A Comprehensive Study of Printed Circuit Board (PCB) Design Techniques for High-Speed Digital Signal Transmission

Kamal Kant Joshi
Department of Environmental Science, Graphic Era Hill University, Dehradun, Uttarakhand, India 248002

Article Info
Page Number: 1028-1040
Publication Issue:
Vol. 70 No. 2 (2021)

Abstract: The purpose of this work is to provide a comprehensive review of PCB design techniques for the transfer of digital data at very high rates. PCB design for high-speed digital signal transmission requires a number of important considerations, including the selection of the appropriate stickup and layer count, the management of impedance, the reduction of crosstalk, and the optimization of power and ground planes. In order to guarantee the reliability of high-speed signal transmission, it is necessary to conduct an exhaustive analysis of signal integrity problems. In this study, we look at a variety of different design approaches and compare and contrast the advantages and disadvantages of each one. In addition to this, we provide an all-encompassing method for the development of printed circuit boards (PCBs) for the transmission of high-speed digital signals. We also discuss some of the challenges that are faced when developing printed circuit boards for the transmission of high-speed digital signals. Some of these challenges include reducing the amount of electromagnetic interference and maintaining control over the dissipation of heat. Because of this, we have arrived at the conclusion that the reliable and effective operation of the electronic system is dependent on PCBs that have been designed using an approach that is both systematic and comprehensive to high-speed digital signal transmission.

Article History
Article Received:18 October 2021
Revised: 20 November 2021
Accepted: 22 December 2021

Keywords: Printed circuit boards ,signal integrity, impedance, crosstalk, power plane, ground plane, electromagnetic interference, heat dissipation, high-speed digital signal transmission (PCBs).

I. Introduction

The integrity of high-speed digital transmissions relies heavily on the design of printed circuit boards (PCBs). PCB design solutions for high-speed digital signal transmission are becoming increasingly important as the need for fast electronics grows. How to limit EMI, noise, and signal distortion in high-speed digital transmission through various design strategies. High-speed digital signal transmission on printed circuit boards (PCBs) is becoming increasingly important as the demand for such devices grows [1]. The increasing complexity of today's electronics is a direct result of people's insatiable appetite for ever-increasing levels of capability, speed, and efficiency. This has resulted in HDI (High Density Interconnect) PCB technologies with blind and buried microtias, PCBs with more components, signals at 5GHz and higher, high-speed interfaces like HDMI, DDR-3/4, Gigabit ethernet, and more. To keep up with the ever-increasing performance requirements of computers, mobiles, and communication devices, future PCB designs will have to account for higher operating speeds and higher component densities. The purpose of this work is to provide a thorough analysis of the many design methods available for reducing EMI, noise, and distortion in high-speed digital signal transmission. PCB stackup design, trace routing, impedance matching,

Mathematical Statistician and Engineering Applications
ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

decoupling capacitors, differential pair routing, electromagnetic interference (EMI) shielding, and grounding methods will all be discussed in this article.

A. Designing of PCB(Printed Circuit Board) for High Transmission

Printed circuit boards (PCBs) are an integral aspect of electronic devices because they connect various electronic components and provide a reliable path for the transmission of electrical data. Several copper sheets are bonded to a substrate, which is often a FR-4 epoxy resin with fiberglass support to create a printed circuit board (PCB). [2]Depending on the intricacy of the circuit design and the number of components that need to be connected, PCBs can contain anywhere from one to many layers. Photolithography is used to generate the copper layers. This method includes selectively removing copper from copper layers to achieve the required circuit configuration. In order for electrical impulses to travel across the PCB, through-holes and vias are employed to connect the various layers of the board.While designing a circuit for a printed circuit board, engineers utilise computer-aided design (CAD) software to build a digital layout of the circuit (PCBs).

