtained are presented in Figure 10, Figure 11, and Figure 12:

- The base measurement with the powered-off AMT49105 are shown in Figure 10. These measurements are performed first to ensure that emissions from the AMT49105 are not present prior to power-up of the device. Figure 10 demonstrates that both the peak and average data are significantly below the limits of CISPR 25 Class 5.
- The measured conducted emissions with the powered-on AMT49105 device with supply decoupling capacitors connected as close as practicable to the VBB terminal are shown in Figure 11: From 150 kHz to 1 MHz, both the peak data and the average data exceed the emissions presented in Figure 10 for the same bandwidth; however, the measured conducted emissions data in Figure 11 are significantly below the CISPR 25 Class 5 limits. Thus, these results are acceptable per the CISPR 25 Class 5 limits.
- The measured conducted emissions with the powered-on AMT49105 device with supply decoupling capacitors connected as far as practicable from the VBB terminal are shown in Figure 12: The average data failed to pass 1) the average CISPR 25 Class 5 limit of 50 dBµV for the frequency bands 0.15 MHz to 0.2625 MHz and 0.267 MHz to 0.3 MHz, and 2) the average CISPR 25 Class 5 limit of 34 dBµV for the frequency bands 0.53 MHz to 0.5775 MHz, 0.591 MHz to 0.6495 MHz, 0.6765 MHz to 0.726 MHz, 0.7575 MHz to 0.8025 MHz, and 0.8565 MHz to 0.879 MHz.

Table 1: Conducted Emissions, Test Parameters

Parameters	Settings	
Frequency [MHz]	**Range**	**Step**
	0.15 to 30	0.009
	30 to 108	0.120
Test Signals	VBB	

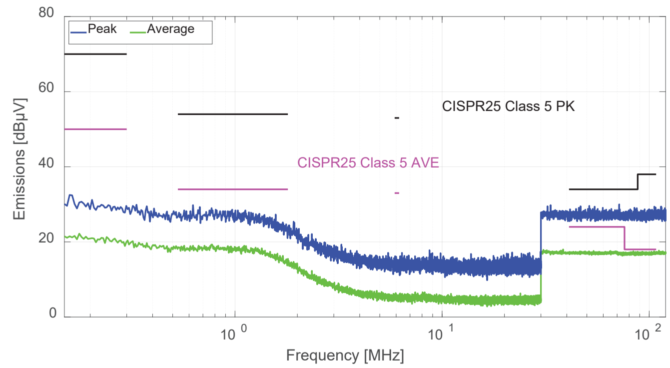

Figure 10: CE Base Measurement, Powered-Off AMT49105

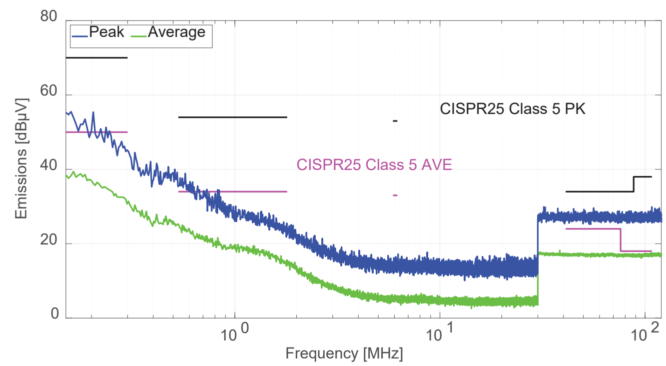

Figure 11: CE Measurement with Capacitor Close to VBB, Powered-On AMT49105

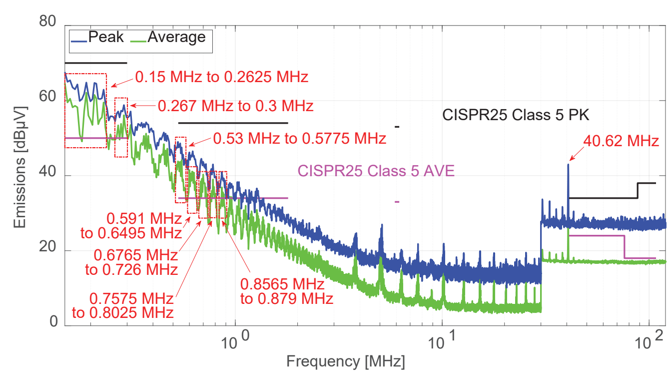

Figure 12: CE Measurement with Capacitor Far From VBB, Powered-On AMT49105

RADIATED EMISSIONS

Radiated emissions testing is the measurement of the electromagnetic field of the emissions that are unintentionally generated by any device under test. To span the entire frequency band of interest, two different antennas were used in the tests. Tests of the radiated emissions of the VBB and ground terminals were performed per the CISPR 25 standard.

