BALANCED TLM AND COUPLING MODEL FOR ANALYSIS OF POWER/GROUND RESONANCE, NOISE COUPLING, AND EDGE RADIATION

Jun So Pak, Jongbae Park, Hyungsoo Kim, Heejae Lee*, Cheol-seung Choi*, and Joungho Kim

Terahertz Interconnection and Package Laboratory, EECS Dept. KAIST,
Samsung Electronics*,
E-mail: joungho@ee.kaist.ac.kr

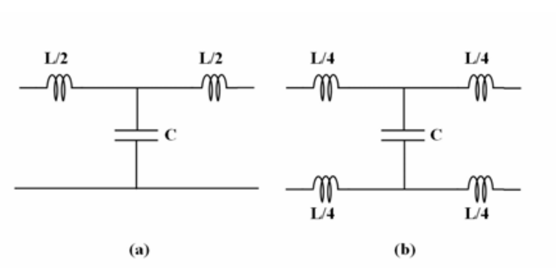
Abstract: In this paper, balanced TLM and via coupling model are proposed for an efficient simulation of SSN, noise coupling, radiated field emission problem from power/ground plane structure in multi-layer package and PCB. It is demonstrated that the simulation agrees fairly well with the measurement, which confirms preciseness and usefulness of the proposed model.

Key words: TLM, SSN, Noise coupling, Edge radiation, Plane cavity.

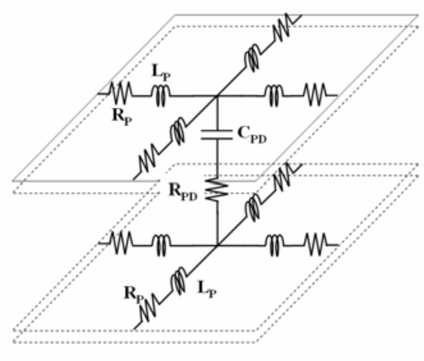
2. Balanced TLM and Via coupling Model

The TLM is a field solving method used in various waveguides by using well known relationship between electromagnetic field quantities, and voltage and current on transmission lines. If transverse magnetic (TM) field to z-axis is considered in planar structures like multi-layer interconnections in package and PCB, the TLM become an efficient two-dimensional analysis method, which is the conventional TLM. The conventional TLM uses transmission line model as shown in figure 1(a). In this paper, we used the balanced TLM to analyze the electromagnetic field distribution inside the power/ground plane cavity of the multi-layer PCB. The schematic of the balanced TLM mesh is illustrated in figure 1(b). The ground line inductance per unit length represents the inductor model at the return current path.

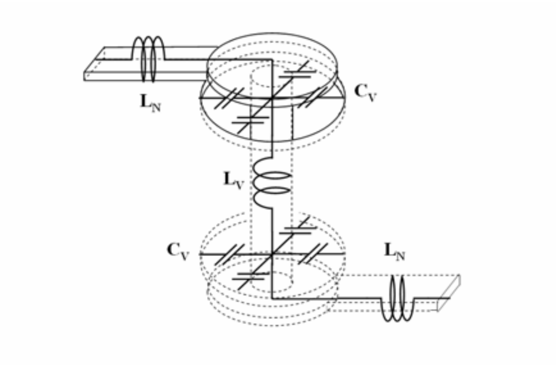
Especially, when the signal trace is switching the reference planes though the signal via in the multi-layer stack, the ground return current is broken at the via, and the ground impedance becomes extremely high at the resonance frequencies of the power/ground plane cavity. Figure 2 and 3 shows the circuit schematic of the proposed balanced TLM and via coupling model employed to the analysis of four layer PCB including a though-hole signal via. The proposed balanced TLM model can be easily extended to multi-layer stack structure of more than four layers. In order to describe the return current path break at the signal via, where the signal trace exchanges the reference planes, and in order to model the excitation of the power/ground plane cavity resonance, the via model is included as a form of a coupling model to the plane cavity. The capacitive coupling is expressed as coupling capacitors connected between the via metal column and the edge of the planes. The via column is described as an inductor and the via neck effect is modeled as a series inductor to the signal line. By using the model, we were successfully able to induce s-parameters, plane impedance, and edge radiations.