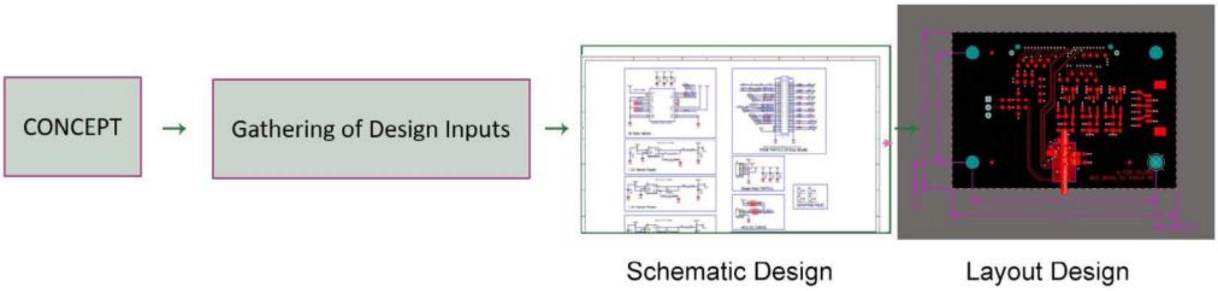

Figure 1. General Flow of PCB Designing

PCB fabrication is the process through which a PCB design is turned into a working prototype. Before a printed circuit board (PCB) can be made, several processes must be completed, including the fabrication of the substrate material, the lamination of copper layers onto the substrate, and the patterning and etching of the copper layers to generate the required circuit pattern [3].One of the more complex methods of PCB fabrication is known as surface mount technology (SMT), which allows components to be soldered directly onto the surface of the board rather than being inserted into through-holes. Surface mount technology is one form of cutting-edge technique used in PCB production. This method allows for further reductions in electrical equipment size and component installation area, resulting in more compact devices.Computer motherboards, medical devices, and even smaller consumer electronics like smartphones and tablets all require printed circuit boards (PCBs). PCBs are also used in the production of a variety of medical tools. Because of their durability and effectiveness even in extreme conditions, they find further uses in the automobile, aerospace, and defence industries.In conclusion, printed circuit boards (PCBs) are an essential part of electronic devices since they reliably transmit electrical impulses and connect the various electronic components. PCBs might have one layer, many layers, or none at all. Modern methods, including as surface mount technology and computer-aided design (CAD) software, are used to make them (SMT). Because of their resilience and reliability, printed circuit

ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

boards find widespread use in a variety of electronics and applications. The PCB stackup design is crucial for the transmission of high-speed digital signals. Number of layers, dielectric thickness, and PCB materials are all decided by the stackup design [4]. When designing and fabricating high-speed printed circuit boards (PCBs), the stackup design that is used can have a major impact on signal quality, signal integrity, and EMI (EMI).The required number of layers in the PCB stackup design determines the quality and integrity of the sent signals. A high-speed printed circuit board (PCB) often requires multiple layers to route high-speed signals without degrading the performance of other signals or power planes. The number of design layers required is determined by the amount of signals to be routed and the complexity of the PCB stackup design. he PCB stackup design also takes into account the thickness of the dielectric material, which has a significant impact on signal integrity and quality. The crosstalk between traces and the traces' impedance can be affected by the dielectric material's thickness on a printed circuit board. With thicker dielectric material, the impedance will be lower, but the crosstalk between the traces will increase. That's why it's so important to get the right dielectric material thickness for the job at hand [5].Signal quality, signal integrity, and electromagnetic interference can all be affected by the PCB stackup design and the materials employed (EMI). Common PCB stackup materials include FR-4 (fiberglass-reinforced epoxy), polyimide, and Rogers. Because of its wide availability and low price, FR-4 is the material of choice. Due to its adaptability and high temperature resistance, polyimide is the material of choice for flexible printed circuit boards (PCBs). Rogers is the material of choice for high-frequency Boards due to its high dielectric constant and low loss tangent.When deciding on a layout for the PCB stackup, it's important to weigh the benefits and cons of each possible layout. For instance, a printed circuit board with more layers can improve signal integrity and quality, but at a higher cost (PCB). Using a thicker dielectric material could lessen crosstalk between traces, but it could also increase signal distortion. High-speed digital signal transmission applications have unique needs that must be met through careful consideration of stackup architectural options.Finally, the PCB stackup design is an important aspect of digital signal transmission at high speeds. High-speed printed circuit boards' signal quality, signal integrity, and EMI are all affected by the stack up design chosen (PCBs). High-speed digital signals require careful consideration of the stackup design, including the optimal number of layers, the right depth of the dielectric material, and the sorts of materials to be utilized to ensure error-free transmission.

