Influence of the PCB Dielectric Material on the Coupling of PCB Traces to Enclosure Cavities

Christian Poschalko
Diesel & Gasoline Systems
Robert Bosch AG
Vienna, Austria
christian.poschalko@at.bosch.com

Siegfried Selberherr
Institute for Microelectronics
Technische Universität Wien
Vienna, Austria
Selberherr@TUWien.ac.at

***Abstract*—PCB structures like traces couple to the cavity field inside of an enclosure. It has been presented before that the coupling of a trace inside of a homogeneous cavity is caused just by its vertical segments on the trace ends. For an inhomogeneous cavity, consisting of a FR4 PCB layer and an air layer, we show that a trace couples along its whole length. The different coupling must be considered in the PCB design, for instance, for trace shielding. We present examples for a PCB trace coupling to a cubical enclosure, utilizing an efficient cavity model.**

Cavity mode; coupling; EMI; enclosure fields; shielding;

I. INTRODUCTION

The coupling from components and layout structures on a PCB to an enclosure cavity is relevant for the electromagnetic interference of a device, because the cavity field is coupled to the external environment by the enclosure apertures and by interface cables [1], [2]. An efficient cavity model has been utilized for power planes and for slim enclosure cavities [3], [4], [5]. PCB traces inside of the cavity have been considered by introduction of ports to the parallel plane cavity model at both end positions of the trace. The coupling factor between the trace currents and the currents at the parallel plane ports were calculated by [6] and by the geometric coupling factor d/h [7], where d is the vertical distance of the trace to the bottom plane of the cavity and h is the vertical distance from the bottom plane to the top plane of the parallel plane cavity. The currents on the trace are calculated by application of transmission line theory. This model is sufficient for parallel plane cavities with electrically small plane separation. It is efficient for many applications, such as power ground plane cavities on PCBs and for slim metallic enclosures, like those of automotive control devices or mobile devices. However, the vertical PCB structures are not just relevant for the coupling of the PCB to a parallel plane cavity. The relevance of the vertical PCB currents for the coupling to its electromagnetic environment is general. A direct relation between the coupling of PCB traces to parallel plane cavities and the common mode coupling of these traces to cables, which are connected to the PCB without an enclosure, was presented by [8]. Therefore, the investigation of the coupling from PCB structures can generally be performed, by application of the parallel plane cavity model. [6] and [7] calculated the coupling just from the vertical segments on the trace ends inside a homogenous cavity. However, a real PCB consists of a dielectric material with a dielectric constant, which is significantly higher, than that of an air layer between the PCB and a metallic cover plane of an enclosure. Thus, the cavity consists of two different layers and we show in Section II that a trace on the PCB inside such an inhomogeneous cavity couples not just at its trace end, but along its whole length. We describe the different coupling and show, how to introduce the traces in the parallel plane cavity model. The coupling has to be considered in the PCB design. In Section III we present consequences of the coupling on PCB trace routing and shielding, and a conclusion is given in Section IV.

II. COUPLING OF PCB TRACES TO INHOMOGENEOUS PARALLEL PLANE CAVITIES

A. Models for the evaluation of the coupling

To evaluate just the coupling from a trace to a parallel plane cavity, we compare the results from two three-dimensional full wave simulations with HFSS® from Ansys®. Both models consist of a slim enclosure with three metallic walls and a slot on one edge. The length of the slot L=134 mm, the width of the enclosure W= 104 mm and the separation of the top to the bottom plane h= 7 mm. The HFSS® model in Fig. 1 depicts the enclosure with transparent cover. It contains a trace above the bottom plane, with two ports connected between the trace and the bottom plane. For the calculation, we drive one port with a current of Is=1 A, and we terminate the second port with 50 Ω. The bottom plane of the cavity is also the ground plane of the PCB.

