Signal Integrity Analysis And Design of Signal Traces For Highspeed Pcbs

Dr.A.B.Gurulakshmi[1st], Dr.N.Sureshkumar[2nd], Dr.S.Pavalarajan[3rd]

Department of Electronics and Communication Engineering [1st, 2nd], *Department of Information Technology*[3rd]

New Horizon College of Engineering, Bangalore [1st], *Vellammal College of Engineering and Technology, Madurai*[2nd]
P.S.N.A. College of Engineering and Technology, Dindigul[3rd]
gurulakshmiab@gmail.com[1st], *nskece2002@yahoo.co.in*[2nd], *rajan@psnacet.edu.in*[3rd]

Abstract *- The rapid increase in the demand for high speed electronics devices, designers need to optimize the layer count and hence Printed Circuit Board (PCBs) trace width must be reduced to ensure good Signal Integrity (SI). This physical constrain limits the data rate of high speed PCBs signals, making it more difficult to meet the next generation product's design expectations. While designing high speed systems, high data rate is achieved by increasing the clock frequency to GHz which intern leads to doubting the quality of signal handled. Signal integrity is a major issue that requires great consideration from researchers and engineers alike to achieve good quality. In this paper, the importance of signal integrity in high speed system designing has been discussed by pointing out the major key factors affecting signal integrity in traces. Some of the major issues of concern for signal integrity are Crosstalk, Discontinuity, Overshoot, Propagation delay and so on. The main objective of this paper is to estimate and analyse the signal integrity issues in term of crosstalk estimation in high speed PCBs by designing and simulating different shield traces to be used between the signal traces using Advanced Design System (ADS) Version 2011.05. The simulated results are validated by fabrication and testing process. A novel shield traces Multi conductor Transmission lines (MTL) are proposed to meet the requirements of the modern high speed RF sectors.*

Keywords*: Crosstalk, discontinuity, overshoot, propagation delay, high speed PCBs, signal integrity, multi conductor transmission lines, shielded traces*

I. INTRODUCTION

SIGNAL INTEGRITY (SI) is the standard to measures of the quality of alternating signal handled by high speed PCBs. The attenuation associated with PCB transmission lines constitute a
great consideration in area of high-speed simulation/design and signal integrity. Some of the main issues of concern for signal integrity are ringing, discontinuity, crosstalk, distortion, signal loss, and power supply noise.

MULTICONDUCTOR TRANSMISSION LINES (MTL)

MTL is a specialized traces or other structure designed to conduct alternating current of radio frequency and higher, that is, currents with a frequency higher that their behaviour must be taken into account. Transmission lines are used for purposes such as connecting signal in radio transmitters and receivers with their antennas, distributing cable television signals, trunk lines routing calls between telephone switching centres, computer network connections and high speed computer data buses.

HIGH SPEED PCBs - Printed circuit boards (PCBs) are the most common method of assembling modern electronic circuits. It comprised of a overlapping of one or more insulating layers and one or more conducting layers which contain either signal traces or the powers and grounds traces, the design of the PCB layout is equally important as the design of the electrical circuit of high speed boards.

A Printed circuit board (PCB) provides both mechanically and electrically support to connect electronic components using tracks, pads and other features imprint from conductor sheets laminated onto a non-conductive substrate. Recent PCBs may contain passive components like capacitor and resistor along with the active devices embedded in the single substrate. Printed circuit boards are the simplest requirement of electronic products design.

II. ANALYSIS OF PCBs

Traces of the PCBs are considered for the discussion. Traces are modelled as the multi conductor transmission lines as shown in Figure 1

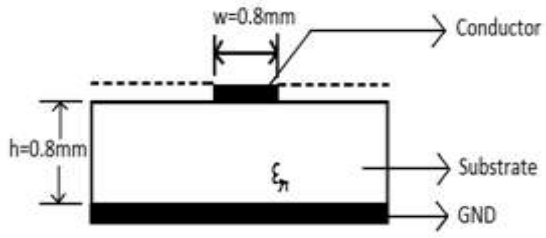


Figure.1.Basic Multi conductor Transmission line model

A. MATHEMATICAL ANALYSIS:

PCB trace design starts with the mathematical modelling of the trace structure which enables the designer to solve the field entities.