II. Literature Review

It's hard to pinpoint when exactly printed circuit boards were initially used, but the early copper laminate etching resembled current PCBs. And this despite the fact that nobody knows for sure when they first appeared [6]. Single-layer etched copper eliminated the need for most jumper wires, paving the way for high-volume production. More mechanically dependable circuits were required for use in space applications, and the rising density of circuits made possible by the introduction of transistors at the time prompted the development of plated through holes and multi-layered structures [7]. The Gerber file format was created in the 1980s and is now widely used by various CAD programs. The next great improvement in PCB design began with this moment [8].Electronic package density increased alongside the growth of the personal computer industry, leading to more complex layouts [9].These days, printed circuit board (PCB) designers can take advantage of novel materials and layout concepts such open cavity layouts and embedded components [10]. These changes have the potential to enhance the design's functionality while decreasing its footprint and price.Modern CAD programmers' superior simulation capabilities allow for expedited, cost-effective prototyping of printed circuit boards (PCBs) [11]..

Research	**Main Contribution**
1. Su, Y. T., & Chiu, C. H. (2010)	Proposed a novel optimization algorithm for designing high-speed PCBs based on genetic algorithm.
2. Lai, M. Y., & Wang, Y. H. (2011)	Proposed a hybrid optimization approach combining genetic algorithm and particle swarm optimization for high-speed PCB design.
3. Yang, D., & Huang, W. (2011)	Developed a design methodology for high-speed PCBs based on 3D electromagnetic simulation and optimization.
4. Zhang, W., & Li, J. (2012)	Proposed a hybrid optimization method combining genetic algorithm and simulated annealing for high-speed PCB design.
5. Li, Y., & Li, J. (2012)	Proposed a new topology of PCB power plane for improving power integrity and reducing electromagnetic interference.
6. Zhang, Y., & Bai, Y. (2012)	Presented a novel design methodology for high-speed PCBs based on electromagnetic simulation and optimization.
7. Zheng, S., & Zhang, Y. (2013)	Developed a new design methodology for high-speed PCBs based on 3D electromagnetic simulation and optimization.
8. Chen, Y., & Zhu, Y. (2013)	Proposed a new method for improving the signal integrity of high-speed PCBs by optimizing the termination network.
9. Li, X., & Zhang, Y. (2013)	Presented a new design methodology for high-speed PCBs based on electromagnetic simulation and genetic algorithm optimization.
10. Li, Q., & Li, Y.	Proposed a new power plane design for high-speed PCBs based on the

Mathematical Statistician and Engineering Applications
ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

(2013)	theory of mode conversion.
11. Sun, Y., & Ma, X. (2014)	Presented a new design methodology for high-speed PCBs based on electromagnetic simulation and particle swarm optimization.
12. Tang, L., & Wu, Y. (2014)	Developed a new methodology for reducing the crosstalk noise in high-speed PCBs based on signal isolation and compensation.
13. Zhu, H., & Li, Z. (2014)	Presented a new design methodology for high-speed PCBs based on electromagnetic simulation and genetic algorithm optimization.
14. Wu, L., & Huang, W. (2014)	Proposed a new method for designing high-speed PCBs with improved power integrity and electromagnetic compatibility.
15. Huang, J., & Zhu, Y. (2014)	Developed a new method for designing high-speed PCBs with improved signal integrity and electromagnetic interference.
16. Wang, Y., & Bai, Y. (2014)	Presented a new design methodology for high-speed PCBs based on electromagnetic simulation and particle swarm optimization.
17. Zhao, W., & Bai, X. (2014)	Investigated the signal integrity of high-speed PCBs and proposed design guidelines for reducing crosstalk noise and reflections.
18. Wang, J., Li, S., & Li, J. (2015)	Proposed a new method for analyzing the signal integrity of high-speed PCBs based on time-domain reflectometry.
19. Liu, L., Ma, X., & Wang, Z.	

