



African Journal of Biological Sciences



Validation of PCB design for ABS product in automotive industry.

Smriti Sinha¹, A Asuntha², Raheem Basha³

1. Department of Electronics and Control Engineering, SRM Institute of Science and Technology, Kattankulathur, Tamil Nadu

2. Department of Electronics and Instrumentation Engineering, SRM Institute of Science and Technology, Kattankulathur, Tamil Nadu

3. Continental Automotive India³

e-mail: asunthaa@srmist.edu.in

Abstract — This paper investigates a meticulous approach to validate Printed Circuit Board (PCB) designs specifically tailored for Anti-lock Braking System (ABS) products within the automotive sector. The primary focus is on the application of Design for Manufacturability (DFM) and Design for Assembly (DFA) principles to optimize the reliability, manufacturability, and efficiency of ABS PCBs. The paper presents detailed analyses, case studies, and practical insights into integrating DFM and DFA in ABS PCB design and validation processes.

Keywords—: *DFM, DFA, PCB CAM tool, PCB, ABS, Validation techniques, Immersion tin.*

Article History

Volume 6, Issue 5, 2024

Received: 22 May 2024

Accepted: 29 May 2024

doi: [10.33472/AFJBS.6.5.2024.](https://doi.org/10.33472/AFJBS.6.5.2024.7894-7902)

7894-7902

I. INTRODUCTION

The Anti-lock Braking System (ABS) plays a pivotal role in the dynamic landscape of automobile safety, mitigating risks associated with sudden braking. Central to this technology is a sophisticated network of Printed Circuit Boards (PCBs) orchestrating the ABS system's operation. Given the perpetual evolution in car engineering, ensuring the reliability and optimal functionality of PCB designs is paramount. This study explores the realm of PCB design validation for ABS products, with a specific emphasis on enhancing the effectiveness of these crucial components by integrating Design for Manufacturability (DFM) and Design for Assembly (DFA) concepts.

The automotive industry is witnessing a paradigm shift towards advanced safety features, necessitating a corresponding evolution in underlying technologies. ABS systems, designed to prevent wheel lock-up during braking, epitomize this progress. The seamless coordination of multiple components is essential for ABS functionality, with PCBs acting as the central nervous system orchestrating intricate signals and controls. The efficacy of ABS relies heavily on the dependability of its constituent PCBs, necessitating a robust validation procedure that surpasses standard practices.

A foundational aspect of this research lies in Design for Manufacturability (DFM), recognizing the inherent interconnection between designs and manufacturing processes. DFM principles guide the PCB design process, optimizing it for productive and economical manufacture. Ensuring alignment with manufacturing capabilities, DFM reduces the likelihood of errors and expedites the production process. Within the ABS domain, where precision is paramount, DFM becomes an indispensable facilitator for achieving maximum production efficiency while upholding product quality. This encompasses considerations from component location to signal integrity, temperature management, and material selections.

Concurrently, Design for Assembly (DFA) principles contribute to the efficiency of the subsequent assembly process. Given the integration of a diverse array of electronic components in ABS products, the ease with which these components come together during assembly is crucial. DFA principles guide design to enhance manufacturability, simplify assembly procedures, and decrease the probability of errors during assembly.

The research integrates DFM and DFA principles into the validation process, aiming to enhance the reliability and performance of ABS products' PCB designs. Through a comprehensive analysis and systematic validation approach, the study seeks to offer valuable insights to the automotive engineering community, fostering advancements in ABS technology. Consequently, this research aspires to be at the forefront of shaping the future of ABS technology through meticulous PCB design validation, contributing to the ongoing evolution of automotive safety standards and enhancing the overall safety and performance of modern vehicles.

Design for Manufacturability (DFM) in ABS PCBs:

- Defines key principles of DFM and their relevance to ABS PCB design.
- Explores strategies for optimizing ABS PCB layouts to streamline the manufacturing process.
- Discusses material selection, component placement, and routing considerations to enhance manufacturability.

Design for Assembly (DFA) in ABS PCBs:

- Introduces DFA principles and their significance in ABS PCB design.
- Analyzes strategies for simplifying assembly processes and reducing production time.
- Examines the integration of automated assembly techniques to enhance the overall efficiency of ABS PCB manufacturing.

Benefits and Impact:

- Quantifies the benefits of incorporating DFM and DFA in ABS PCB design.
- Analyzes the impact on manufacturing costs, reliability, and time-to-market.
- Presents a comparative analysis of ABS PCBs designed with and without DFM and DFA principles.