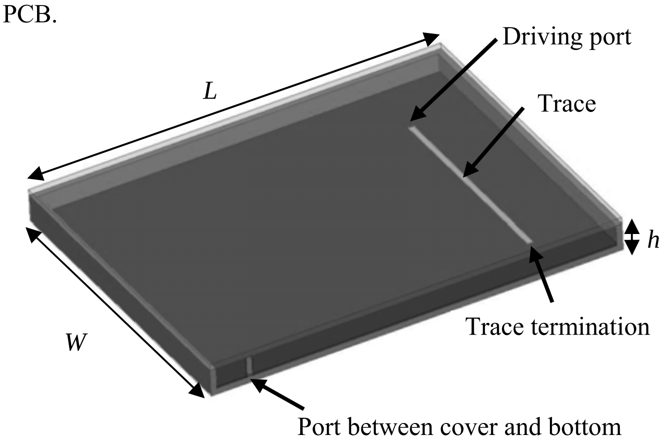

Figure 1. Enclosure cavity model containing a trace

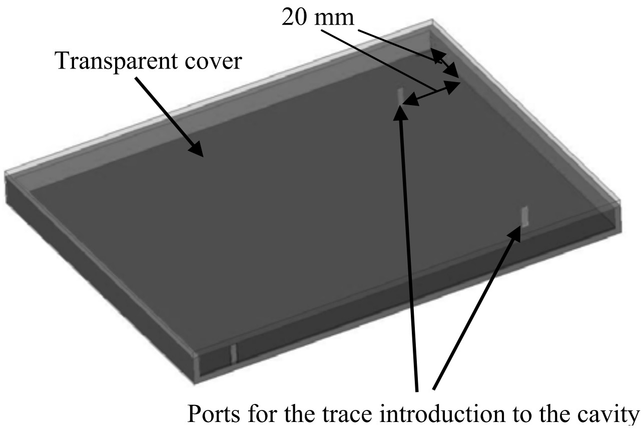

Figure 2. Enclosure cavity with ports between the cover and the ground plane, instead of a trace

The port at the slot is defined between the cover and the ground plane to enable the calculation of the coupling from the source current *Is* to the parallel plane voltage. The HFSS® calculation was carried out applying an air box with a radiation boundary condition surrounding the model in Fig. 1. The trace height above the bottom plane is *d*=0.65 mm. A second HFSS® model, depicted in Fig. 2, contains two ports between the cover and the bottom planes instead of the trace. The ports are placed at the same positions on the planes as the two trace ports in Fig. 1. We simulate the voltage on the slot port, caused by the trace current *Is* in Fig. 1. Secondly, we introduce the currents on the trace from the first calculation to the ports of the second structure in Fig. 2, weighted with the coupling factor *d*/*h*. Finally we compare the results of the two calculations, which must be identical, when the trace coupling has been correctly considered by this method.

B. *Calculation of the trace to cavity coupling*

Fig. 3 shows the very good agreement of the results of a 70 mm long trace, if the dielectric material inside of the cavity is homogeneous. We have carried out the same comparison with an inhomogeneous cavity, by adding an epoxy FR4 layer with a dielectric constant ε_r=4.5 and a thickness of d=0.65 mm between the bottom plane and the trace. Fig. 4 shows in this case completely different results between the HFSS® model with the trace and that with the ports. Thus, the trace introduction to the cavity model cannot be done in the same way for an inhomogeneous dielectric cavity as for a homogeneous one.

A trace above an FR4 layer can be described by a trace above an air layer and by adding additional capacitors between the trace and the ground plane along the trace. These capacitors consider the higher dielectric constant of the FR4 material. The coupling of a trace above an air layer to the cavity is correctly considered by introduction of the vertical currents with the weighting factor *d/h* to the cavity model, as described above. Thus, the coupling of a trace above an air layer, with additional capacitors distributed along the trace length, can correctly be calculated by introducing of all the vertical currents on these capacitors to the cavity model. Therefore, the difference in the coupling can be explained by additional capacitive vertical currents, flowing from the trace to the bottom plane. Since, the material FR4 with the increased dielectric constant is added below the whole trace, some vertical current flows also along the whole trace. Therefore the trace inside the inhomogeneous cavity couples to the cavity along its whole length and not just by the vertical currents at the trace ends. To introduce the trace correctly, we calculate the cavity as if it would be homogeneous and consider the dielectric material of the PCB by adding additional capacitors along the trace. The distance between the additional capacitors must be kept small compared to the wavelength of the maximum simulated frequency.

