Study of Impedance Control of Differential Transmission Lines for High Speed Circuits on Printed Circuit Boards

André Luiz Printes
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
aprintes@uea.edu.br

Italo Tony da Costa Alves
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
eng.italotony@gmail.com

Lennon Brandão F. do Nascimento
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
lennonbfn@gmail.com

Yago Costa de Oliveira
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
yago.cst.oliveira@gmail.com

Fábio Arozo Albuqurque
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
faalbuquerque@hub-uea.org

Fábio de Sousa Cardoso
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
fcardoso@uea.edu.br

André Wagner Lima Osório Filho
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
andrewagner.hub@gmail.com

Daniel Guzman Del Rio
Embedded Systems Laboratory
HUB – Tecnologia e Inovação
Manaus - AM, Brazil
drio@uea.edu.br

***Abstract* — This study investigated the proper design of differential transmission line impedance on a printed circuit board (PCB). The objective was to achieve an impedance of 100 Ohms by varying the trace thickness within the range of 0.02mm to 0.12mm, while keeping the spacing between the trace's constant at 0.127mm. The results demonstrated that the trace with a thickness of 0.1mm achieved the desired impedance, considering parameters such as copper height and spacing between PCB layers, providing valuable insights for high-speed circuit design.**

Keywords — multilayer PCB; differential transmission line PCB; microstrip line PCB; system on a module (SOM); impedance control PCB

I. Introduction

Currently, technology development teams for the services that banks around the world provide to their customers have invested heavily in the search for electronic and computer means that allow for greater security during operations. In this context, the industries that produce banknote dispensers, called ATMs or ATMs (Automatic Teller Machines) have sought to improve their systems in order to minimize violations and, therefore, losses for their customers due to fraud or criminal subtractions. of values stored in these devices. Currently, these ATM are made up of processing units that are often used in modules, also called Computer System in a Module, System on a Module (SoM) multiple operations performed. In [1], this current context of banking systems is approached by Neto, showing that [...] "From the point of view of banking and commercial service, for example, we are facing a new reality, with a greater volume of transactions to be managed – and with more customers demanding these services".

With the advancement of electronics technology and the demand for high-speed systems, proper sizing of differential transmission line impedance has become a critical consideration in printed circuit board (PCB) design. In particular, in memory systems, where signal integrity is essential, correct impedance sizing is essential to ensure efficient and reliable data transmission.

ISSN 2447-8326. V.1 © 2023 BTSYM

Differential transmission lines are widely used in PCI for transmitting high-speed signals, such as on memory buses. These lines consist of pairs of adjacent conductive tracks, designed to transmit signals in phase but with opposite polarity, which results in greater noise immunity and better common mode rejection. However, to ensure the proper performance of these lines, it is crucial that the differential imped ance is correctly sized.

Furthermore, as electronic systems become more complex and require greater component density, the construction of multi-layer printed circuit boards has become common practice. These boards are composed of several layers of conductive tracks separated by dielectric substrates. Multilayer construction offers benefits such as reduced electromagnetic interference between tracks and the ability to route tracks across different layers, allowing for more compact and efficient layouts.

However, the construction of multilayer printed circuit boards also introduces additional challenges in dimensioning the differential impedance. Consideration of layers and dielectric materials between tracks is essential to ensure uniformity and consistency of impedance across the entire board. In addition, proper track design, including width, copper thickness, and track spacing, plays a critical role in impedance sizing.

In this context, this article proposes to present an approach for dimensioning the impedance of differential transmission lines on multilayer printed circuit boards, focusing on memory systems. Theoretical concepts, relevant equations and practical considerations for determining the appropriate impedance will be explored. In addition, the construction stages of multilayer PCBs will be discussed, considering the dimensions of the tracks, the dielectric materials used and other relevant parameters.

II. THEORETICAL BASIS

A. *Multilayer PCBs*

Multilayer boards are key components in the modern electronics landscape, playing a vital role in the manufacture of increasingly sophisticated and compact devices. While single or double sided printed circuit boards (PCBs) have traditionally been used in electronic applications. Multi-layer boards have emerged as an indispensable solution to meet increasing demands for greater component density and improved electrical performance.

