Crosstalk Reduction in the Coupled Microstrip Lines with Vias

Avali Ghosh[1] Sisir Kumar Das[2] Annapurna Das[3]

[1,2,3]Department of Electronics & Communication Engineering

[1,2,3]Guru Nanak Institute of Technology 157/F Nilgunj Road, Sodepur, Kolkata-700114

Abstract— This paper describes the cross-talk analysis of high-speed PCB design. Some models using multiple PCB traces above a ground plane having guard traces and vias are taken and the results are obtained from modeling and simulation using Ansoft HFSS software tool and are compared with the experimental results. Reduction techniques of crosstalk are highlighted.

Key words: Signal Integrity, Micro-Strip, Crosstalk, NEXT, FEXT

I. Introduction

Electromagnetic Compatibility (EMC) and Signal Integrity (SI) become very important issues for the modern electronic products of high-speed, high-frequency, small size and lower supply voltage. Since high speed circuits demand very low rise time of pulses, requirement of large frequency band-width of any system and circuits become essential, which in turn increases the risk of EMC due to possible electromagnetic emission from the systems and circuits. This emission interferes with other co-located systems /adjacent circuits. Crosstalk between PCB traces is a major factor for EMC and SI in printed circuit boards (PCBs). Cross talk occurs due to inductive and capacitive coupling between traces [1] – [3].

In this paper some models using multiple PCB traces above a ground plane having guard traces with vias and without vias, serpentine gaurd trace with vias and without vias are considered. The results obtained from modelling and simulation using Ansoft HFSS software are compared with the experimental results to validate the effectiveness of the via for crosstalk reduction. Reduction techniques of crosstalk problems are highlighted [4]-[6].

II. Crosstalk between Adjacent Microstrip Lines

The crosstalk performances of six microstrip configurations as shown in Fig. 1 are investigated in this section [4]-[8]. The analytical and experimental study reported here is original using Ansoft HFSS software tool. The first geometry shown in Fig. 1(a) composed of two traces, with spacing s, and equal widths w, placed on a lossless dielectric substrate of thickness h, dielectric constant ε_r and permeability μ_0. The ground plane below the substrate and the traces are assumed to be perfectly conducting. The second structure is the same microstrip line structure but contains a ground trace in between conducting traces, as shown in Fig. 1(b). The third case has the same specifications as the second case, except that the ground trace is having 2 vias, as shown in Fig. 1(c). The fourth case has the same specifications as the third case, except that the ground trace is having 12 vias, as shown in Fig. 1(d). The fifth case has the same specifications as the first case but contains a serpentine gaurd trace in between conducting traces, as shown in Fig. 1(e). The sixth case has the same specifications as the fifth case but contains vias, as shown in Fig. 1(f).

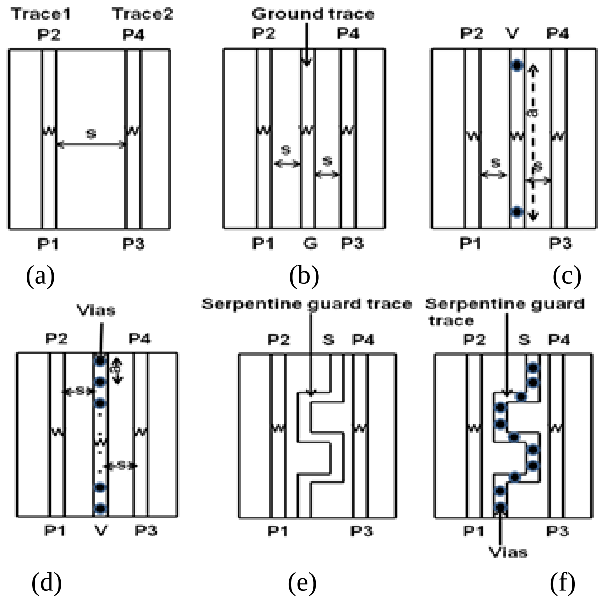

Fig. 1: Microstrip line geometry (a) Two conductor traces (b) Ground trace in between (c) Ground trace with 2 vias in between (d) Ground trace with 12 vias in between (e) Serpentine gaurd trace in between (f) Serpentine gaurd trace with vias in between

III. Modeling And Simulation Using Hffs

A. Two Conductor Model For Crosstalk

For all the cases, a FR4 material ($\varepsilon_r = 4.4$ and $tan\delta = 0.02$) is used as the substrate with the substrate thickness of 1.6mm. The width of the copper trace W = 3.1mm, which yields a characteristic impedance of 50Ω and the thickness of the copper trace t is assumed negligible i.e. t = 0.035mm, spacing between two conducting traces is S = 3W. In this design copper plane is on one side and three parallel traces are on the other side.