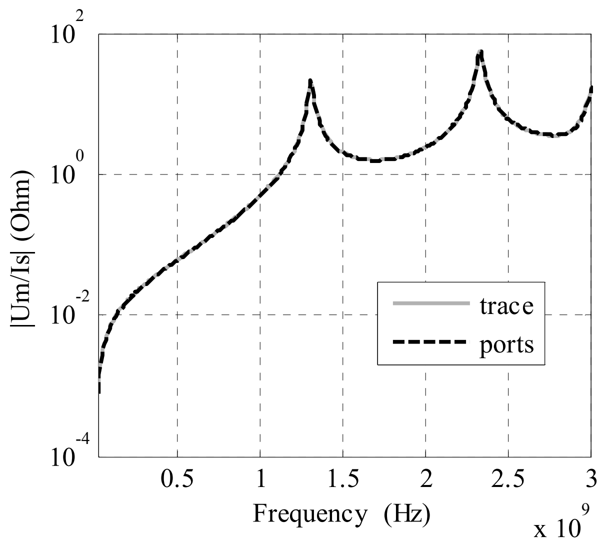

Figure 3. Transfer impedance from the current *Is* on the drive port of the trace to the voltage between the planes *Um*: Comparison between the results from the HFSS® model in Fig. 1 (grey line) and the results from the model in Fig. 2 (dotted line). The dielectric material inside the whole cavity is air.

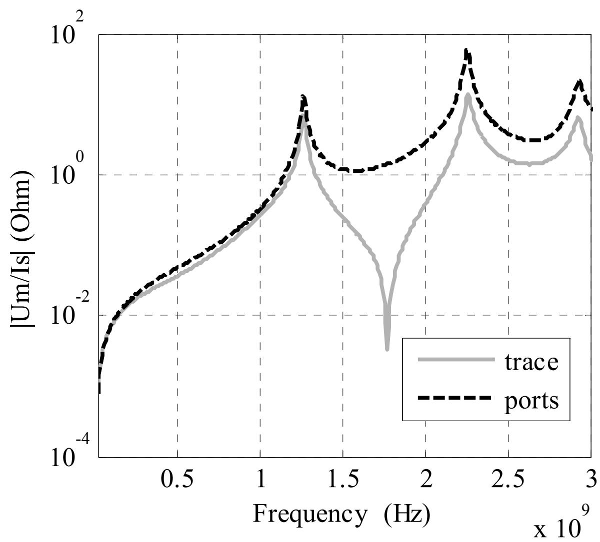

Figure 4. Transfer impedance from the current *Is* on the drive port of the trace to the voltage between the planes *Um*: Comparison between the results from the HFSS® model in Fig. 1 (grey line) and the results from the model in Fig. 2 (dotted line). The cavity volume consists of a FR4 layer below the trace and an air layer between the trace and the cover plane of the enclosure.

The result comparison from the models in Fig. 1 and Fig. 2, obtained with this trace introduction model show good agreement in Fig. 5. For this example, the 70 mm long trace has been replaced by a 20 mm long trace and the consideration of the FR4 material in the calculation has been carried out by connection of two capacitors to the trace ends. The value of the capacitors is