Validation Techniques for DFM and DFA in ABS PCBs:

- Provides an overview of simulation tools and methodologies for validating DFM considerations.
- Describes testing protocols to verify the effectiveness of DFA strategies in ABS PCB assembly.
- Discusses the integration of prototyping in the validation process to ensure real-world applicability.

II. LITERATURE SURVEY

Xiaoxiao Song, Keyu Wang, Zhuo Chen, Kaixu Ren, Peng Liu. The impact of four typical surface finishes on electrochemical migration in printed circuit board production was studied using a modified water drop test, and the electrochemical migration phenomena of each surface finish at different bias voltages were analyzed. Based on the analysis of the experimental results, the four surface finishes were ranked in terms of their ability to resist electrochemical migration, as follows: electroless nickel/immersion gold (ENIG) > lead-free hot air solder leveling (HASL) > HASL > Cu. In the PCB manufacturing industry, metallic surface finishing of the exposed electrodes on the PCB is critical for improving reliability, conductivity, and solderability. The sensitivity of different PCB electrode surface finishes to ECM (Electrochemical Machining) varies widely. Several studies have been conducted to analyze the ECM phenomenon with common metallic materials on PCBs, including pure tin,[2][8] tin- based solder alloys,[1][9][10] and sintered nano- silver.[11][12] Similarly, some of the studies were based on different surface finishes of PCBs, including hot air solder leveling (HASL),[13] immersion silver (ImAg),[14][15] copper-clad laminate (Cu),[16] and electroless nickel/immersion gold (ENIG).[17][18].

III. METHODOLOGY

To validate the any design of the Printed Circuit Board it must gone through multiple stages. According to the customer need and specification, specification sheet is prepared which will contain all the requirements of the board like PCB Size, Base material, Base copper thickness of inner layer and the outer layer, Maximum Application temperature, Smallest / Largest hole diameter, requirement of press fit / single pin design, Minimum line width / minimum spacing laser drill vias (micro vias) requirement, Minimum Tg of base material, DSC (Differential Scanning Calorimetry), Thermal cycle requirement, capable for lead free profile in SMD, Copper surface finish like OSP (Organic Solderability Preservative) , HASL (Hot air solder leveling), Immersion silver, Immersion tin, ENIG (Electroless Nickel Immersion Gold), Closed hole requirement, Impedance, Carbon ink / silk screen, Fine pitch or BGA application, CAF (Conductive Anodic Filament) level and so on. Once project is awarded design of the PCB is ready to validate with the DFM and DFA. The file format should be open data base format so that tool is used to validate the design should be function properly. Analyzing the design according to the DFM & DFA guideline will prepare the deviation list which consist of blocking point during the manufacturing and hence send for the correction in the design.

Base material: The base material in PCB (Printed Circuit Board) serves as the foundation upon which the conductive traces and components are mounted. It provides mechanical support and electrical insulation for the circuitry and temperature resistance.

Common base materials include FR-4: It consists of woven glass fabric impregnated with epoxy resin. FR- 4 offers excellent electrical insulation properties, good mechanical strength, and high temperature resistance, making it suitable for automotive applications like ABS systems)

Polyimide (PI): Polyimide prepregs, often referred to as "kapton," are known for their exceptional thermal stability and resistance to high temperatures. Polyimide materials can withstand temperatures well beyond those encountered in typical automotive environments, making them suitable for ABS PCBs operating in extreme conditions.

BT (Bismaleimide Triazine): BT resin-based prepregs offer high thermal stability, low dielectric constant, and excellent mechanical properties. These characteristics make BT prepregs suitable for high- frequency applications and environments with elevated temperatures, such as those found in automotive electronics.

Base copper thickness: It is the conductive material for routing electrical signal and power throughout of the PCB. Its value could be 35µm, 70µm, or 105 µm. The thickness value for outer and inner layer can be different or equal, depending on its requirement.

Check for requirements like Leaser drill requirement: Requirement of microvias like Blind vias, buried vias; Thermal cycle requirement; Minimum line width and spacing requirement; Z-axis routed PCB; Minimum Tg of base material; TMA (Thermos mechanical Analysis); DSC (Differential Scanning Calorimetry).

Check points for analyzing the design before sending it for manufacturing.

Component outline to pad spacing: This is the clearance between the component pads to component outline. For manufacturing purpose this clearance should be the 300 μm according to IPC standard.

Component spacing: This is clearance between component outline to another component outline. And its minimum value should be 400 μm according to IPC standard.

Component pad to pad spacing: This is the clearance between the component pad to another component pad and it should be 300 μm according to IPC standard

Component AOI: Component AOI (Automated Optical Inspection) spacing in PCB (Printed Circuit Board) manufacturing refers to the minimum distance required between components on the board to allow the AOI system to effectively inspect each component. This spacing ensures that the inspection equipment can accurately analyze and detect any defects or misalignments in the soldering, placement, or orientation of the components. The specific spacing requirements may vary depending on the capabilities of the AOI system and the size and type of components being used.