As shown in Fig.2(a), one of the trace was excited with RF source impedance of 50 Ohms and terminated with a 50 Ohms at the other end. Remaining traces are also terminated with 50 Ohms at both the ends. The coupling length of the 50 ohm traces are taken as λ/10 corresponding to highest frequency of operation 2 GHz for analog signals and pulse rise time of 1 ns for digital signals. Fig. 3 shows the result obtained by simulation and modelling using HFSS.

After fabrication process, measurement is done to collect required data by using Agilent PNA Series Network Analyser E8363B (10 MHz – 40 GHz). The coupling parameters S31 (Near-end) and S41 (Far-end) are measured.

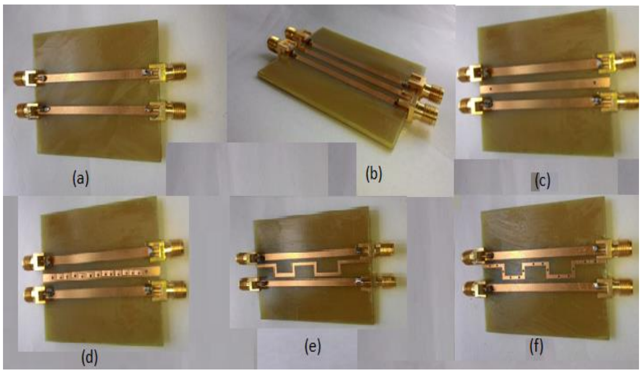

Fig. 2: Microstrip line geometry (a) Two conductor traces (b) Ground trace in between (c) Ground trace with 2 vias in between (d) Ground trace with 12 vias in between (e) Serpentine gaurd trace in between (f) Serpentine gaurd trace with vias in between

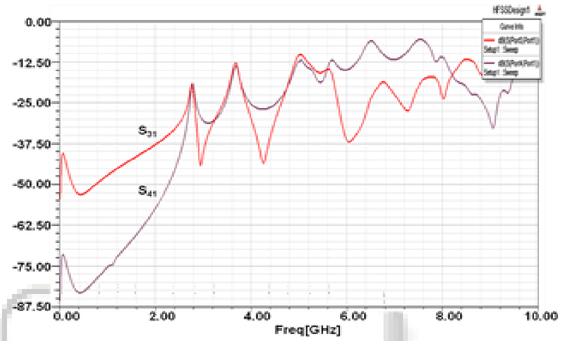

Fig. 3: NEXT and FEXT results obtained by using HFSS

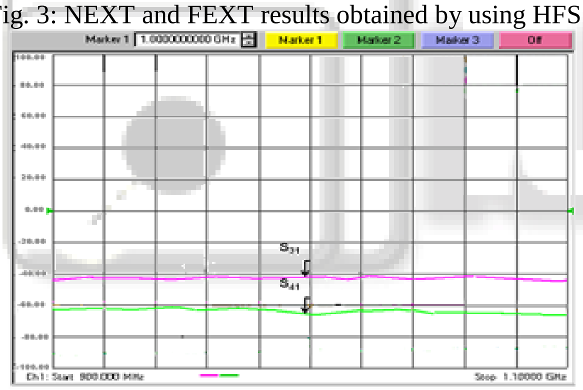

Fig. 4: Experimental results for NEXT and FEXT

B. Ground Trace Without Vias In Between To Reduce Crosstalk

Fig. 2(b) represents the coupled microstrip lines, which are terminated with 50 ohms at both the ends. A guard trace is in between the aggressor and victim lines to reduce the coupling between them. A grounded shield line is used to reduce the capacitive coupling of on-chip interconnects.