B. MICROSTRIP IMPEDANCE CALCULATION:

Micro strip impedance is calculated by using four parameters such as height of the structure, thickness of the conductor, width of the conductor, permittivity of the substrate [5].

Four parameters:

h=0.8mm, T=0.3mm, W=0.31, $\mathcal{E}_r$=4.6 [FR4]

The characteristic impedance of the MTL is calculated by,

$$Zo = \left(\frac{87}{\sqrt{\mathcal{E}r + 1.41}}\right)\left[ln\left[\frac{5.98h}{(0.8W) + T}\right]\right] ohm \dots (1)$$

Where,

Ɛ R: relative dielectric constant of the board material FR4(Ɛ r=4.6)

W: width of the conductor (line width)

T: thickness of the line (copper thickness), and

h :substrate thickness.

$$Zo = 49.87\Omega$$

The characteristic capacitance of the MTL is estimated by,

$$Co = \frac{0.67(\mathcal{E}r + 1.41)}{ln\left(\frac{5.98h}{(0.8W)+T}\right)} \dots \dots (2)$$

$$Co = 1.858\, pF/inch$$

The propagation delay is estimated by the characteristic impedance and capacitance by the following relation,

$$Tpd = Co \times Zo \frac{psec}{inch} \dots \dots (3)$$

$$Tpd = 142.82 \frac{psec}{inch}$$

One of the main ratio in the design of MTL systems is the ratio of width to height which can be either less than or greater than one. In the proposed method width to height ratio is founded as,

$$\frac{W}{H} = \frac{0.31mm}{0.8mm} = 0.3875$$

The width to height ratio is estimated as 0.38 which means the ratio is less than unity.

C. TRACE WIDTH TO CALCULATE CHARACTERISTIC IMPEDANCE:

The effective permittivity of the PCB trace is calculated depending on the width to height ratio

The PCB Microstrip trace for $\left(\frac{W}{H}\right) < 1$,

$$\mathcal{E}eff = \frac{\mathcal{E}r + 1}{2} + \frac{\mathcal{E}r - 1}{2} \times \left[(1 + 12\left(\frac{H}{W}\right)^{1/2} + 0.04\left(1 - \left(\frac{W}{H}\right)\right)^{2}\right].$$

$$\left(\frac{W}{H}\right) < 1; \varepsilon eff = 4.4 \dots \dots (4)$$

From the above mathematical solving, it is found that the effective relative permittivity of the material is 4.4, which lead to the selection of material as FR4 which has the effective relative permittivity as 4.6 and widely available commercially for the fabrication process of PCBs.

D. CHARACTERISTIC IMPEDANCE INTERMS OF EFFECTIVE PERMITTIVITY:

The objective of a PCB traces is to carry an electrical signal to a predefined point without the addition of any distortion, dispersion or interference to that signal. Same is ensured as the characteristic impedance, Zo, of the transmission line is constant at every point throughout the length of the line. The width and height of the conductor trace plays a vital role in impedance calculation and achieve characteristic impedance.

$\left(\frac{W}{H}\right) < 1$ ratio is used to calculated the characteristic Impedance by using the equation (5).

$$\text{Zo} = \frac{\frac{120\pi}{\sqrt{\mathcal{E}eff}}}{\left(\frac{W}{H}\right) + 1.393 + 0.667 \ln\left[\left(\frac{W}{H}\right) + 1.444\right]} \dots (5)$$

$$Zo = 49.9\ \Omega$$

The characteristic impedance of the MTL PCB traces is founded as 49.9Ω which is rounded to 50Ω

E. EQUIVALENT CIRCUIT:

Understanding and characterization of the electrical response of MTLs, i.e. traces is done by developing an equivalent circuit. The equivalent circuit is developed to accurately model the response and replicate the working

of traces due to distributed element effect and make it practical enough to develop compensation process.

The equivalent circuits for the structures are derived by the mathematical solver Partial Element Equivalent circuit (PEEC) method is shown in Fiq.2, 3 and 4.

