Automotive EMI Webinar

Hosted by Christian Kueck, Senior FAE

June 6, 2018

Q&A

A copy of the presentation may be found here.

What do you think about using ground islands?

Use them whenever possible and ground them with at least one via. GND should be the second layer so the GND layer will be kept intact. You improve the E-Field and H-Field (eddy current) shielding for the top-side components.

What was the value of the capacitor used for the EMI probe?

1uF 0805. But other than the case size, the capacity does not matter much since the impedance will be low against the characteristic coax cable impedance of 50Ohm for the frequency of interest.

For 24v to 5v step down what should be the frequency?

For EMI reasons, it should be below the AM band since the switch node moves 24V and creates a lot of E-Field, and with its switch capacities there's a lot of H-Field in the hot loop. Fixed frequency EMI goes up with 6db/octave. With perfect spread spectrum, still 3dB/octave.

For Bucks and Boosts its more or less clear. What about SEPIC, where to expect the most critical EMI nodes there? Is it similar to boost in that sense, or something else?

SEPIC hot loop is GND-SW-SEPIC_capacitor-output_diode/switch-output_ capacitor-GND. There is no inductor involved in the hot loop. Keep the hot loop small or even fold it to an 8.

I have read about ground traces between signal traces also help. Could you explain how that works?

Every conductor where current can flow in the opposite direction of a signal trace will reduce the E and H field radiation. From a distance you will "see" the sum of all currents so with opposite direction flow you will get reduced fields.

Some designs leave an open GND cut under a phantom inductor (A voltage phantom feed for AM/FM that rejects AM/FM band) and it is debated as to what is best practice. What is the best practice?

A solid GND under the inductor is a short circuit winding where the eddy current will reduce efficiency and create noise in that plane. However, for a non-isolated design, the layer stack should be kept as quiet as possible; place under the inductor in the top layer a solid copper area connected to Vout for buck or Vin for boost. This will take some of the eddy current density out of the next solid GND layer and keeps it more quiet.

Does capacitor help for emissions above 1GHz ?

Yes. As long as the impedance is lower than the characteristic trace or cable impedance they decouple. That goes above 1GHz for the typical SMD cases. Even if the ceramic gets lossy at high frequency (X7R etc) it does decouple. You can prove that effect easily with a small wideband magnetic probe like Langer MSA-R-0, 2-75.

Will the EMC issue have more influence if the Buck Boost voltage output is connected to LED load connected away from driver PCB?
As soon as you have any connections from input or output off the board, you will need to filter it, because the remaining RF currents on those traces radiate as they would in any antenna with the length of the cable. So yes, you will need additional filtering if you go off board with the output i.e. to a LED string.

If emission is coming from the internal freq. of DC-DC converter what is the best solution?
Internal oscillators on the IC have very little energy and do not radiate if its supply is decent decoupled. The switch currents are several orders of magnitude higher and they form usually with the hot loop the main H-field/magnetic EMI problem.

is it important to keep the feedback loop short? Same question for snubber?
Fb is a high impedance node that can in most cases not be simply decoupled by capacitors because this influences the regulation loop. Keep it short because it will easily pick up any E-Field noise due to its high impedance. If you need to route the Vout over a longer distance use the low impedance Vout signal and place the Fb network close to the IC but away from the switch node and hot loop.

How to select the input EMI filter, can we get any good application notes from MPS?
For automotive supplies, use a 4th order input filter with two inductors. Place the filter away from the hot loop or use at least any shielding components between the Hot-loop, Switch node and inductor. Make distance between the two filter inductors and their capacitors since they couple magnetic. *(And stay tuned for an application note)*

Assuming a shielded inductor, what impact does core material selection have? Which type is preferred?
There are some "shielded" inductors that fill the gap with some magnetic material. That works typical below the FM range. There are with a conductor shielded inductors like Vishay IHLE. That shielding works but you lose some efficiency due to eddy currents and the switch connection is still large and radiates E-Field. Currently there is no perfect inductor available, some work better on EMI than others. In general, the larger the physical size (especially height), the more they radiate.

Separate board of Buck boost converter and separate board for LED eg Lamp?
See answer above. You need to filter because the cable will radiate.

If you place/route a switched mode power supply in U shape to reduce the loop in the input components with the output components, do you recommend separate the input capacitors to reduce the magnetic coupling between capacitors?
If you plan to use more than one MLCC input capacitor, place them in a line with GND to the outside and Vin in the middle. Ripple current in each capacitor is cut in half (-6dB H-field) and you get a cancellation pattern with some quiet spots. Probe around with an H-Field probe to get an idea.

The input hot loop consists of switch FET and Diode (or synchronous FET). But they do not switch at the same time and the loops are somewhat different. So the effects should not add. Can you elaborate?
The inductor will force its current any time. If you switch the top FET off and the bottem FET on with some delay (dead time), the body diode of the bottom FET will conduct in the meantime. In the transition, the current path changes from the bottom FET to the top FET and with that the hot loop sees a transition from zero to full current in the transition time which is single digit nanoseconds, if not faster.

Could you show where grounds connect together? Where is the common point?
For higher current (1A+) automotive DC-DC, you try to separate the noisy GND (hot loop) from the quiet GND where the EMI filter is connected to. Connection can be a single via block or anywhere in quiet areas.

If I´m designing for a truck powered by 48V I should use lower frequency?
Yes. Definitely below the AM band.

Can I take I sense signal under inductor ....?
Under the inductor there is still a lot of magnetic noise. If you have several solid GND layers under the inductor in a 6 layer board or more you can try to route sense signals under the inductor. If you have 4 layers or less no chance. In general, avoid routing sensitive lines under or near the inductor. You can check with a magnetic field probe if your backside under the inductor is quiet.