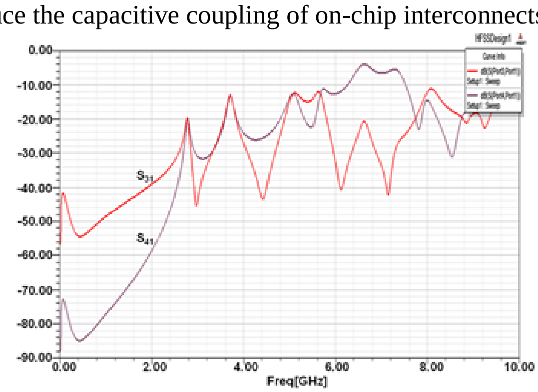

Fig. 5: NEXT and FEXT results obtained by using HFSS

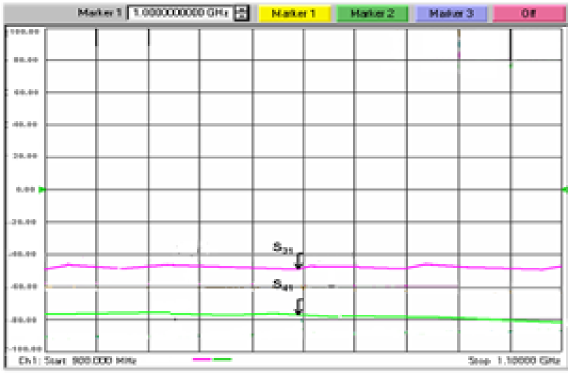

Fig. 6: Experimental results for NEXT and FEXT

From the results in Fig. 5 and 6, it is concluded that shielding of sensitive traces reduces cross-talk.

C. Ground Trace With 2 Vias In Between To Reduce Crosstalk

Fig. 2(c) represents the two microstrip lines, which are terminated at both ends. A guard trace with the 2 vias connections to ground plane is in between the aggressor and victim lines to reduce the coupling between them. The width of the gaurd trace is same as the width of the microstrip lines, vias radius R = 0.762mm and the spacing between vias a = 46mm.

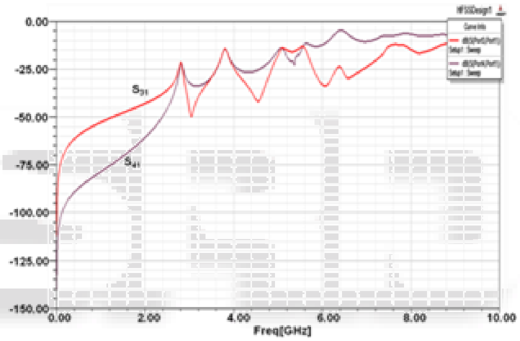

Fig. 7: NEXT and FEXT results obtained by using HFSS

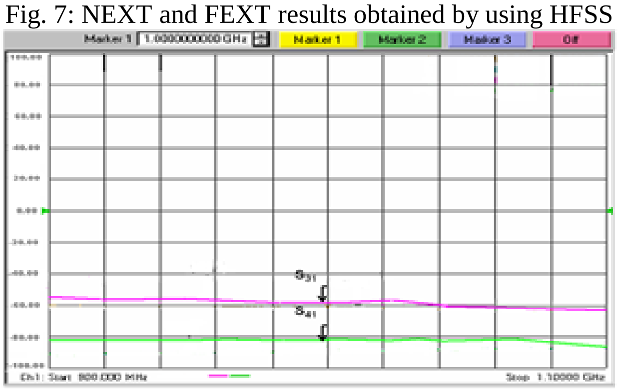

Fig. 8: Experimental results for NEXT and FEXT

From the results in Fig. 7 and 8, it is concluded that shielding of sensitive traces with the guard trace having via connections reduces more cross-talk.

D. Ground Trace With 12 Vias In Between To Reduce Crosstalk

Fig. 2(d) represents the two microstrip lines, which are terminated at both ends. A guard trace with the 12 vias connections to ground plane is in between the aggressor and victim lines to reduce the coupling between them. The width of the gaurd trace is same as the width of the microstrip lines, vias radius R = 0.762mm and the spacing between vias a = 4.18mm.