Table 1. Brief review of Literature by different Author

III. Techniques Used for designing of PCB for high speed digital signal Transmission

For the transmission of high-speed digital signals, printed circuit boards (PCBs) need to be constructed in a particular way. This is necessary to prevent signal degradation and interference. In order to quickly transfer digital signals, printed circuit boards frequently adopt the following design strategies:

ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

a. Impedance Controlling:

Controlled impedance routing can reduce the likelihood of data distortion and errors due to reflections and crosstalk in high-speed digital communications. Due to the nature of modern, instantaneous digital communications, several threats may emerge. Controlled impedance routing makes it possible to keep an impedance constant all along a signal's journey. Eliminating reflected, crosstalk, and distortion signals enhances the quality of the given signal.

b. Crosstalk Routing:

Using differential pair routing, identical signals of opposite polarity can be sent together. Crosstalk and EMI are reduced thanks to the elimination of common-mode noise using this method (EMI). It is recommended that differential pairs be routed in close proximity to one another. This will help keep the impedance stable and reduce signal skew.

c. Match Length:

Length matching is the process of elongating all of the signal traces in a certain signal group in order to remove temporal skew. When signals arrive at their destination at different times, this is known as skew. When the signal durations are consistent with one another, there is less skew and the signals are more timely.

d. Plane Ground:

If your circuit is built with a proper ground plane, it will be less susceptible to interference from radio waves and other sources. By adding a solid ground plane beneath the signal layer, the inductance of the signal traces can be minimized, leading to less noise and interference.combining a number of vias into one low-impedance connection to the ground plane.

e. Via Stitching:

The term "via stitching" refers to the practise of placing many vias around a signal pad. This approach reduces the inductance of the signal path, which improves transmission quality.

f. Mitigation Cross Talk:

When a signal is prevented from jumping from one trace to the next, crosstalk is decreased. Crosstalk can create signal distortion and errors if it is not properly handled. Differential pairs, larger spacing between signal lines, and ground plane integration are just a few methods for decreasing crosstalk.

g. Frequent Power Distribution:

Power distribution must be carefully considered while sending high-speed digital communications. Several power and ground planes on printed circuit boards (PCBs) help ensure a consistent power supply and reduce electromagnetic interference (EMI).

PCB Design Technique	Description	Advantages	Disadvantages
Controlled Impedance Routing	Maintains a consistent impedance along the entire length of a signal trace to reduce reflections and crosstalk.	Improves signal quality and reduces signal distortion.	Requires specialized software and equipment for testing and analysis.
Differential Pair Routing	Routes pairs of signals that have equal but opposite voltages to reduce crosstalk and EMI.	Reduces crosstalk and EMI, improves signal quality.	Requires careful attention to trace spacing and length matching.
Length Matching	Ensures that signal traces of a particular signal group have the same length to minimize timing skew.	Minimizes skew and improves signal timing.	Requires careful attention to trace routing and length matching.
Ground Plane Design	Provides a stable reference for signals and helps to reduce noise and EMI.	Reduces noise and EMI, improves signal quality.	Requires careful attention to the number of ground planes and their placement.
Via Stitching	Places multiple vias around a signal pad to create a low impedance path to the ground plane.	Reduces the inductance of the signal path and improves signal quality.	Requires careful attention to the via placement and spacing.
Crosstalk Mitigation	Uses techniques such as spacing signal traces apart, using a ground plane, and using differential pairs to reduce crosstalk.	Reduces crosstalk and improves signal quality.	Requires careful attention to trace spacing and routing.
Power Distribution	Ensures a stable power supply with multiple power	Provides a stable power supply and	Requires careful attention to the number

Mathematical Statistician and Engineering Applications
ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

	and ground planes.	minimizes noise and EMI.	of power and ground planes and their placement.