The test bench comprises:

- A high-frequency Rohde and Schwartz ESPI7 receiver with a frequency band of 9 kHz to 7 GHz.
- Aa Rohde and Schwartz line-impedance stabilization network (LISN) V-NETWORK ESH3-Z6.
- A low-frequency (9 kHz to 30 MHz) active rod HFH2-Z6 antenna.
- A high-frequency (30 MHz to 3 GHz) HL562 ultralog antenna.
- A 12 V car battery to supply the board and the AMT49105 Allegro device.

Similar to the conducted emissions tests, the radiated emissions test involves:

1. Base measurements of the powered-off AMT49105 device on the board.
2. Measurements of the powered-on AMT49105 with the supply decoupling capacitors connected as close as practicable to the VBB terminal, as shown in Figure 8.
3. Measurements of the powered-on AMT49105 with the supply decoupling capacitors connected as far as practicable from the VBB terminal, as shown in Figure 9, per the radiated emissions test parameters in Table 2.

The radiated emissions data obtained for the low-frequency band of 150 kHz to 30 MHz with the rod antenna are shown in Figure 13, Figure 14, and Figure 15:

- The base measurements with the powered-off AMT49105 are shown in Figure 13. These results, which do not show a failure, are used as a forerunner to the powered-on device tests.
- The measured radiated emissions with the powered-on AMT49105 with supply decoupling capacitors connected as close as practicable to the VBB terminal are shown in Figure 14: From 150 kHz to 0.4 MHz, the peak and average signals both exceed the emissions in Figure 13 for the same frequency band. Nevertheless, the measured radiated emissions data in Figure 14 are significantly below CISPR 25 Class 5 limits. Thus, the results obtained are acceptable per the CISPR 25 Class 5 limits.
- The measured radiated emissions with the powered-on AMT49105 with supply decoupling capacitors connected as far as practicable from the VBB terminal are shown in Figure 15: The average data failed to pass 1) the average CISPR 25 Class 5 limit of 26 dBμV/m for the frequency band of 0.15 MHz to 0.3 MHz, and 2) the average CISPR 25 Class 5 limit of 20 dBμV/m for the obtained frequency bandwidth of 0.53 MHz to 0.915 MHz.

Table 2: Radiated Emissions, Test Parameters

Parameters	Settings	
	Range	Step
Frequency [MHz]	0.15 to 30	0.009
	30 to 1000	0.120
Test Signals	VBB and GND	

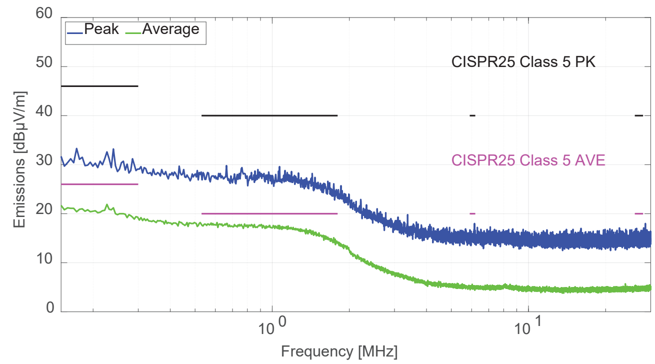

Figure 13: RE Base Measurement with Powered-Off AMT49105 and Rod Antenna

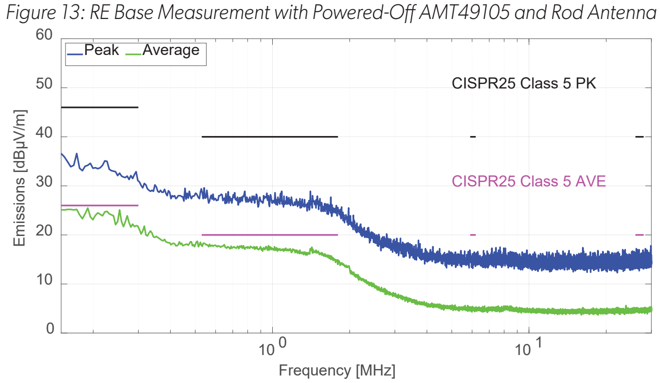

Figure 14: RE Measurement with Capacitor Close to VBB, Powered-On AMT49105, and Rod Antenna