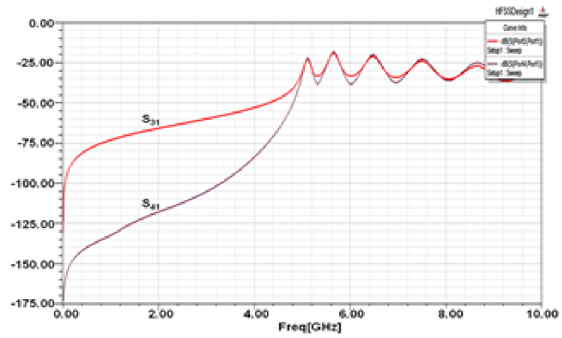

Fig. 9: NEXT and FEXT results obtained by using HFSS

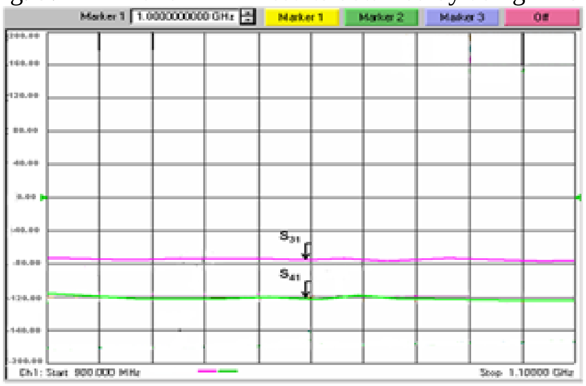

Fig. 10: Experimental results for NEXT and FEXT

From the results in Fig. 9 and 10, it is concluded that shielding of sensitive traces with the guard trace having more via connections reduces more cross-talk.

E. Serpentine Gaurd Trace In Between To Reduce Crosstalk

Fig. 2(e) represents the coupled microstrip lines, which are terminated with 50 ohms at both the ends. A serpentine guard trace is in between the conducting traces which reduces the coupling between them. The width of the serpentine gaurd trace is 1.5mm.

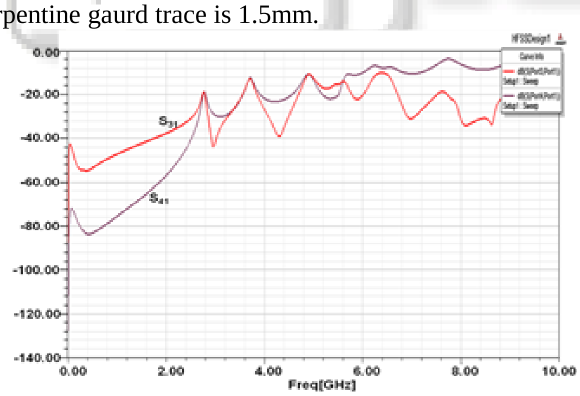

Fig. 11: NEXT and FEXT results obtained by using HFSS

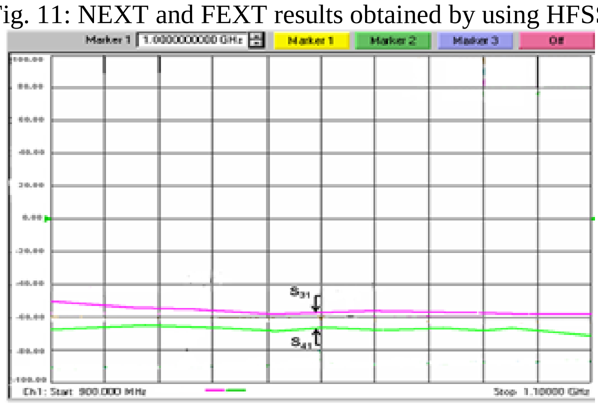

Fig. 12: Experimental results for NEXT and FEXT

From the results in Fig. 11 and 12, it is concluded that the serpentine guard trace reduces more cross-talk as compared to conventional guard trace.

F. Serpentine Guard Trace with Vias in Between To Reduce Crosstalk

Fig. 2(f) represents the coupled microstrip lines, which are terminated with 50 ohms at both the ends. A serpentine guard trace with vias in between the conducting traces to reduce the coupling between them. The serpentine trace is considered as the same as in the previous case and vias radius R = 0.762mm.