The multilayer board consists of a sandwich of pairs of double-sided copper inner layers, each separately processed and etched, with their track patterns separated by layers of "pre-preg", a simple glass-epoxy material without lamination. copper. The outer layers of copper are applied as sheets, initially unprocessed. Due to this assembly method, multilayer boards always consist of an even number of copper layers: four, six, eight and so on [2].

It is possible to visualize, in Fig. 1, the representation of a set of several layers of a laminated base material of a multilayer board. There are different variations of this type of construction, some containing holes that go through all the layers, while others have internal holes that connect only the inner layers.

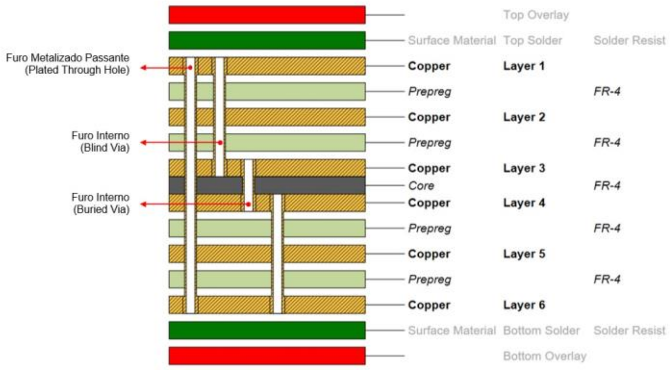


Fig. 1. Representation of layer overlaps with through and internal holes of a multilayer board.

In addition, the proper selection of dielectric materials is critical to ensure the desired electrical performance on the board. Relative dielectric constant is a critical parameter that affects track impedance and signal propagation speed. Reference [3] shows us that the dielectric constant (or relative permittivity, εr) of a PCB material has several characteristics that must be considered at the design stage and are often ignored until the actual performance of the circuit requires attention.

Dielectric materials with lower dielectric constants are generally preferred to reduce propagation velocity and improve high frequency performance.

The most common laminates are epoxy glass and phenolic paper. Phenolic paper or synthetic resin paper is cheaper and can be easily perforated, therefore its main application is in high volume, domestic and other non-critical sectors. It is electrically inferior to epoxy glass, has a limited temperature range, absorbs moisture easily, but is not suitable for plated-through-hole construction [2].

This fundamental technology has enabled the continued evolution of electronic devices. Its laminated structure and advanced manufacturing process allow for higher integration density, better electrical performance and a wide range of applications. In this context, innovative solutions have been developed, such as physics-based pathway models and impedance tuning techniques, to ensure the optimized performance of multilayer boards at higher frequencies.

B. *Trasmission Line*

A printed circuit board transmission line is an essential component in electronics and communications systems, designed to transmit high frequency electrical signals with integrity and efficiency. High frequency transmission lines and waveguide structures are unquestionably the most essential building blocks in radio frequency, microwave and millimeter wave circuits and systems.

Several types of transmission lines are used, depending on the specific system requirements. The two most common types are the microstrip line and the Coplanar Waveguide - CPW.

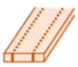
Two types of typical structures are often used as high-speed and high-frequency interconnects for signal transmission, the microstrip line and the coplanar waveguide [4].

Transmission lines are manufactured in the form of non-planar and planar metallic lines, such as rectangular waveguides, coaxial lines, strip lines, microstrip lines and coplanar waveguides (CPW), for electronic components and circuits. , or lines dominated by dielectrics, such as dielectric waveguides and fiber optics, for photonic components and circuits [5].

Waveguide Integrated into the Substrate (WIS) is a transmission line structure that was proposed in the 2000s and can be synthesized in planar form for integration with other transmission lines. It allows the creation of low cost and high integration microwave and terahertz circuits, combining features of planar and non-planar structures. SIW is widely used in microwave and millimeter wave applications such as 5G communications, automotive radar and satellite systems. Its popularity has grown and several variants have been developed to meet different requirements. The SIW represents a disruptive fifth generation of microwave circuits, allowing a transition from the exclusive use of Transverse Electromagnetic Modes (TEMs) to other guided wave structures [6-7].