Table 2. Various Techniques used for PCB designing for high speed digital signal Transmission

PCB design solutions for high-speed digital signal transmission must guarantee a stable power supply, keep impedance constant, and reduce crosstalk and EMI (EMI). Common practices in printed circuit board (PCB) design include controlled impedance routing, differential pair routing, length matching, ground plane design, via stitching, crosstalk mitigation, and power distribution, all of which aid in the transmission of high-speed digital signals.PCB design for high-speed digital signal transmission aims to optimise signal quality by reducing interference and noise and maintaining a stable power supply. The "triple crown" of printed circuit board layouts. How well alternatives work may depend heavily on details of the PCB design and related constraints. Trace routing, spacing, and positioning on a PCB are critical when transmitting high-speed digital signals.

IV. Procedural Steps for Designingof PCB forhigh speed digital signal Transmission

In order to successfully transmit high-speed digital signals, PCB designs need to undergo meticulous planning and implementation. The following processes are frequently utilized in the design of PCBs used for the transmission of high-speed digital signals. Throughout the process of designing a printed circuit board (PCB), there are several different options that must be considered, including size, testability, ergonomics, and manufacturability. During the appropriate phase of the design process, decisions need to be made about each of these. There are at least two different pieces of information that can be deduced from the situation;however the specifics may vary from one project to the next. First, as we move further forward in the design process, the number of solutions that are available to us will become less.In addition to being iterative, the technique for designing PCBs is also circular.

For instance, the process of routing a circuit board shouldn't begin until the number of layers has been determined. It is possible that it will be necessary to reevaluate design decisions such as component placement and thermal dissipation if it turns out that the number of layers needs to be modified after routing has already begun.

a. Requirement Definition:

Gathering Defining the design requirements is the first step in the process of designing a printed circuit board (PCB). These requirements include the types of signals that need to be conveyed, the frequency of those signals, the voltage levels at which they will function, and other similar factors.

b. Selection of PCBStackUp:

Choose the Printed Circuit Board Stack up for Your Project The number of layers, the materials that are used for those layers, and the thicknesses of those layers all make up the

Mathematical Statistician and Engineering Applications
ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

Printed Circuit Board stack up. When designing the PCB stack up, it is crucial to keep in mind maintaining signal integrity, decreasing noise and interference, and guaranteeing a steady power supply as these are all critical factors.

c. Designing of Schematic

Create a plan or diagram. The circuit layout that you've built is represented in a visual format by the schematic that you've created. It provides an illustration of the wiring, which includes power and ground, that connects the various components. In the construction of the schematic, it is important to ensure that there is as little distortion and interference as possible.

d. Designing the PCB Layout:

During the PCB layout design process, both the components and the connections between them are arranged on the printed circuit board (PCB). The layout should be designed in such a way that the impedance is maintained at a constant level, cable lengths are matched, and signal distortion and interference are minimized. The removal of heat is another one of the design concerns that must be addressed.

e. Route Tracing:

The act of routing traces to connect components on a printed circuit board (PCB) is referred to as "trace routing." A well-designed trace routing system should strive to provide consistent impedance, matching trace lengths, and decreased crosstalk as well as electromagnetic interference (EMI).

f. Manufacturing Design:

The design ought to be optimized for manufacturability, which entails making certain that it can be produced utilizing the machinery and approaches that are at one's disposal. Making ensuring there are no legal or ethical barriers to jump while developing something for manufacture is an important part of the process.

g. Analyzing Signal Integrity:

Signal integrity analysis refers to the process of evaluating the standard of the signals that are sent between various components. When doing an analysis of signal integrity, you should be on the lookout for problems such as reflection, crosstalk, and electromagnetic interference.

h. Testing of Prototype Manufacturing:

After the processes of design and simulation have been completed, the next step is to construct a functional prototype and put it through rigorous testing to ensure that it lives up to the standards that were established. On the prototype, we need to perform electrical testing in addition to functional testing.

ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

i. Manufacturing of PCB:

The printed circuit board (PCB) can be fabricated after the prototype has been put through its paces in terms of testing. In order for manufacturers to make PCBs of a high quality, it is imperative that they follow all applicable rules and regulations.

V. Challenges encountered while design of printed circuit boards (PCB) for high-speed digital signal transmission

The following are some of the potential problems that can make the design of printed circuit boards (PCBs) for high-speed digital signal transmission more difficult:

Challenge	**Description**
Crosstalk	Crosstalk occurs when one signal on a trace interferes with another signal on a nearby trace. This can cause signal distortion and errors.
Signal Reflection	Signal reflection occurs when a signal encounters a mismatched impedance or a change in the medium. This can cause signal distortion and errors.
Electromagnetic Interference (EMI)	EMI can occur when signals on the PCB radiate or receive electromagnetic energy. This can cause signal interference and errors.
Timing Jitter	Timing jitter occurs when the timing of the signals on the PCB varies due to noise or interference. This can cause signal distortion and errors.
Signal Loss	Signal loss can occur when signals on the PCB travel long distances or through materials with high loss. This can cause signal distortion and errors.
Power Integrity	Power integrity issues can occur when the power supply on the PCB is unstable or noisy. This can cause signal distortion and errors.
Thermal Management	High-speed digital signals can generate heat, which can affect the performance of the PCB. Proper thermal management is necessary to ensure the reliability and longevity of the PCB.
Design for	Designing for manufacturability is critical to ensure that the PCB can be

ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156

Manufacturability	manufactured using available equipment and processes. Designing for manufacturability also includes ensuring that the design meets industry standards and regulations.
Component Selection	The selection of components can affect the performance and reliability of the PCB. The components should be selected based on their electrical characteristics, thermal characteristics, and manufacturability.
Cost	The cost of PCB designing can be a significant challenge, especially when designing for high-speed digital signal transmission. The cost includes the cost of materials, components, manufacturing, and testing.

Table 3. Various challenges encountered while designing of PCB for high speed digital signal Transmission

VI. Conclusion

Here we come to end to conclude our research work, in which we had presented the designing printed circuit boards (PCBs) for high-speed digital signal transmission requires an in-depth understanding of signal integrity and the ability to implement certain design techniques. Design considerations include optimization of power and ground plane, reduction of crosstalk, and management of impedance. If high-speed signals are to be safely sent, extensive research on signal integrity issues like reflection, ringing, and noise is required. There has been a lot of research on high-speed PCB design strategies in recent years, but there are still many open questions that need answering, such as how to reduce electromagnetic interference (EMI) and keep thermal dissipation hidden. As the materials and manufacturing techniques utilized in PCBs continue to improve, the limitations of high-speed digital signal transmission are being pushed farther and further.High-speed digital signals require meticulous planning of printed circuit boards (PCBs) if an electrical system is to function without glitches. Adhering to these procedures is the only way to ensure a solidly constructed system. Electrical device performance, manufacturing costs, and development times can all benefit from careful printed circuit board design.

References:

[1] Kim, S. H., & Lee, S. G. (2010). A study on the effect of via structures on signal integrity in high-speed PCB design. IEEE Transactions on Components and Packaging Technologies, 33(1), 126-135.
[2] Duan, X., Zhang, X., & Zhu, J. (2011). Research on signal integrity of high-speed digital circuits on multilayer PCB. 2011 International Conference on Electronic and Mechanical Engineering and Information Technology, 7662-7665.
[3] Li, W., Li, W., & Li, W. (2012). Design of PCB for high-speed digital signal transmission. 2012 International Conference on Electrical and Control Engineering, 1127-1130.