$$C_{diff} = \left(C'_{FR4} - C'_{ait}\right) \cdot l \cdot 0.5\,, \qquad (1)$$

where l is the length of the trace and C'_{FR4} and C'_{air} are the length capacitances of the trace above an FR4 and an air dielectric layer, respectively. Since we introduce two capacitors at both ends of the trace, the whole length difference of the trace capacitance is multiplied with 0.5 in (1). The short trace has been used to illustrate the method. For longer traces more capacitors along the trace must be added. In such a case just the two values of the capacitors at the trace ends have to be multiplied with 0.5. To obtain the currents at the source position, which has to be introduced to the cavity model, the current, flowing through the additional capacitor at the source must be subtracted from the source current, while at the load position, the load current is added to the current of the additional capacitor. This is consistent to the port definition in Fig. 2, where the port currents are flowing into the cover plane of the cavity. HFSS® models have been utilized, because they do not introduce any additional uncertainties to the result and thus enable to investigate the coupling from traces to planes most accurately. However, the trace introduction method together with the two-dimensional cavity model of [3] provides an efficient method for the EMC simulation of PCBs and enclosures [7]. Thus we are utilizing the cavity model of [7] instead of the HFSS® models in Subsection C.

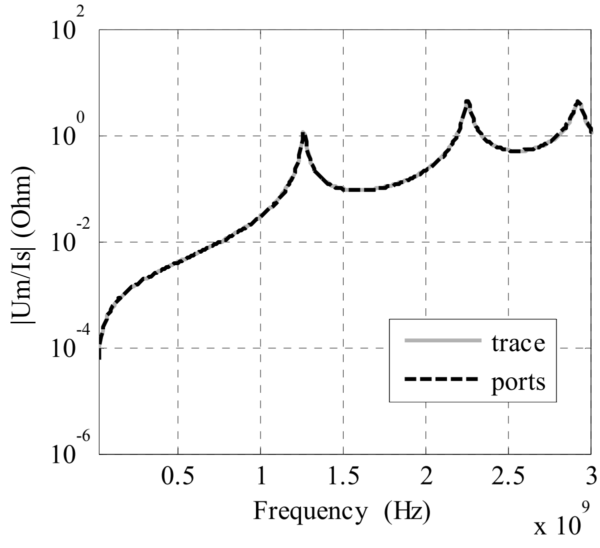

Figure 5. Transfer impedance from the current *Is* on the drive port of the trace to the voltage between the planes *Um*: Comparison between the results from the HFSS® model in Fig. 1 (grey line) and the results from the model in Fig. 2 (dotted line). The cavity volume consists of a FR4 layer below the trace and an air layer between the trace and the cover plane of the enclosure. FR4 was considered by capacitors at both ends of the 20mm long trace.

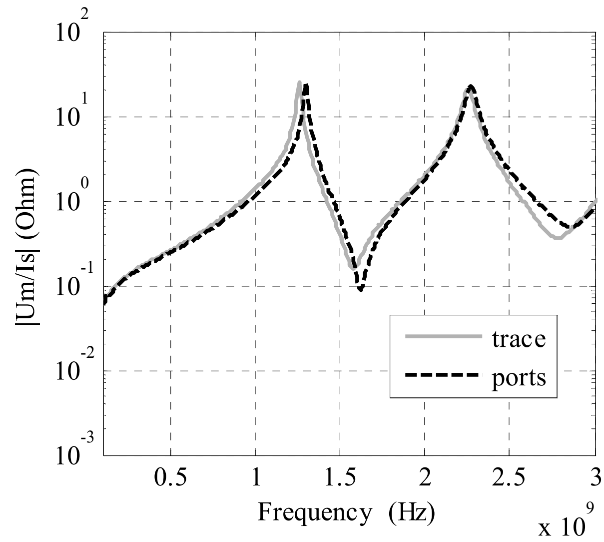

Figure 6. Transfer Transfer impedance from the current *Is* on the drive port of the trace to the voltage between the planes *Um*: Comparison between the results from the HFSS® model in Fig. 1 (grey line) and the results obtained with the cavity model of [7] for a trace with 70 mm length. As a difference to the previous examples the port between the planes is in the middle of the slot.

C. Trace Introduction to the Cavity Model

Fig. 6 depicts a comparison between the results of a HFSS® model containing a trace (Fig. 1), and the results from the cavity model of [7], where the trace was introduced as described before. The radiation loss at the slot of the cavity was considered in this model, by applying the method of [9]. The dielectric layer below the trace was FR4 and the layer between the trace and the cover was air. For the coupling simulation, the trace was calculated in air with additional capacitors every 10 mm along the trace. The currents on these capacitors, the source, and the load have been introduced to the cavity model. Fig. 6 shows good agreement between the results of the full wave three-dimensional simulation and the analytical cavity model.