1) DOUBLE TRACE:

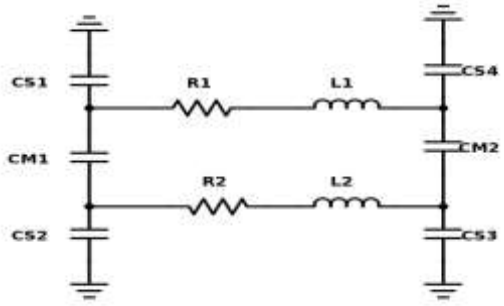


Figure. 2. Equivalent circuit of double trace

Where C_s- upper pad and lower pad capacitance and C_m-cylinder pad capacitance

2) DOUBLE TRACE WITH CONTINUOUS SHIELD TRACE (STRUCTURE 1):

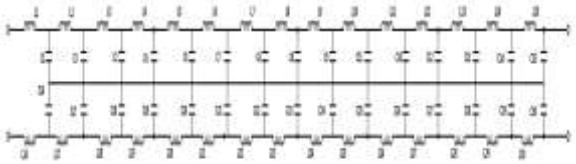


Figure. 3. Equivalent circuit of Structure 1

3) DOUBLE TRACE WITH DISCRETE SHIELD TRACE` (STRUCTURE 2):

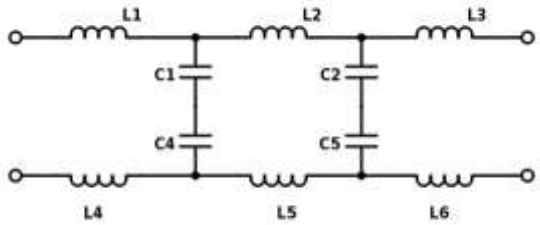


Figure. 4. Equivalent circuit of Structure 2

These circuits are the resultant of the mesh solved by the PEEC solver to find the field distribution

III. DESIGN OF PCBs DOUBLE TRACE

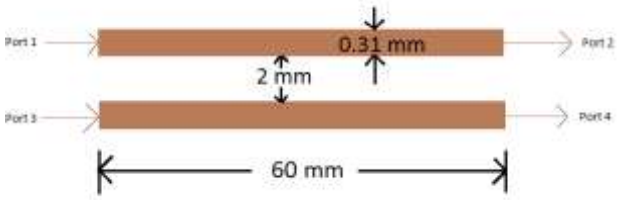


Figure 5 Dual trace multi conductor transmission line

Figure 5 shows double PCB trace designed using ADS . In order to validate the simulation result, the designed structure is fabricated in Fr4 substrate with thickness *h=0.8mm*.The structure's total length is about 60mm, width(W) 0.31mm,and space(S) 2mm. For all structure characteristic impedance is set as a constant 50 ohms. Input is given in port 1 and the output is analysis in port 2 of the transmission line port 3 and port4 are the nearby transmission line. The distance between two transmission line are reduced which is used to analysis the crosstalk, discontinuity, overshoot, propagation delay.

A.STRUCTURE 1:

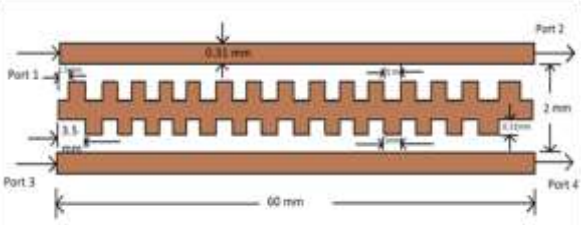


Figure.6. Double traces with continuous shield trace (Structure 1)

The frequency used for the above designing is about 3-5 GHZ and simulated in ADS.

B. STRUCTURE 2:

Figure. 7. Double trace with discrete shield trace (Structure 2)

The distance between the conducting trace and shield trace is reduced to get the maximum signal integrity. Computational Electromagnetic Modeling by means of Method of Moments and PEEC method can be analyzed using ADS tool.

VI. ESTIMATION OF SIGNAL INTEGRITY

1) SIGNAL INTEGRITY:

Signal integrity is ensured by quality of the electrical signal with is responsible for the signal transmission. Maximum Losses in the traces results in SI issues. Signal integrity issues are listed as crosstalk, discontinuity and propagation delay.