Transmission line configurations and guided wave structures have gained popularity due to their effectiveness in meeting the specific requirements of diverse applications in this frequency spectrum, Table I. These structures are designed with bespoke geometries and materials to ensure integrity and performance. optimum of the signals in both the time domain and the frequency domain.

TABLE I. TRANSMISSION LINES AND WAVEGUIDE STRUCTURES FOR PROJECTS AND APPLICATIONS IN THE FREQUENCY RANGE FROM MEGAHERTZ TO TERAHERTZ.

Tipo	**Substrate Integrated Waveguide**	**Microstrip**	**Coplanar Waveguide**	**Strip Line**
Illustration				
Fundamental Mode	Quasi-TE	Quasi-TEM	Quasi-TEM	TEM
Modal Dispersion	3	4	4	5
Bandwidth	4	4	3	4
Transmission Loss	3	1	2	2
Power Dissipation	3	2	2	3
Physical Size	3	5	4	3
Ease of Manufacturing	4	5	5	3
Integration	5	5	5	4
Packaging and Shielding	4	2	2	3

Thus, high frequency transmission lines and waveguide structures play a critical role in the design and implementation of advanced electronic systems, enabling the efficient and reliable transmission of signals in a wide range of applications, from wireless communications to high speed and extremely high frequencies.

C. *Microstrip line*

Microstrip lines are one of the most widely used transmission structures in high-frequency circuits due to their advantages of low cost, ease of fabrication and integration into printed circuit boards.

The microstrip line is the most widely used interconnect in radio and microwave frequencies. It is also the main member of a broad class of transmission lines built on printed circuit boards [8].

The microstrip line is composed of a conductive metallic track printed on a layer and separated by a dielectric, as illustrated in Fig. 2. In this way, the ground plane is located in the layer below the dielectric, performing the function of conducting the return of the signal, reduction of electromagnetic radiation, impedance stabilization and protection against external interference.

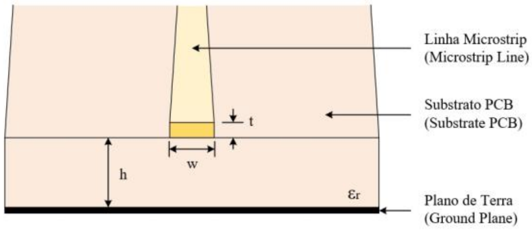


Fig. 2. Microstrip transmission Line.

The cross-sectional geometry, consisting of the width (w) of the strip and the thickness (h) of the substrate, determines the relationship between the voltage and current signals propagated along the microstrip line. This ratio is known as the characteristic impedance of the line and is critical to ensuring reliable signal transmission. To achieve a constant characteristic impedance, it is essential that the cross-sectional geometry is kept uniform along the line.

The characteristic impedance of the microstrip line is also geometrically defined, now by the width of the metallic track and by the distance between the track and the ground plane, which is just the thickness of the circuit board dielectric [8].

Microstrip transmission lines play a key role in high-frequency circuits, providing a versatile and cost-effective platform for transmitting signals and data. With continued research and advancements in this area, microstrip lines will continue to contribute to the next generation of communication systems and high frequency technologies. They are sized following the equations provided in the IPC-2141A documents.

D. Impedance control

Impedance control on printed circuit boards is a crucial step in the design of high frequency and radio frequency electronic systems. Accurate impedance matching is essential to ensure efficient signal transfer and minimize unwanted reflections that can degrade circuit performance.

The characteristic impedance (Zo) is the most important parameter for any transmission line. It is a function of the geometry of the transmission line, as well as the materials used, and is a dynamic value independent of the transmission line length [2].

