Version N° 1.1

LaurTec

EMC Testing

PCB Layout and noise coupling

Sponsored by

PCBWay

Author: *Mauro Laurenti*

ID: EMC-010-EN

License

Abstract

Engineers do not get surprised by hearing that for precision or RF applications, the PCB must be considered as an electronic component part of the system itself. Each trace routed on the PCB is part of the signal chain and must be carefully designed. This perspective should be actually taken beyond the precision and RF applications, indeed stringent regulations, such as the CISPR standards, apply to any electronic system. CISPR standards cover automotive, industrial and commercial systems and are used by different countries, such as Europe and US, as basis to validate the conformity of a certain electronic system from EMC (Electromagnetic Compatibility) perspective. In this article it will be shown how different trace layouts made on a standard FR4 PCB, may drastically affect the system radiation that may compromise the end product certification, either from conducted or radiated tests perspective. The article will highlights how a PCB should be carefully designed as part of the system, whatever the application would be, to make sure that you achieve the right performances but also you will comply with the EMC standards required for the FCC and CE marking.

PCB Test boards layouts

The PCB used for the tests, have been made using standard FR4 material with a typical εr=4.3. The trace width has been sized to get 50Ω typical line impedance with two layers design and a PCB thickness of 1.6mm. The additional choice of avoiding impedance control was made to get closer to a real application. Indeed impedance control would add additional cost that typically are worth and affordable for RF applications but the rest of customers will try to avoid it. Nevertheless it is important to know whether or not your PCB manufacturer partner can support it. Indeed, out of the first impedance computation the manufacturer can further trim and match the traces to the wanted impedance. The PCBs used for the measurements have been produced by PCBWay, which offers, beside impedance control service, also other PCB materials support that may be beneficial for high performance PCBs, such as Aluminum and Rogers material. The PCBs that have been designed are shown in Figure 1 together with the transversal section. A brief description is the following:

- 50Ω Microstrip with bottom GND on the edge of the trace (**ID**: 1-1)
- 50Ω Microstrip with good layout, GND covering bottom layer (**ID**: 1-2)
- 50Ω Coplanar Microstrip on the edge – top GND is cut (**ID**: 1-3)
- 50Ω Coplanar Microstrip with good layout – top GND (**ID**: 1-4)
- 50Ω Coplanar Microstrip with good layout – top and bottom GND (**ID**: 1-5)
- 50Ω Coplanar Microstrip on the edge – top and bottom GND is cut (**ID**: 1-6)

Each PCB has two SMA connectors to enable the connection of 50Ω termination on one side and connect the spectrum analyzer on the other side. To ease the tests, allowing a good repeatability, a PCB fixture has been designed and 3D printed in PLA material.