2) CROSSTALK:

Crosstalk is the coupling of Electromagnetic (EM) energy intentionally or unintentionally from one transmission line to another transmission line in a multi conductor line or traces placed nearby. Undesirable signal from neighbouring transmission circuits are the basics effect of crosstalk. It is usually caused by undesired capacitive, inductive or conductive coupling from one circuit, part of a circuit, or channel to another.

$$CROSSTALK = \frac{\left[\frac{Z_{oe}}{Z_{od}-1}\right]}{\left[\frac{Z_{oe}}{Z_{od}+1}\right]} \ldots\ldots\ldots (6)$$

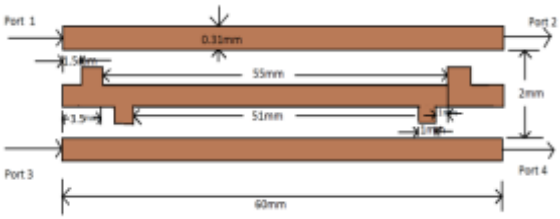
Where,

$$Z_{oe} = 276\ log\left(\frac{2D}{W}\right)\sqrt{\frac{1}{1+\left(\frac{D}{2h}\right)^2}}$$

$$Z_{oe} = Odd\ mode\ impedance$$
$$Z_{od} = Even\ mode\ impedance$$

A. CROSSTALK CALCULATION:

Crosstalk is calculated by solving equation 6 which in term calculates the even and odd mode Impedance. Various parameters are used to calculate like height of the



substrate (h), distance between the two transmission lines (D), and width of the conductor (w).

$$Z_{oe} = 276\ log\left(\frac{2D}{W}\right)\sqrt{\frac{1}{1+\left(\frac{D}{2h}\right)^2}} \ldots\ldots\ldots (7)$$

$$\boldsymbol{Z_{oe} = 191.47\Omega}$$

$$Z_{od} = 69\ log\left(\frac{4h}{W}\right)\sqrt{\frac{1}{1+\left(\frac{2h}{D}\right)^2}} \ldots\ldots\ldots (8)$$

$$\mathbf{Z_{od} = 54.57\Omega}$$

From equation (6),

$$CROSSTALK = \frac{\left[\frac{Z_{oe}}{Z_{od}-1}\right]}{\left[\frac{Z_{oe}}{Z_{od}+1}\right]}$$

$$= \frac{\left[\frac{191.47}{54.57-1}\right]}{\left[\frac{191.47}{54.57+1}\right]}$$

$$\boldsymbol{Crosstalk = 1.037}$$

The Signal integrity is been estimated by verifying insertion loss, return loss, impedance and radiation pattern.

The cross talk of the MTL PCB traces is founded as 1.037 which is rounded to 1.

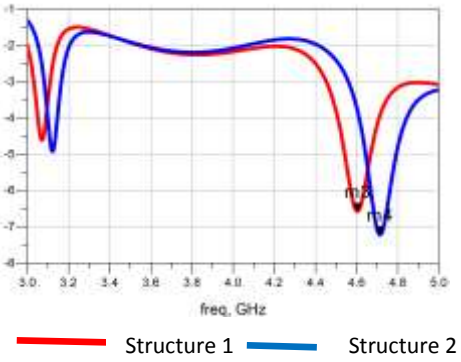


Figure 8 Comparison of Insertion losses S_{21} of both structures

Insertion Loss (dB) is defined as the loss in power as a signal travels in forward direction through the board . This value not only includes the reflected incoming signal, but also the attenuation of the component.

$$\text{Insertion Loss (dB)} = 10 * \text{Log10}\left(\frac{PR}{PT}\right)$$

Insertion Loss is the measure of total loss occurred in signal as it travels through a component. Therefore insertion loss must be as low as possible. In the above graph it is inferred that structure 2 has minimum insertion loss (17.744 dB) when compared to structure 1(13.776 dB). Henceforth structure 2 is observed to be a better structure compared to structure 1.