When designing electronic circuits, we carefully adjust the width of the signal path, spacing between the path and the ground plane, thickness of the substrate, and the type of dielectric material used in the transmission line to ensure that the impedance of the circuit matches. to the specified characteristic impedance for the line. This is particularly important in high-frequency and radio-frequency systems, where impedance matching is critical to prevent signal dropouts and unwanted reflections.

Thus, the geometry of the transmission line plays a key role in determining the characteristic impedance, which in turn directly affects the efficiency and integrity of the signal transmitted along the line. A proper match between the characteristic impedance of the line and the source and load impedances is essential to ensure efficient transmission free of unwanted reflections.

It is widely known that in a printed circuit board, the parasitic resistance of the transmission line is very small and can be ignored compared to its designated characteristic impedance, which usually has a typical value of 50 Ohms. However, for a uniformly printed transmission line, due to its limited conductivity and extremely thin thickness, the parasitic resistance becomes very large and represents a significant proportion of the impedance. This causes a mismatch in the system impedance and results in distortions in the transmitted signal. To reduce parasitic resistance, multi-layer printing can be adopted to increase the transmission line thickness. However, even so, parasitic effects cannot be completely eliminated, and the cost will increase proportionally with the number of layers printed [4].

Impedance mismatches between adjacent parts of a transmission line path can result in signal distortion and failure of component electrical functions. The characteristic impedance control procedure has been considered one of the most necessary tools in the production of high electrical performance interconnections [9].

For high frequency applications, advanced techniques such as buried tracks, blind tracks, bypasses and meander tracks can be used to control impedance more accurately. Circuit simulations and modeling are also valuable tools for predicting transmission line behavior and optimizing impedance control on complex printed circuit boards.

Finally, impedance control on PCBs is a critical step in the design of high-frequency electronic systems. Ensuring proper impedance matching is essential to avoid signal reflections and power losses. However, careful approaches such as proper geometry selection, materials and advanced design techniques are critical to achieving the desired impedance control in PCBs. Knowledge and proper application of these strategies are essential for the development of efficient and reliable electronic systems.

III. MATERIALS AND METHODS

This section describes the steps adopted to carry out the sizing of the differential transmission line impedance on printed circuit boards for the differential signal. Initially, the multilayer structure of the printed circuit board is designed. The number of necessary layers and the location of the reference planes were considered as requirements. Next, the differential microstrip routing technique is selected for the transmission lines. In addition, the target impedance to be achieved for the differential transmission lines at 100 Ohms was defined, considering the system needs and design standards. Figure 4 shows the differential transmission line to be dimensioned.

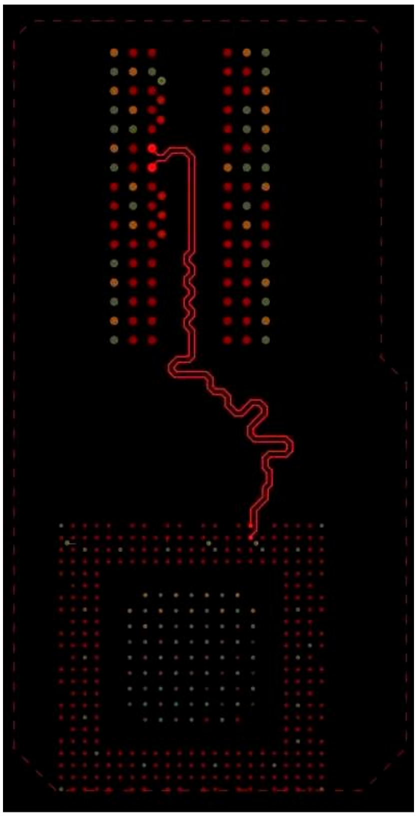


Fig. 3. Illustration of the microstrip differential transmission line used.

After that, using theoretical equations and microstrip models for differential transmission lines, the track width, copper thickness, track spacing fixed at 0.127mm and dielectric height were considered. The dielectric constant of the material used in the plate was also considered, which was defined as 4.32. Then, the range from 500 kHz to 3 GHz was considered, covering the typical operating frequencies of the differential memory system.