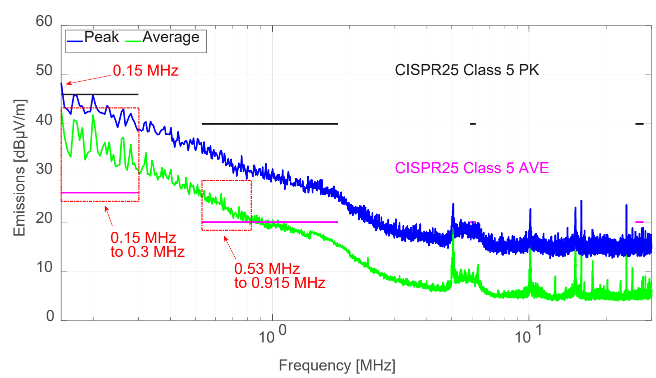

Figure 15: RE Measurement with Capacitor Close to VBB, Powered-On AMT49105, and Rod Antenna

The radiated emissions data obtained for the high-frequency band of 30 MHz to 1 GHz using an ultralog antenna in the horizontal position are shown in Figure 16, Figure 17, and Figure 18; and the radiated emissions results for the same high-frequency band using an ultralog antenna in the vertical position are shown in Figure 19, Figure 20, and Figure 21:

- The base measurement results with the powered-off AMT49105 with the ultralog antenna in the horizontal and vertical positions are shown in Figure 16 and Figure 19, respectively.
- The base measurement data do not show a failure, and the signal patterns of the peak and average data are similar to the measured radiated emissions for the powered-off device with supply decoupling capacitors connected as close as practicable to the VBB terminal for the horizontal and vertical positions, as shown in Figure 17 and Figure 20, respectively.

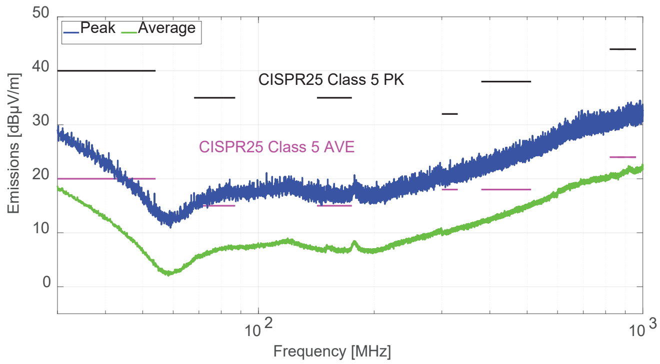

Figure 16: RE Base Measurement, Powered-Off AMT49105, Ultralog Antenna in Horizontal Position

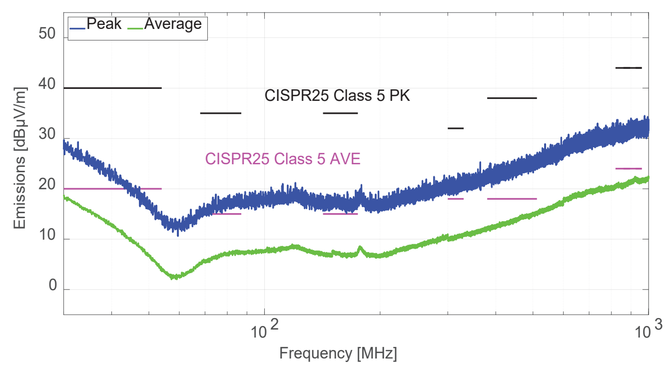

Figure 19: RE Base Measurement, Powered-Off AMT49105, Ultralog Antenna in Vertical Position

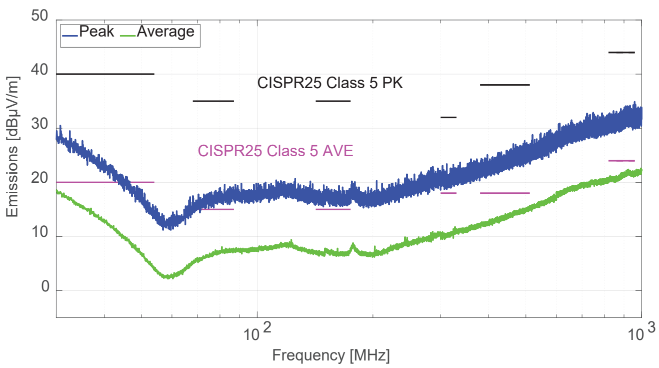

Figure 17: RE Measurement with Capacitor Close to VBB, Powered-On AMT49105, Ultralog Antenna in Horizontal Position