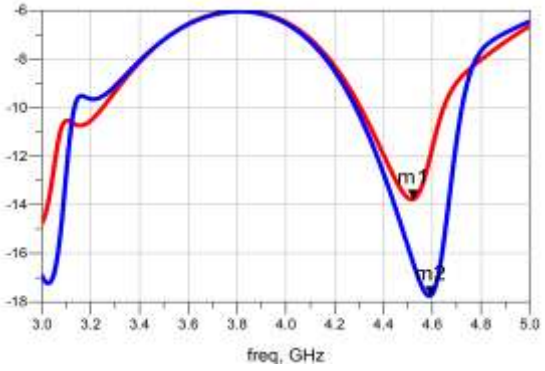


Figure 9 Comparison of Return loss S_{11} of both structures

Return Loss (dB) is defined as a ratio of the incoming signal to the same reflected signal at the same port. Return Loss is a frequency domain parameter analogous to the time domain impedance profile.

$$\text{Return Loss (dB)} = 10 * \text{Log } 10 \left(\frac{\text{Pr}}{\text{Pt}}\right)$$

In the above graph it is observed that return loss of structure 2 is minimum (6.578 dB) as compared to that of structure 1(7.222 dB).

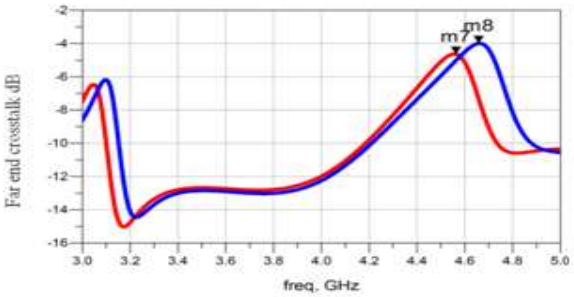


Figure 10 Near end crosstalk S_{31} of both structures

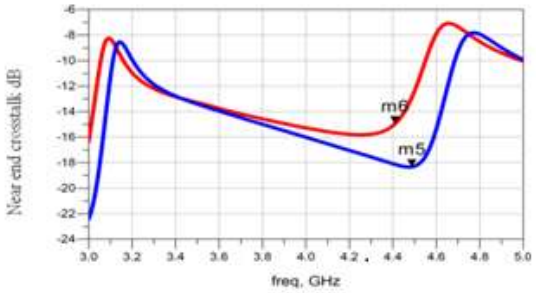


Figure 11 Far end crosstalk S_{41} of both structure

Figure 11 shows the Far end crosstalk of proposed transmission lines. The crosstalk is improved near 4.6dB without any marginal changes in insertion and return loss.

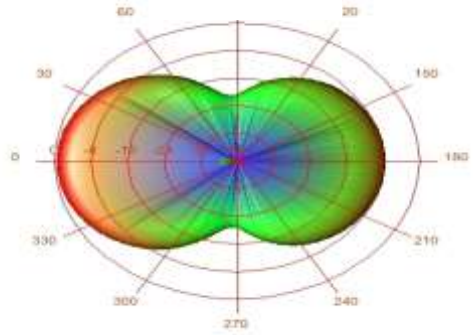


Figure 12 Radiation pattern for Structure 1

Figure 12 shows the radiation pattern of structure 1 which shows the appearance of green colour is maximum and ensure the less radiation which shows rich signal quality.

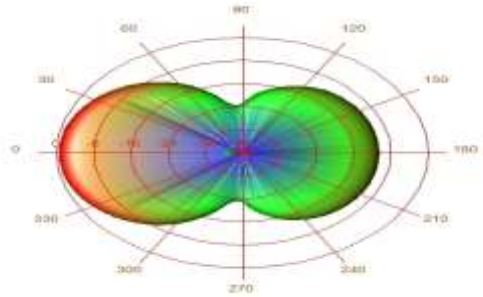


Figure 13 Radiation pattern for Structure 2

Figure 13 shows the radiation pattern of structure 2

V. RESULT ANALYSIS TABLES

Table 1. Comparison of simulated results between structures

S. No	Structure	Insertion Loss(dB)	Return Loss(dB)
1	Structure 1	-13.782	-6.578
2	Structure 2	-17.755	-7.194

Characteristic impedance=50Ω

Table. 2. Crosstalk estimation

Structure	Z_{oe} (Ω)	Z_{oo} (Ω)	Crosstalk(c)
Structure 1	191.47	54.57	1.037
Structure 2	191.47	54.57	1.037

VI SCHEMATIC ANALYSIS OF STRUCTURES

Schematic analysis of structures is done using schematic window of the ADS tool.