Finally, through the equations for the model, the results obtained are analyzed, where the impedance reached at different frequencies within the defined range is evaluated, verifying the desired value of 100 Ohms. Other parameters were also analyzed, such as impedance stability over the frequency range and signal attenuation. Next, in Fig. 3, a description of the steps mentioned for this dimensioning is presented.

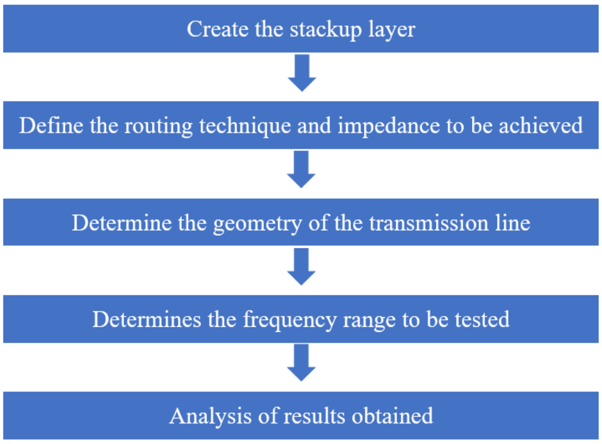
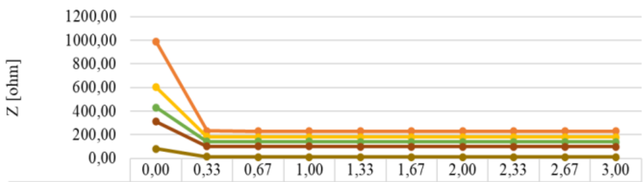


Fig. 4. Step-by-step description of the development, conformation and testing of the SoM module.

IV. Results

In this section, the results obtained by varying the trail thickness of the differential pair at different frequencies are presented.

In Figure 5, the impedance curves of each thickness are plotted as a function of frequency. It was observed that larger track thicknesses resulted in lower impedances, while smaller track thicknesses resulted in higher impedances. However, only the 0.1mm track provided the desired value within the frequency range of interest.

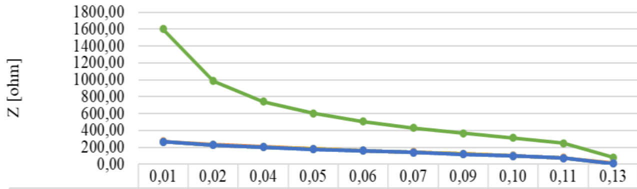


	0,00	0,33	0,67	1,00	1,33	1,67	2,00	2,33	2,67	3,00
0.022778mm	986,86	232,99	231,02	230,14	229,62	229,27	229,02	228,82	228,67	228,54
0.048333mm	601,39	182,51	181,09	180,46	180,09	179,84	179,66	179,52	179,41	179,32
0.073889mm	430,13	142,60	141,40	140,86	140,54	140,33	140,17	140,05	139,96	139,88
0.099445mm	311,86	101,80	100,60	100,06	99,74	99,52	99,36	99,23	99,14	99,05
0.125000mm	80,40	12,92	11,67	11,06	10,68	10,42	10,22	10,06	9,94	9,83

Freq [GHz]

Fig. 5. Differential impedance for different frequencies.

In Figure 6, the impedance is plotted on the y axis as a function of the track thickness on the x axis, with curves corresponding to different frequencies ranging from 500kHz to 3GHz. It is observed that, as the track thickness increases, the impedance decreases for all frequencies. However, this relationship is not linear, and different frequencies show different behaviors. This result indicates that the track thickness must be carefully selected based on the frequency range of interest to achieve the desired impedance. For a band close to 1GHz, the impedance behavior was approximately 100 Ohms for a thickness between 0.09 and 0.1mm.