[Figure 1] (a) Conventional TLM scheme, (b) Balanced TLM scheme.



[Figure 2] Balanced TLM for power/ground plane cavity.



[Figure 3] Via coupling model.

3. Reflection Loss and Insertion Loss by Resonance

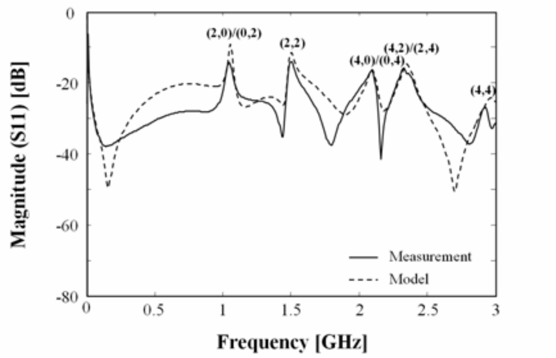
With the balanced TLM unit cells and via coupling model, we have composed the models of the test boards. Then the model parameters were obtained by fitting the simulated S-parameters to the measured S-parameters of the test boards.

The simulated and the measured S-parameters are shown in figure 4 and 5 to verify the modeling and model parameters. S11 and S21 is graphed in figure 4 and 5. Both simulation and measurement confirm fair agreement of the resonance frequencies and associated modes depending on the via positions. The numbers in round brackets inside the graph are the mode numbers of the resonances at the power/ground plane cavity. The resonance frequencies are selected at high impedance peaks in figure 4 and 5, and can be calculated with following equations,

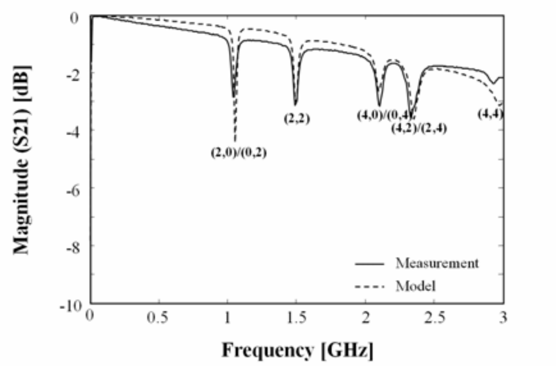
$$(f_r)_{mn} = \frac{1}{2\pi\sqrt{\mu\varepsilon_r\varepsilon_0}}\sqrt{\left(\frac{m\pi}{a}\right)^2 + \left(\frac{n\pi}{b}\right)^2}$$

where f_r is the resonance frequency related to mode number (m, n), and *a* (14cm) and *b* (14cm) are width and length of the power/ground plane cavity. The resonance frequencies are independent to dielectric thickness.

All the resonances of the 'odd' mode numbers are considerably suppressed by positioning the via at the center of the PCB. It is because of the mode properties and the open edge boundary condition of the rectangular PCB shape. As results, the high impedance produced by the power/ground cavity resonance imposes the high impedance at the return current path discontinuity. Then, the total input impedance at the feeding point of the microstrip line becomes extremely high at the 'even' numbered mode resonances. Consequently, S11 magnitude increases and S21 magnitude decreases at the 'even' numbered mode resonances, that is, the return loss increases and the insertion loss increases, respectively. The excited cavity resonance is eventually propagated to the edge of the PCB, resulting in the edge radiated emission problem from the PCB edge.



[Figure 4] Calculated and measured return loss of the signal line by the power/ground plane cavity resonance.