STRUCTURE 1:

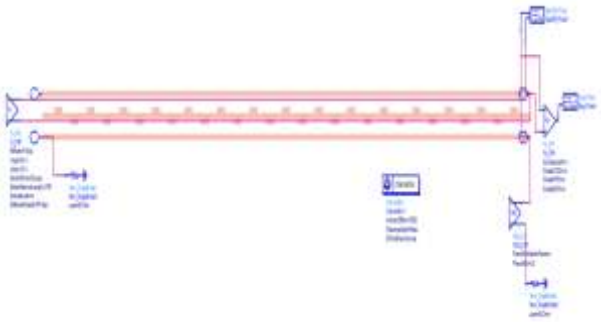


Figure 14 Schematic design for Structure 1

The schematic design of continuous shield conductor is connected along a perfect crosstalk component, which is used to analyse crosstalk performance. The crosstalk with 5Gbps and jitter value of 7e-3 is used for identifying crosstalk performance.

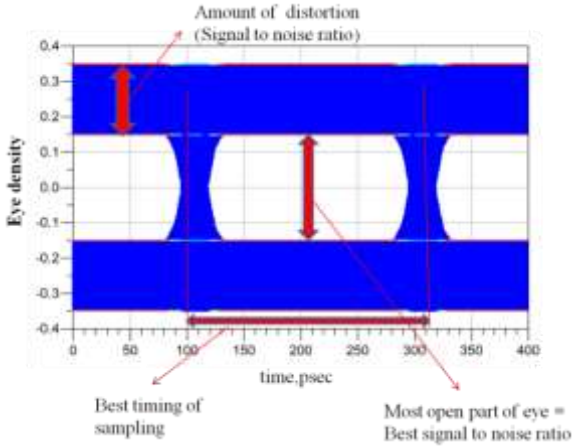


Figure 15 Eye pattern for Structure 1

STRUCTURE 2:

Figure 16 Schematic design for Structure 2

The schematic design of discrete shield conductor is connected along a perfect crosstalk component, which is used to describe crosstalk performance. The crosstalk with 5Gbps and jitter value of 7e-3 is used for identifying crosstalk performance.

The larger opening of the eye diagram at any given sampling time illustrates the margin over noise. Thus structure 2 will be a best structure for implementing in a multiconductor transmission line on a PCB.

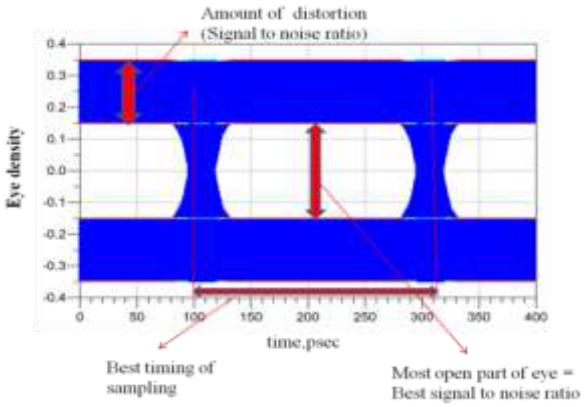


Figure 17 Eye pattern for Structure 2

Crosstalk has been reduced in structure 2 when compared to structure 1. This is inferred from the eye diagram given above.

VII. DISCONTINUITIES

All signal distributed circuits, like waveguides, coaxial lines or any other propagation structure, must inherently contain discontinuities.

A. DISCONTINUITIES CALCULATION

Discontinuity is calculated by using the open end transmission line and it can be analysis by using the equation,

$$\frac{\Delta l}{h} = \frac{\varepsilon 1\, \varepsilon 3 \varepsilon 5}{\varepsilon 4} \dots\dots\dots (9)$$