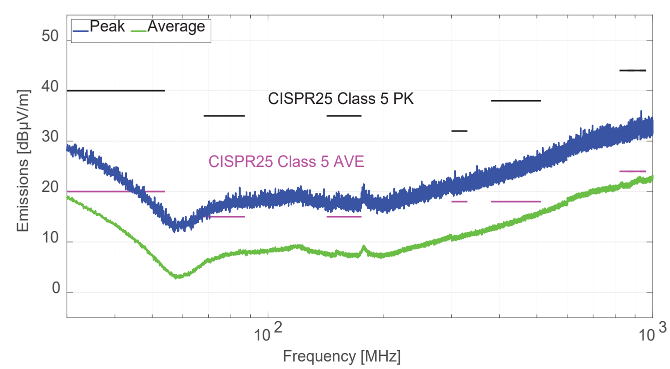

Figure 20: RE Measurement with Capacitor Close to VBB, Powered-On AMT49105, Ultralog Antenna in Vertical Position

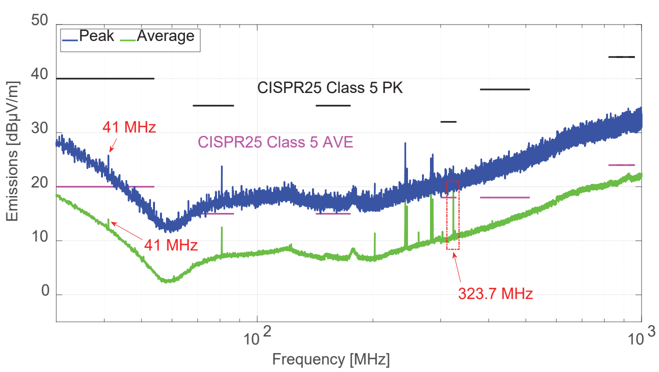

Figure 18: RE Measurement with Capacitor far from VBB, Powered-On AMT49105, Ultralog Antenna in Horizontal Position

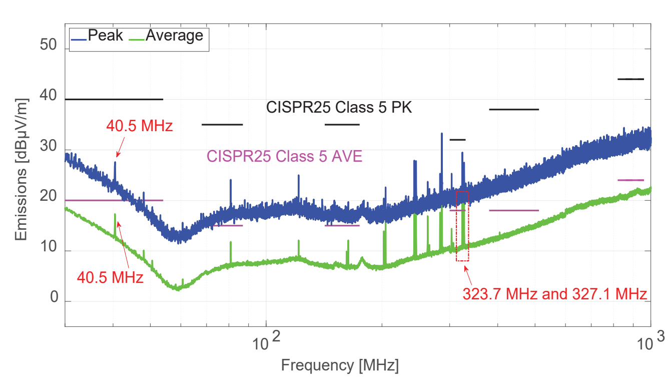

Figure 21: : RE Measurement with Capacitor far from VBB, Powered-On AMT49105, Ultralog Antenna in Vertical Position

- Experimental data with the powered-on AMT49105 with supply decoupling capacitors placed as far as practicable from the VBB terminal are shown in Figure 18 and Figure 21. The average results failed to pass the average CISPR 25 Class 5 limit of 18 dBμV/m at 323.7 MHz for the horizontal position of the ultralog antenna in Figure 18; and the average data failed to pass the average CISPR 25 Class 5 limit of 18 dBμV/m for the frequency bandwidth of 323.7 MHz to 327.1 MHz for the vertical position of the ultralog antenna.

CONCLUSION

Obtaining an automotive electromagnetic interference solution for gate drivers of motor control systems is a major technical challenge. Experimental results demonstrate that a supply decoupling capacitor placed as close as practicable to the VBB terminal of the gate driver can improve the performance of the conducted and radiated emissions.

EMI should be considered at the beginning of a project, during the design or planning phase, well before the qualification stage.

Many different techniques to reduce EMI are presented in this application note. Provision should be made for as many of these solutions as possible. Although all solutions might not be required, due to the complex nature of EMI, it is always wise to have available options: Arriving at a suitable solution can often be an iterative process, and several PCB layouts with different filter arrangements might be necessary.

Revision History

Number	Date	Description	Responsibility
–	July 6, 2026	Initial release and publication to Allegro website	Dr. K. Tshiloz

955 PERIMETER ROAD • MANCHESTER, NH 03103 • USA
+1-603-626-2300 • FAX: +1-603-641-5336 • ALLEGROMICRO.COM