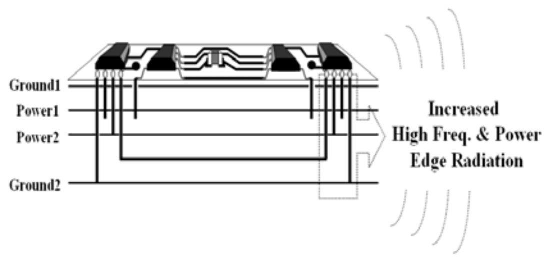
[Figure 5] Calculated and measured insertion loss of the signal line by the power/ground plane cavity resonance.

4. SSN Noise Coupling and Edge Radiation

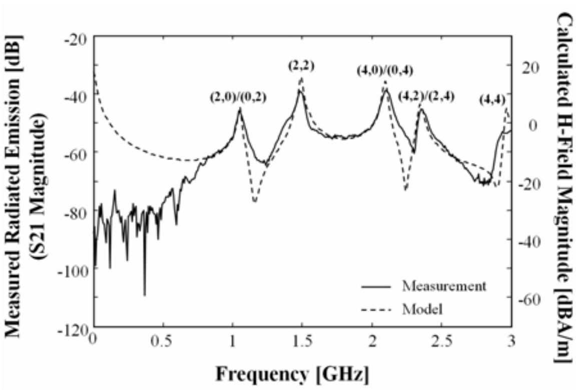
Voltage fluctuation is induced across the power/ground cavity by the signal return current of the through-hole signal via. Then the power/ground voltage fluctuation is generated and propagated toward to the edge of the power/ground plane cavity, producing standing waves in the cavity at the resonance frequencies. Consequently, the standing electromagnetic waves are distributed with the same formation as the impedance curve. Consequently it produces maximum magnetic current at the open edges of the power/ground plane cavity. As a result, the magnetic current at the open ended power/ground plane cavity edge becomes the source of the radiated field emission (Figure 6).

Figure 7 shows the calculated and measured radiated field emission from a test board. The proposed model well predicts the radiated field emission from the power/ground plane cavity edge excited by the though-hole signal via. The numbers in round brackets appear at the same frequencies at insertion loss measurement and radiated field emission measurement. This means that the radiated field emission is strongly correlated with the increased insertion loss of the microstrip line. The radiated field emission is maximized at the resonance of the power/ground plane cavity, where the insertion loss of the microstrip line including the through-hole signal via is also maximized.

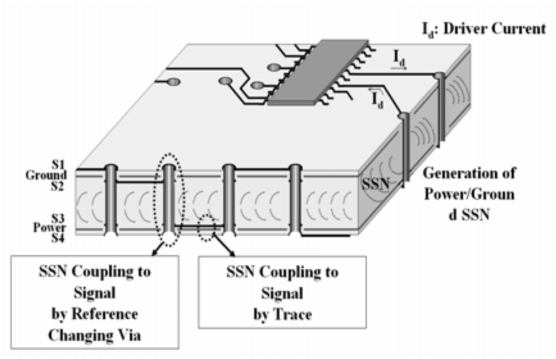
Finally, we have simulated and measured the SSN coupling to signal line and via. It is also found that the SSN is strongly coupling to signal or clock lines when the resonances are build up at the cavity resonance frequencies. Figure 8 and Figure 9 show SSN noise coupling to line and via, and SSN noise coupling mechanism. Figure 10 shows the frequency domain coupling noise, and figure 11 shows the time domain coupled clock waveform.



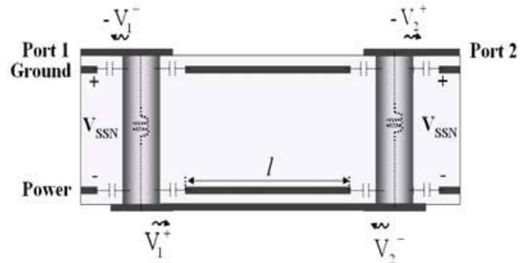
[Figure 6] Increased radiated field emission from the power/ground plane cavity edge, due to the increased routing density of PCB and consequently number of via and embedded line.