Where,

$$\Delta l - change\ in\ length$$

H: substrate thickness

Figure 18 Discontinuity diagram

$$\varepsilon 1 = 0.434907\left(\frac{\varepsilon re0.81 + 0.26\left(\frac{W}{H}\right)0.8544 + 0.236}{\varepsilon re0.81 - 0.189\left(\frac{W}{H}\right)0.8544 + 0.87}\right) \dots (10)$$

$$\varepsilon 2 = 1 + \left(\frac{\left(\frac{W}{H}\right)0.371}{2.35\varepsilon r} + 1\right) \dots\dots (11)$$

$$\varepsilon 3 = 1 + \left(\frac{0.5274\tan^{-1}\left[\frac{0.084\left(\frac{W}{H}\right)1.9413}{\varepsilon 2}\right]}{\varepsilon re0.9236}\right) \quad (12)$$

$$\varepsilon 4 = 1 + 0.037\ tan^{-1}\left[0.067\left(\frac{W}{H}\right)1.456\right][6 - 5\,e^{(0.036(1-\varepsilon r))}] \dots (13)$$

$$\varepsilon 5 = 1 - 0.218\,e^{\left(-7.5\left(\frac{W}{H}\right)\right)} \dots\dots (14)$$

Where,

$$W = 0.31, H = 0.8$$

Therefore,

By applying the parameter values in equation (10)

$$\varepsilon 1 = 0.434907\left(\frac{4.60.81 + 0.26(0.3875)0.8544 + 0.236}{4.60.81 - 0.189(0.3875)0.8544 + 0.87}\right)$$

$$\varepsilon 1 = 0.39$$

By applying the parameter values in equation (11)

$$\varepsilon 2 = 1 + \left(\frac{(0.3875)0.371}{2.35(4.6)} + 1\right)$$

$$\varepsilon 2 = 1.059$$

By applying the parameter values in equation (12)

$$\varepsilon 3 = 1 + \left(\frac{0.5274\ \tan^{-1}\left[\frac{0.084(0.3875)1.9413}{1.059}\right]}{\varepsilon re0.9236}\right)$$

$$\varepsilon 3 = 1.059$$

By applying the parameter values in equation (13)

$$\varepsilon 4 = 1 + 0.037\ \tan^{-1}[0.067(0.3875)1.456][6 - 5\,e^{(0.036(1-4.6))}]$$

$$\varepsilon 4 = 1.057$$

By applying the parameter values in equation (14)

$$\varepsilon 5 = 1 - 0.218\,e^{(-7.5(0.3875))}$$

$$\varepsilon 5 = 0.988$$

By applying the parameter values in equation (9)

$$\frac{\Delta l}{h} = \frac{0.39x1.1059x0.988}{1.057}$$

$$\boldsymbol{Discontinuities = 0.403}$$

A straight continuous length of transmission structure would be of less of engineering use, and in case of signal distribution like junctions are essential. Whereas, these junction causes discontinuity in PCBs.

Discontinuity in structure 2 is reduced compared to that of in structure 1, this is done by reducing the number of sharp edges in the shield trace of structure 2.

VIII. PROPAGATION DELAY

The transit time for a micro strip signal trace from source point to destination. The delay constant in ns/ft or ps/ft is a function only of the dielectric constant, and not the trace dimensions. This is the satisfied condition. It means that, with a given PCB material (and given εr), the propagation delay constant is fixed for various impedance lines.

A. PROPAGATION DELAY CALCULATION

The delay is expressed in ns/ft is a function only of the dielectric constant of the material. The dielectric constant for the substrate [ε] is 4.6 for FR4.

$$T_{pd}\left(\frac{ns}{ft}\right) = 1.017\sqrt{0.475\varepsilon r + 0.67} \dots\dots\dots (15)$$

Apply the permittivity values in equation (15),

$$T_{pd}\left(\frac{ns}{ft}\right) = 1.71\frac{ns}{ft}$$

This delay constant can also be expressed in terms of ps/inch, a form which will be more practical for smaller PCBs.

$$T_{pd}\left(\frac{ps}{inch}\right) = 85\sqrt{0.475\varepsilon r + 0.67} \dots\dots\dots..16$$

Apply the permittivity values in equation (16),

$$= 85\sqrt{0.475 \times 4.6 + 0.67}$$

$$\boldsymbol{T_{pd}\left(\frac{ps}{inch}\right) = 143.6\ \frac{ps}{inch}}$$

IX. FABRICATION PROCESS

From the above declarations, it is calculated that the structure 2 is better compared to that of structure 1.Structure 1 and 2 are fabricated accordingly and tested for validation of the simulated results. Fabrication process follows the given specification to ensure the practical Implementation.