III. Consequence for PCB Design, Regarding Trace Routing and Shielding

A trace on a PCB substrate material with a higher dielectric constant than one, such as FR4, couples to its air environment along the whole trace length. This is the case, when the PCB is inside a metallic enclosure cavity, but also if it is located in any other environment, which has a different dielectric constant as the PCB substrate. This coupling has to be considered in the PCB design. A trace within a homogeneous cavity couples just at its ends and thus different trace routing on the PCB does not change the coupling. Inside of an inhomogeneous cavity, or more generally, on each PCB, which has a substrate with a different dielectric constant than the surrounding environment, the trace routing changes also the coupling. Thus, a critical trace on a PCB inside of a metallic enclosure cavity should be routed, considering the coupling inside of the enclosure. For a slim enclosure this can efficiently be performed by application of the two-dimensional cavity model, as described in Section II. Another design consequence regards PCB trace shielding, which can be applied to reduce the coupling, since the coupling from the vertical signal currents is partially compensated by the coupling from the currents on the shields. However, it must be considered, that the PCB traces couple along their whole length. For traces in an air environment the effect of shielding can be achieved efficiently, just by connecting the shield traces to the ground plane at their ends. For a PCB consisting of a material such as FR4, the shield must be connected to the PCB ground plane along the whole length, which means that the distance between two ground connection vias on the shields should be electrically small.

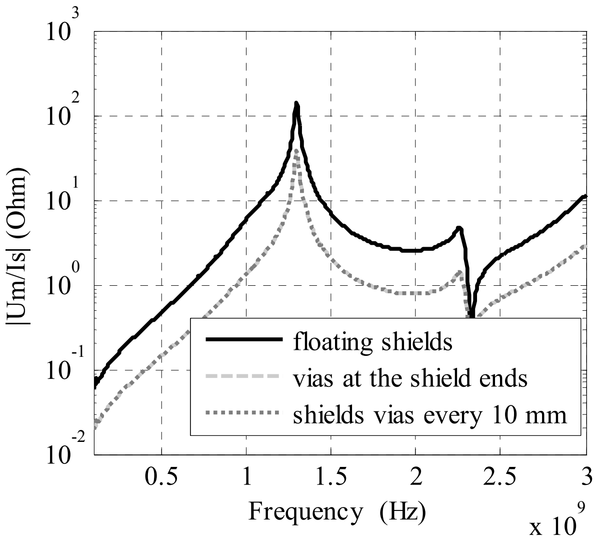

Figure 7. Transfer impedance from the current *Is* on the drive port of the trace to the voltage between the planes *Um*. The comparison is performed for a homogenous air cavity

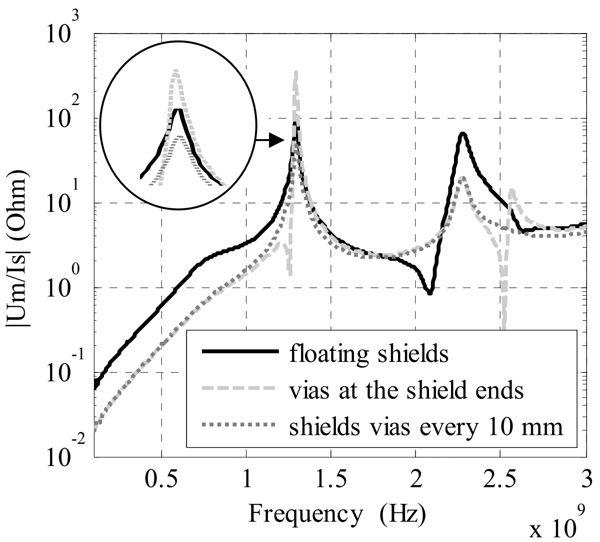

Figure 8. Transfer impedance from the current *Is* on the drive port of the trace to the voltage between the planes *Um*. The comparison is performed for an inhomogenous cavity, consisting of a PCB layer with FR4 substrate below the trace and an air layer between PCB and metallic enclosure cover.