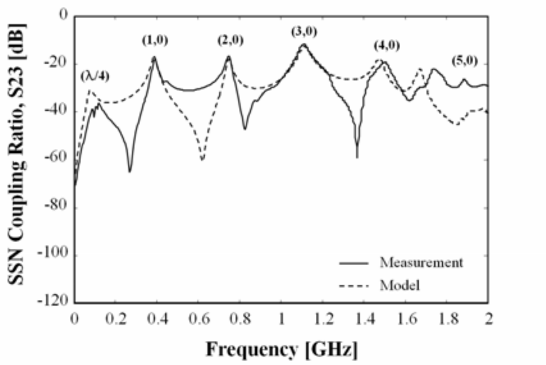
[Figure 7] Calculated and measured radiated field emission from the power/ground plane cavity edge at the resonance frequencies.



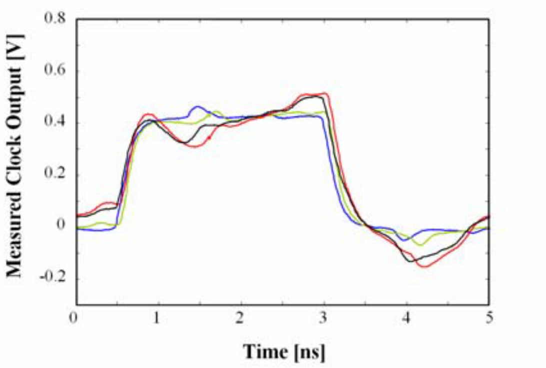
[Figure 8] SSN coupling to line and via



[Figure 9] SSN noise coupling mechanism.



[Figure 10] Calculated and measured SSN noise coupling to signal line from the power/ground plane cavity at the resonance frequencies.



[Figure 11] Time domain simulated waveform of the clock signal, which suffers huge SSN coupling.

5. Conclusion

Especially, when the signal trace is switching the reference planes though the signal via in the multi-layer stack, the ground return current is broken at the via, and the ground impedance becomes extremely high at the resonance frequencies of the power/ground plane cavity. It is demonstrated that the simulation agrees fairly well with the measurement, which confirms preciseness and usefulness of the proposed model. It is also shown that the through-hole signal via is a considerable source of the signal loss and SSN coupling as well as the radiated field emission.

References

[1] Jonghoon Kim, Hyungsoo Kim, and Joungho Kim, " Efficient on-chip decoupling capacitor design on an 8-bit microcontroller to reduce simultaneous switching noise and electromagnetic radiated emission," in *IEICE Trans. on Communications,* vol. E86-B, No. 6, pp. 2077-2080, Jun. 2003.

[2] S. Radu and D. Hockanson, "An investigation of PCB radiated emissions from simultaneous switching noise," in *Proc. IEEE Int. Symp. Electromagnetic Compatibility*, Seattle, WA, Aug. 1999, pp. 893-898.

[3] M. Leone, "The radiation of a rectangular power-bus structure at multiple cavity-mode resonances," in *IEEE Trans. Electromagnetic Compatibility*, vol. 45, pp. 486-492, Aug. 2003.

[4] I. Novak, "Reducing simultaneous switching noise and EMI on ground/power planes by dissipative edge termination," in *IEEE Trans. Advanced Packaging*, vol. 22, No. 3, pp. 274-283, Aug. 1999.

[5] X. Yee, D. M. Hockanson, M. Li, Y. Ren, W. Cui, J. L. Drewniak, and R. E. DuBroff, "EMI mitigation with multilayer power-bus stacks and via stitching of reference planes," in *IEEE Trans. Electromagnetic Compatibility*, vol. 43, pp. 538-548, Nov. 2001.