Substrate : FR4, 0.8; Conductor: Copper, 0.3
GND : Perfect conductor, 10 Microns

A.TESTING AND FABRICATION RESULTS

Figure 19 Double trace with continuous shield trace

S.no	Parameter	S_{11}		S_{21}		S_{31}		S_{41}	
		Si.data (dB)	M. data (dB)	Si data (dB)	M. data (dB)	Si. Data (dB)	M. data (dB)	Si. Data (dB)	M .data (dB)
1.	Structure I	-13.78	-35.67	-6.578	-22.51	-15.507	-24.70	-4.640	-4.58
2.	Structure II	-17.75	-39.54	-7.194	-28.10	-18.182	-38.66	-4.006	-4.61

Table 3.Comparison between simulated (Si.) and measured(M.) data

X. CONCLUSION

The Maxwell approach for solving field entities is handled by analysing signal integrity of the complex configuration of trace lines has been studied. Various structures are studied to verify the effectiveness and optimal ability of this approach. The results were verified by comparing them with those of measurement and simulation for the proposed system. The proposed approach satisfactorily estimates the crosstalk for a transmission line model.

The proposed approach can be applied to analysis the crosstalk in the complex structures such as a variety of multi conductor lines. The return loss of structure 2 is better than structure 1 and obtained the crosstalk as explained in the result comparison table. Thus a stable return loss, Insertion loss is obtained for the structure 2 is satisfied than structure 1.

The non-radiating structure is flexible for implementing in multi conductor transmission line on PCB. The experimental result confirms that distortion is less which is analyzed via inter symbol interference, and eye pattern applicable for crosstalk reduction on PCB. Further the non-radiating structure can be implemented in multi conductor transmission line on PCB, which can be a best structure for reducing crosstalk and ensuring high signal integrity. The proposed method can be applicable for high frequency applications and high-density digital electronic equipment.

Discontinuity can be reduced by reducing the number of sharp edges in a structure. By this it is known that structure 2 has less number of sharp edges than that of structure 1 and hence structure 2 is better than structure 1. The discontinuity in structure 2 is better than structure 1.

Propagation delay depends on the permittivity of the substrate used. Therefore the substrate must have a permittivity which is very much less. Hence FR4 is used instead of alumina to reduce the propagation delay. Because FR4 has a permittivity of 4.6 which is less compared to alumina that has a permittivity of 9.6.

REFERENCE

[1] A.B. Gurulakshmi and Dr. N. Sureshkumar, "*Electromagnetic Modeling of Multi conductor Transmission Lines in the Analysis of Crosstalk at the bents of PCB*", International Journal of Applied Engineering Research, Vol. 10, No. 75, Pp. 106-109, 2015

[2] A.B. Gurulakshmi and Dr.N. Sureshkumar, "*Analysis and Estimation of Crosstalk in Multi Conductor Transmission Lines*", International Journal of Printing, Packaging & Allied Sciences, Vol. 5, No. 1, February 2017, PP. 740-749

[3] Ahmed Ali Hussein 2014, '*Analysis of cross-talk and discontinuity problems in microstrip lines*', Ph.D. thesis, Eastern Mediterranean University.

[4] Kachout Mnaouer, Bel Hadj Tahar Jamel &Choubani Fethi 2011, '*Crosstalk prediction for three conductors non uniform transmission lines: theoretical approach & numerical simulation*', Journal of Electromagnetic Analysis and Applications, vol. 3, pp. 319-327.

[5] Luca De Camillis, Francesco Ferranti, Dries Vande Ginste&Daniël De Zutter 2012, '*Parameterized partial element equivalent circuit method for sensitivity analysis of multiport systems'*, vol. 2, no. 2, pp. 248-255.

[6] Peerawut Yutthagowith, Akihiro Ametani, Naoto Nagaoka & Yoshihiro Baba 2011, '*Application of the partial element equivalent circuit method to analysis of transient potential rises in grounding systems'*, vol. 53, no. 3, pp. 726-736.

[7] Sang Wook Park, Yoon Eui Nahm& Yoshio Kami 2014, '*Coupling analysis of complex-layout traces using a circuit-concept approach'*, IEEE transactions on Electromagnetic Compatibility, vol. 56, no. 1, pp. 208-220.