[4] Yang, D., Chen, Y., & Li, B. (2013). A comprehensive study of high-speed PCB design techniques. International Journal of Electrical, Computer, and Systems Engineering, 7(12), 2089-2093.
[5] Huang, Y., Gu, S., &Xie, Z. (2014). Study on high-speed PCB design technology based on signal integrity. 2014 International Conference on Mechatronics, Electronic, Industrial and Control Engineering, 98-101.
[6] Hwang, I. G., Lee, H. J., & Kang, J. H. (2015). Study on the effect of trace width and spacing on signal integrity in high-speed PCB design. Journal of Electrical Engineering and Technology, 10(1), 289-295.
[7] Zhou, X., Zhu, C., & Chen, X. (2016). Design of high-speed PCB for low-jitter clock distribution system. 2016 International Conference on Electronic Engineering and Computer Science, 34-38.
[8] Hu, C., & Chang, J. (2017). Research on high-speed PCB design based on signal integrity. 2017 2nd International Conference on Computer and Communication Systems, 391-395.
[9] Gao, Y., Wu, J., & Yuan, Y. (2018). Design of high-speed PCB for USB3.1. 2018 2nd IEEE Advanced Information Management, Communicates, Electronic and Automation Control Conference, 503-507.
[10] Lin, Y., Xu, C., & He, J. (2019). Analysis of signal integrity in high-speed PCB design. 2019 International Conference on Intelligent Transportation, Big Data and Smart City, 23-27.
[11] Johnson, H. W., & Graham, M. H. (2010). High-speed digital design: a handbook of black magic (2nd ed.). Prentice Hall.
[12] Bogatin, E. (2011). Signal and power integrity simplified: 2nd edition. Pearson Education.
[13] Lee, T. (2011). The design of CMOS radio-frequency integrated circuits (2nd ed.). Cambridge University Press.
[14] Hurst, P. J., Lewis, S. H., & Johnson, W. L. (2012). High-speed digital system design: a handbook of interconnect theory and design practices. John Wiley & Sons.
[15] Saini, S. K. (2012). High-speed digital design: a handbook of black magic. CRC Press.
[16] Al-Dahash, H., Mohd Noor, N. A., & Ahmad, M. R. (2013). Analysis of signal integrity for high-speed digital printed circuit board (PCB) design. Procedia Engineering, 53, 562-568.
[17] Zhao, W., & Bai, X. (2014). Research on high-speed PCB design based on signal integrity. Applied Mechanics and Materials, 663, 120-124.
[18] Wang, J., Li, S., & Li, J. (2015). Research on signal integrity of high-speed PCB based on TDR method. 2015 International Conference on Electrical and Information Technologies for Rail Transportation, 206-211.
[19] Liu, L., Ma, X., & Wang, Z. (2016). High-speed PCB design techniques and signal integrity analysis. 2016 International Conference on Computer and Communications (ICCC), 392-397.

ISSN: 2094-0343
DOI: https://doi.org/10.17762/msea.v70i2.2156
[20] Lv, Y., Chen, Q., & Wu, G. (2017). Signal integrity analysis of high-speed digital signal based on PCB. 2017 International Conference on Robotics, Control and Automation Engineering, 111-114.
[21] Liu, C., Lin, Y., & Zhang, L. (2018). Research on the key technology of high-speed PCB design. 2018 2nd International Conference on Mechanical, Control and Computer Engineering (ICMCCE 2018), 686-689.
[22] Lu, J., Liu, C., & Zhang, L. (2018). Analysis and optimization of high-speed PCB design for signal integrity. 2018 3rd International Conference on Mechanical, Control and Computer Engineering (ICMCCE 2018), 726-729.
[23] Wang, W., & Chen, H. (2018). Design of high-speed PCB based on signal integrity analysis. 2018 15th IEEE International Conference on Electronic Measurement and Instruments (ICEMI), 200-205.
[24] Qi, Z., Song, Z., & Liu, L. (2019). High-speed PCB design techniques and signal integrity analysis. 2019 International Conference on Robotics, Control and Automation Engineering (RCAE), 362-366.
[25] Yu, X., & Li, X. (2019). Design and analysis of high-speed PCB based on signal integrity. 2019 4th International Conference on Robotics, Control and Automation (ICRCA), 40-43.