	0,01	0,02	0,04	0,05	0,06	0,07	0,09	0,10	0,11	0,13
500.0000kHz	1598,4	986,86	742,87	601,39	503,98	430,13	368,64	311,86	248,72	80,40
333.7778MHz	270,09	232,99	205,34	182,51	161,88	142,60	123,14	101,80	74,85	12,92
1.333611GHz	266,21	230,14	202,99	180,46	160,02	140,86	121,45	100,06	72,76	11,06
1.666889GHz	265,03	229,27	202,28	179,84	159,46	140,33	120,93	99,52	72,10	10,42
2.000167GHz	264,41	228,82	201,91	179,52	159,17	140,05	120,66	99,23	71,75	10,06
3.000000GHz	264,03	228,54	201,68	179,32	158,99	139,88	120,49	99,05	71,53	9,83

Width [mm]

Fig. 6. Differential impedance for different frequencies.

The results obtained can be attributed to the electrical and geometric properties of the transmission line. Larger track thicknesses have lower impedance due to the increased conductive area, while smaller track thicknesses have the opposite effect. The spacing between the tracks also influences the impedance, but it was kept constant in this study.

V. Conclusion

This study demonstrated that the construction of a PCB for signals that have high frequency signals, the sizing of the transmission lines of these signals is crucial for their correct functioning. From parameters of dielectric constant from the materials used, as well as track width, spacing between differential tracks are variables that must be taken into account so that the impedance of the characteristic of the track is built according to the output impedance values of the transmitting element and input of the receiving element. The results showed that for the differential tracks brought in the study, the thickness of 0.1mm provided the desired impedance of 100 Ohms for the differential transmission line on a printed circuit board. These results highlight the importance of proper track sizing to ensure signal integrity in high-speed circuits. Future research may explore other variables, such as the effect of varying track spacing, for a more comprehensive understanding of impedance sizing on printed circuit boards.

VI. Acknowledgment

Thanks to the PD&I Project SoM module developed in Embedded Systems Laboratory belonging to the Hub of

Technology of the Superior School of Technology of the University of the State of Amazonas.

VII. References

[1] Neto, Matheus. Cybersecurity: The priority from ATM to hyperconnected global commerce. TIinside, January 17, 2022. Available: < https://tiinside.com.br/17/01/2022/ciberseguranca-a-prioridade-que-vai-do-caixa-eletronico-ao-comercio-global-hiperconectado/>. Acessed: 01/08/2023.

[2] Williams, Tim. The circuit designer's companion. Elsevier, 2004.

[3] Coonrod, John. "Understanding the variables of dielectric constant for PCB materials used at microwave frequencies." 2011 41st European Microwave Conference. IEEE, 2011.

[4] Liang, Zhenhao, Yanpeng Xie, and Qiang Chen. "Improving the Signal Integrity, Impedance Control and Attenuation of the Thin Transmission Lines Printed on Flexible Substrates." 2020 IEEE 3rd International Conference on Electronics Technology (ICET). IEEE, 2020.

[5] Wu, K. E., Maurizio Bozzi, and Nelson JG Fonseca. "Substrate integrated transmission lines: Review and applications." IEEE Journal of Microwaves 1.1 (2021): 345-363.

[6] D. Deslandes and K.Wu, "Integrated microstrip and rectangular waveguide in planar form," IEEE Microw. Wireless Compon. Lett., vol. 11, pp. 68–70, Feb. 2001.

[7] K.Wu, D. Deslandes, and Y. Cassivi, "The substrate integrated circuits - A new concept for high-frequency electronics and optoelectronics," in Proc. 6th Int. Conf. Telecom. Modern Sat. Cable Broadcast. Service (TELSIKS), Niš, Yugoslavia, Oct. 2003, pp. 3–10.

[8] Edwards, Terry C., and Michael B. Steer. Foundations for microstrip circuit design. John Wiley & Sons, 2016.

[9] Wang, Te-Chun, and Yin-Guang Zheng. "Effective dielectric constant method for trace impedance control." 2008 International Conference on Electronic Packaging Technology & High Density Packaging. IEEE, 2008.

[10] Barrett, Robert M. "Microwave printed circuits-a historical survey." IRE Transactions on microwave theory and techniques 3.2 (1955): 1-9.