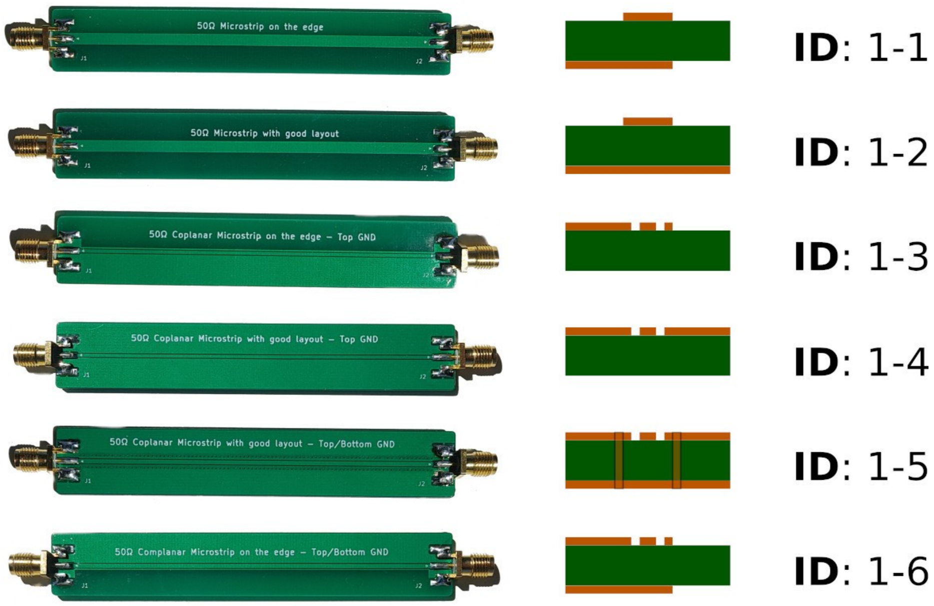

Figure 1: *PCBs used for the tests. The transversal sections is shown for reference.*

The PCBs are kept 6cm from the table and the center of each PCB is kept at 2cm from each other. The setup details are shown in Figure 2.

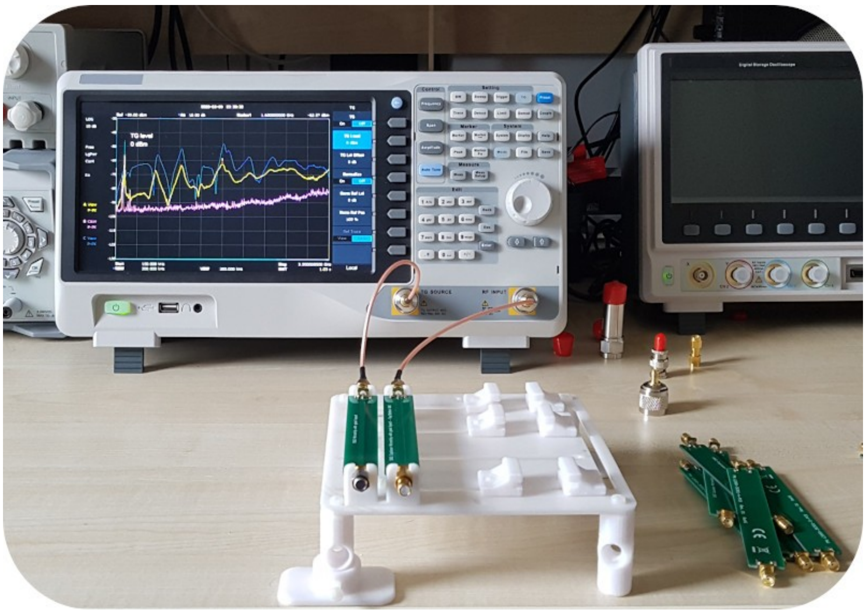

Figure 2: *Test setup showing the PCB fixture details.*

Coupling and conducted noise measurements

To test the PCBs, two types of measurements have been performed:

- Coupling between PCBs using the PLA PCB fixture (Figure 2).
- Coupling between PCB ID: 1-1 and a DC-DC converter board while running a CISPR 25 compliance test.

Test setup 1

Figure 3 shows several curves, between the frequency range of 150KHz and 3GHz. The receiving PCB used as reference was ID: 1-5 (50Ω Coplanar Microstrip with good layout – top and bottom GND connected with vias). The PCB ID: 1-5 was connected to the input of the Spectrum analyzer and terminated with 50Ω load on the other side. All the other PCBs were connected, one by one, to the Tracking Generator output and terminated as well with 50Ω, allowing the frequency scanning between 150KHz and 3GHz. The Tracking Generator output power was 0dBm, thus the amplitude in Figure 3 represents the coupling between the two PCBs or the S21 parameter. It is interesting to see how a very simple layout, that may depict typical layout traces used on simple routing schemes, may create a radiated field and coupling noise delta that varies between 10-30dBm. The PCB resonances may make this delta slightly better or worse. The board ID: 1-5 is the one that behaves better, showing the highest attenuation. It was used as reference since it was supposed to have low coupling, but for reciprocity it was expected to be also a small source of noise. The one that behaves worse than the others, is the ID: 1-1, where the top Microstrip has a GND layer on the bottom that is just on the edge of the trace. This creates an electromagnetic field that has challenges to remain close to the GND, thus it moves farther away from the PCB and couples better with the victim PCB.