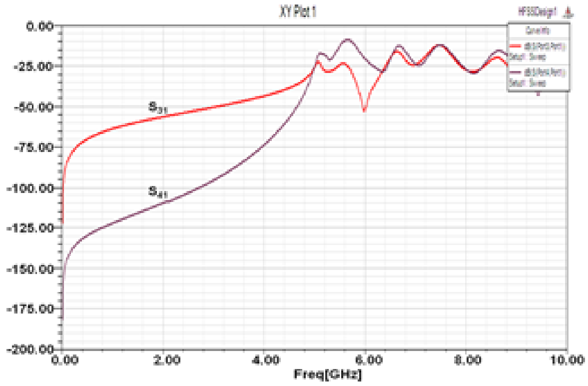

Fig. 13: NEXT and FEXT results obtained by using HFSS

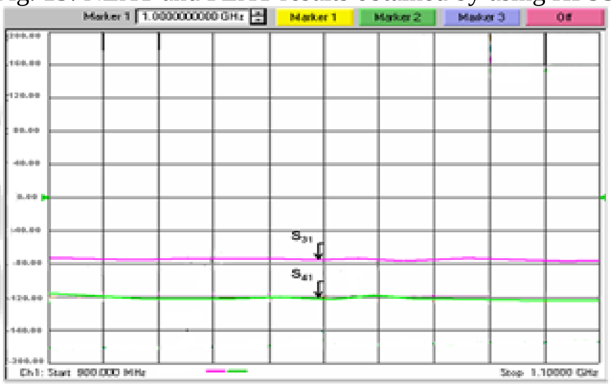

Fig. 14: Experimental results for NEXT and FEXT

From the results in Fig. 13 and 14, it is concluded that the serpentine guard trace with vias reduces more cross-talk as compared to serpentine guard trace without vias.

IV. Conclusions

The conventional ground trace is not effective to reduce the crosstalk of transmission lines on PCB. Because the guard trace cannot maintain the ground potential at all the positions along the trace because the guard trace itself is a transmission line.

The guard trace with the more via connections to ground plane is more effective than the conventional guard and guard trace with less number of via connections. Because the guard trace with more vias can maintain the ground potential at every via point and it reduces coupling as in the grounded shield line.

The serpentine guard trace reduces the far-end crosstalk as compared to the conventional guard. It has an advantage over the via guard, because the backside of PCB is available for routing while it is used up by vias in the via guard.

ACKNOWLEDGMENT

The work is supported by Guru Nanak Institute of Technology, R&D Cell and SAMEER - Kolkata.

REFERENCES

[1] Paul C. R. (1987), 'Introduction to Electromagnetic Compatibility', A Wiley- Interscience Publications, John Wiley & sons, Inc, Newyork
[2] Young-Soo Sohn, Jeong-Cheol Lee, Hong-June Park and Soo-In Cho, 'Empirical Equations on Electrical Parameters of Coupled Microstrip Lines for Crosstalk Estimation in Printed Circuit Board', IEEE Transactions on Advanced Packaging, VOL. 24, NO. 4, November 2001.
[3] Stephen H. Hall and Howard L. Heck, 'Advanced Signal Integrity for high speed digital designs', A John Wiley & Sons Inc Publications, Hoboken, New Jersey.
[4] Felix D. Mbairi, W. Peter Siebert and Hjalmar Hesselbom, 'High-Frequency Transmission Lines Crosstalk Reduction Using Spacing Rules', IEEE Transactions on Components and Packaging Technologies, VOL. 31, NO. 3, Sept 2008.
[5] Asanee Suntives, Arash Khajooeizadeh, Ramesh Abhari, 'Using Via Fences for Crosstalk Reduction in PCB Circuits', 1-4244-0293-X/06/$20.00 (c)2006 IEEE
[6] Kyoungho Lee, Hyun-Bae Lee, Hae-Kang Jung, Jae-Yoon Sim and Hong-June Park, 'Serpentine Guard Trace to Reduce Far-end Crosstalk and Even-Odd Mode Velocity Mismatch of Microstrip Lines by More than 40%' , IEEE Electronic Components and Technology Conference, 2007
[7] W.-T. Huang, C.-H. Lu, and D.-B. Lin, 'Design of Suppressing Crosstalk by Vias of Serpentine Guard Trace', PIERS ONLINE, VOL. 6, NO. 4, 2010
[8] Kyoungho Lee, Hyun-Bae Lee, Hae-Kang Jung, Jae-Yoon Sim and Hong-June Park, 'Serpentine Guard Trace to Reduce Far-end Crosstalk and Even-Odd Mode Velocity Mismatch of Microstrip Lines by More than 40%', 1-4244-0985-3/07/$25.00 ©2007 IEEE