For a shielded trace with a length of l=70 mm, a width of 0.2mm, shield width of 0.2 mm, and trace to shield distance of 0.2mm we compare three different cases of shield to ground connection. In the first case, the shields have no connection to ground, in the second case, the shields are connected to ground just at their ends, and in the last case the shields are connected to ground every 10 mm along their length. Fig. 7 and Fig. 8 depict this comparison for a homogenous air environment, and for an inhomogeneous cavity with an FR4 PCB material, respectively. For the inhomogeneous cavity, Fig. 8, shows, that broad band coupling reduction can only be achieved, by connection of the shields to the ground plane with multiple vias along the whole shield length.

IV. Conclusion

A trace on a PCB, which has a substrate material with a different dielectric constant than the environment of the PCB, couples to the electromagnetic field surrounding the PCB along the whole trace length. We have presented examples for this coupling mechanism for a PCB trace inside an enclosure cavity and we explained the difference to the coupling mechanism in homogenous cavities. Based on the results of the coupling investigation, we presented design consequences for PCB trace routing and shielding.

References

[1] K. Haixin, K. Morishita, T. Hubing, N. Kobayashi, and T. Harada, “Modeling Radiated Emissions Due to Power Bus Noise from Circuit Boards with Attached Cables,” IEEE Trans. on Electromagnetic Compatibility, vol. 51, no. 2, pp. 412 – 416, May 2009.

[2] C. Poschalko, S. Selberherr, “Calculation of the Radiation from the Slot of a Slim Enclosure with a Cavity Resonator Model.” In Proc. of the Asia-Pacific Symp. on Electromagnetic Compatibility, pp. 622-625, May 2008.

[3] L. Ren, Z. Yu, G. Feng, F. de Paulis, Y. Zhang, J. Fan, X. Dong, J. A. Mix, D. Hua, K. Slattery, „Aperture Modeling Using a Hybrid Method for RFI Analysis“, In Proc. of the IEEE Symp. on Electromagnetic Compatibility, pp.1-4, August 2008.

[4] G.-T. Lei, R.W. Techentin, and B. K. Gilbert, “High-Frequency Characterization of Power/Ground-Plane Structures,” IEEE Trans. Microw. Theory Tech., vol. 47, no. 5, pp. 562–569, May 1999.

[5] C. Wang, J. Mao, G. Selli, S. Luan, L. Zhang, J. Fan, D. J. Pommerenke, R. E. DuBroff, and J. L. Drewniak, “An Efficient Approach for Power Delivery Network Design with Closed-Form Expressions for Parasitic Interconnect Inductances”, IEEE Trans. Adv. Packag., vol. 29, no. 2, pp. 320–334, May 2006.

[6] J. E. Bracken, S. Poltyanko, S. Raman, and Z. J. Cendes, “Efficient Full Wave Simulation of High-Speed Printed Circuit Boards and Electronic Packages”, In Proc. of the Int. Symp. IEEE Antennas Propag. Soc., vol. 3, pp. 3301–3304, June 2006.

[7] C. Poschalko and S. Selberherr, “Cavity Model for the Slot Radiation of an Enclosure Excited by Printed Circuit Board Traces with Different Loads”, in IEEE Trans. on Electromagnetic Compatibility, vol. 51, no.1, pp. 18–24, Feb. 2009.

[8] C. Poschalko and S. Selberherr, „Relation Between the PCB Near Field and the Common Mode Coupling from the PCB to Cables“, In Proc. of the Asia-Pacific Symp. on Electromagnetic Compatibility, pp. 1102-1105, April 2010.

[9] C. Poschalko and S. Selberherr, „Domain Separation with Port Interfaces for Calculation of Emissions from Enclosure Slots.”, In Proc. of the IEEE Symp. on Electromagnetic Compatibility, pp. 1-6, August 2008