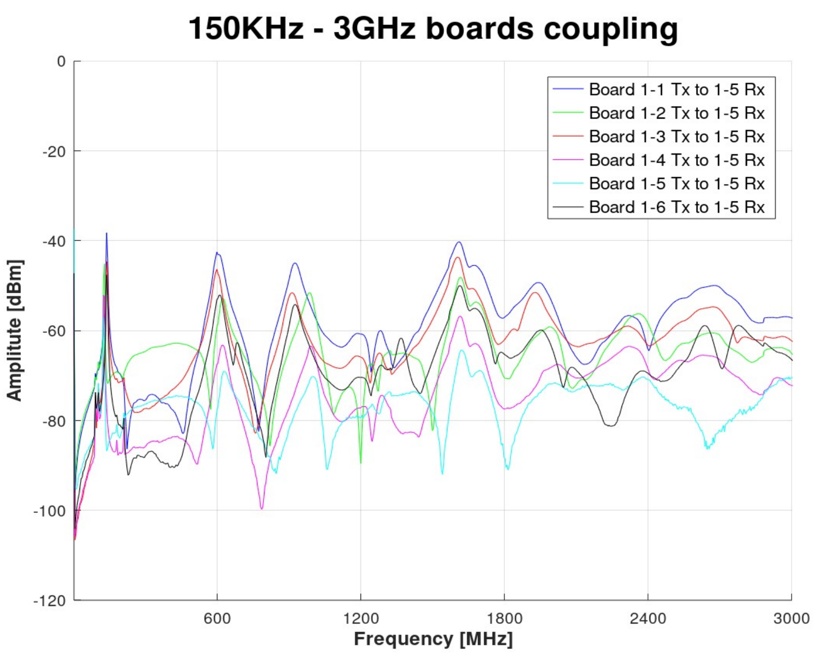

Figure 3: *PCB coupling due to radiated field (BW: 150KHz - 3GHz).*

The other PCBs layout with a ground on the edge of the traces are ID: 1-3 (red line) and

ID: 1-6 (black line). Both layouts are among the ones that have a high radiated field, thus the noise couple pretty well with the PCB ID: 1-5. Those layouts are almost cancelling out the benefit of having a ground layer. This scenario represents the typical use case where the designer, due to space constraints, may place the traces and the ground underneath, close to the edge of the PCB. This must be in general avoided, especially if the trace carries high bandwidth signals. Almost surprisingly the board ID: 1-2, while having a good layout with the ground layer properly covering the trace, does radiate at a certain frequency as worst as ID: 1-3 and ID: 1-6. The reason behind it, is due to the distance of the ground layer (bottom) and the top trace. Indeed the 1.6mm thickness used on standard PCBs causes high stray inductance out of the signal trace, that may generate high radiated field. This is the reason why for better performances, it is typically advisable to have 4 layers layout and not 2 layers as used in this test setup. With 4 layers layout it would be possible to place the GND layer directly underneath the signal layer at a reduced distance, compared to two layers layout. This would reduce the stray inductance that may cause radiated electromagnetic field.

A good example on how reduced GND distance between the signal trace and GND layer may reduce the radiated electromagnetic field, is the layout ID: 1-4. In this case there is only one layer and the GND is on the top layer only. Nevertheless the 50Ω Coplanar Microstrip allows reduced distance between GND and the signal trace, thus the electric field easily closes to GND, while the magnetic field has reduced intensity since the signal trace return current is close by.

Test setup 2

The measures made on the **Test Setup 1**, while showing how different boards behave from the radiated and coupling perspective, does not directly correlate the results to a certain standard. A designer may just say: "-80dBm coupling is a small value and I can ignore it". While those signals coupling are not that big, there are several standards that care about it in a certain bandwidth of interest. Thus you may not neglect some noise without a test supporting the assumption that the noise is small enough. On the Test Setup 2, it has been taken a DC-DC converter evaluation board tested in light mode control mode, for the CISPR 25 Class 5 conducted test. The evaluation board was passing the test with 10dB margin, thus it got a nice PASS. Repeating the test with the board ID: 1-1 on top of the evaluation board at 8mm distance, fed with 2.1MHz sinus signal at -10dBm amplitude, showed different results.

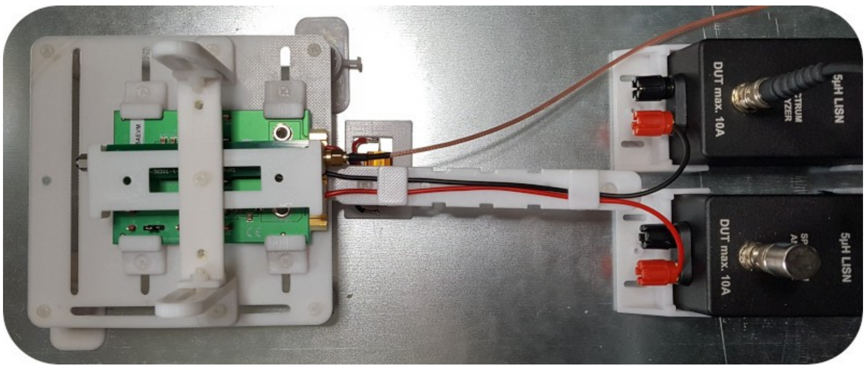

Figure 4: *PCB test setup with ID: 1-1 on top of the DC-DC converter evaluation board.*

Figure 5 shows how the 2.1MHz sinus that fed the PCB ID: 1-1 radiates enough noise that gets coupled with the evaluation board and clearly pops up at 2.1MHz, getting close to the margin (the BW between 30MHz and 108MHz, is not shown).

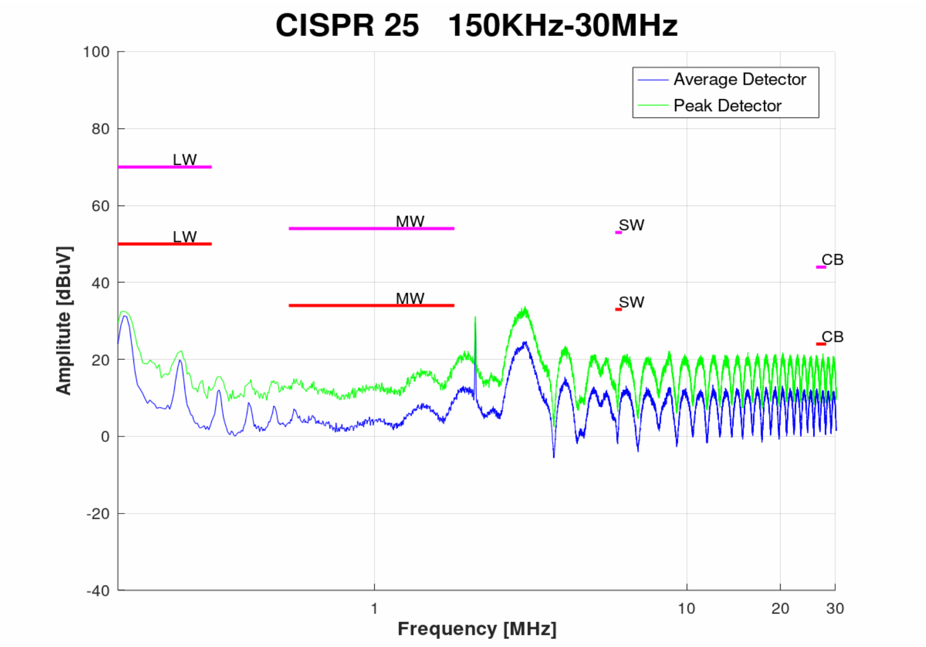

Figure 5: *PCB coupling due to radiated field - board ID: 1-1 (BW: 150KHz - 30MHz).*

Having chosen a frequency between the marked limits MW and SW, it is a typical use case for an automotive DC-DC converter switching frequency. Indeed this frequency range does not have a real limit defined by CISPR 25, nevertheless it clearly shows that the board ID: 1-1 can jeopardize the system certification. The same test run with the board ID: 1-4, designed with a good layout, does not show any noise peak (merely visible) since the energy that gets coupled is covered by the DC-DC converter noise (Figure 6).

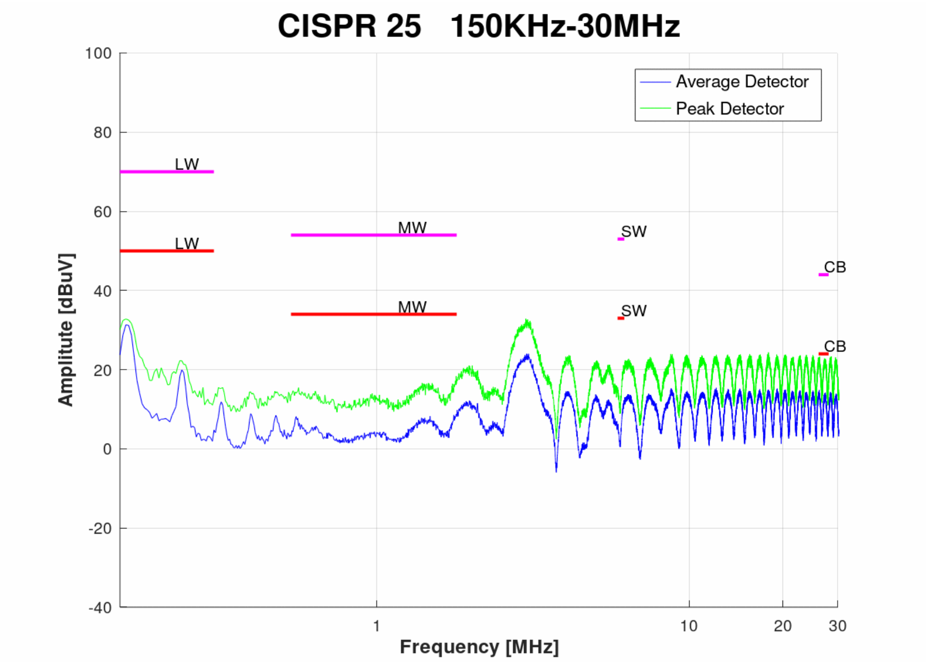

Figure 6: *PCB coupling due to radiated field - board ID: 1-4 (BW: 150KHz - 30MHz).*

Conclusions

Different PCB layouts have been tested to validate typical use cases that can be implemented on the PCB. In particular the traces that are very close to the PCB edge and are partially covered by GND, may radiate enough energy that get coupled by other parts of the PCB. This may create problems on sensitive adjacent circuits but may also create challenges passing EMC standards, either conductive or radiated tests. Proper layout is a key part of the design, thus the PCB must be designed with care, keeping both the system and EMC standards in mind.

About PCBWay

As one of the most experienced PCB manufacturers for prototyping and low-volume production in China, PCBWay is committed to meet the needs of customers from different industries in terms of quality, delivery, cost-effectiveness. With years of accumulated industry experience, PCBWay has customers from all over the world. The brand has became the first choice for the clients thanks to its high strength and special services, such as:

1. PCB prototyping and manufacturing FR-4 and Aluminum boards, but also advanced PCB like Rogers, HDI, Flexible and Rigid-Flex boards.
2. PCB assembly
3. Layout and design

Bibliography

[1] LaurTec.it: official site where you can download the “EMC Testing” series.
[2] PCBWay: official site for the PCB manufacturer.

History

Date	Version	Author	Revision	Description
27. May. 2020	1.0	Mauro Laurenti	Mauro Laurenti	Original version.
15. Nov. 2020	1.1	Mauro Laurenti	Mauro Laurenti	Second publication