Copper in keep out area: This is the copper trace width which is connected to an IC pad shall not increase overall width of the pad.

Incorrect / missing mask clearance for SMD pad: It is the clearance between the SMD pads and solder mask. The minimum clearance is required 75 μm according to IPC standard.

Incorrect / missing mask clearance for PTH drill: It is the clearance between the PTH (Plated through hole) and solder mask. The minimum clearance is required 125 μm according to IPC standard.

Incorrect / missing mask clearance for NPTH drill: It is the clearance between the NPTH (Non Plated through hole) and solder mask. The minimum clearance is required 100 μm according to IPC standard.

Different net spacing: This the spacing error between two different potential.

Same net spacing: This the spacing error between same potential.

NPTH to Rout: This is the distance between the NPTH (Non plated through hole) and rout (copper). The minimum space requirement for outer layer (top and bottom) and Inner layer is different.

PTH to Rout: This is the distance between the (Plated through hole) PTH and rout (copper).

Drill stub: This is the non-connected drills in the board. Preferred to remove drill stub throughout the board.

Copper trace stub: These are the non-connected copper traces. Preferred to remove copper traces stub throughout the board.

Improvable routing: These are the copper traces with 90- degree bend which occupies more space in the board so preferred to give 45 degree of bending throughout the board.

PTH Annular ring: This is the surrounding copper around the plated through hole. This value different for inner and outer layer. It also depends on base copper thickness value of the board.

Copper on NPTH: It is preferred to no copper on NPTH (Non plated though hole).

Missing pad for PTH and vias: It is preferred to have copper pad for all the plated through hole and vias on the board.

Conductor width: This is the minimum value 0.1mm of the copper trace should be followed for good conductivity. And the clearance between the two trace is 0.13mm. These factors depend on the impedance value for the different layer of the board.

Incorrect drill to drill distance for plugged vias: This is the minimum distance between one centers of drill to another drill. Preferred to keep 0.4 mm. The closed vias need plugging.

Via plugging: Plugging is done only for the outer layer of the PCB. It is a process in which vias are filled completely with resin or closed with solder mask. This is different from via tenting where resin/solder mask doesn't fill the via hole but just provide a covering.

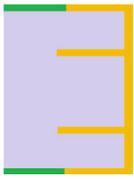


Fig: Open via

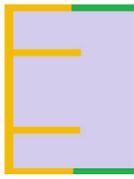
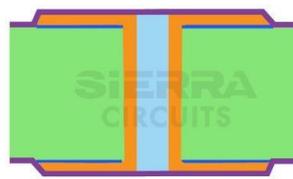


Fig: Via plugging



Missing drill: Drill must be provided for all through holes components.

Acute angle: Preferred to remove all the acute angle found in the board.



Fig: Acute angle in PCB

Coverage: It is the clearance between solder mask clearance and copper pad. The minimum clearance is required 0.075 it vary according to the design of the board like HDI (High density interconnect), Power board.

Thermal via: All power MOSFET or any microcontroller should not have via on pin as closed holes process. Even blind vias should not found on the pin.

Alignment of RC component: The component should be placed in zigzag patent to avoid mistake in AOI inspection.

Need split plane for solder paste layer: The splitting is required for solder paste layer to avoid the risk of soldering issue. There should be proper spacing between the solder paste to avoid heat dissipation.

Solder Paste related check: Check for paste missing area and small paste area throughout the board.

Missing thermal clearance: Thermal relief should be provided around all solder pad areas.

Restriction for selective soldering: Preferred to maintain 6mm clearance between SMD components outline to connector hole or any through holes component drill on soldering side for selective soldering. NPTH (Non plated through hole) should not be in selective soldering or dip soldering area.

Position of fiducial marking: It is a reference marks for proper recognition and positional alignment of the panel / PCB (Printed Circuit Board) in the solder paste printer, placement and other SMT machines.

DMC (Data matrix code) Marking: It is required for product traceability & PCB supplier traceability.

Tombstone error: A tombstone error in PCB (Printed Circuit Board) assembly occurs when one end of a surface- mounted component lifts off the board during soldering, resembling a tombstone. It results from unequal soldering forces or misalignment, causing electrical discontinuity and malfunctioning of the circuit.

It is required to maintain the trace ration of 300% to avoid tombstone effect in chip component for SMD soldering process.

Test point to test point distance: This the distance between one test point center to another test point center. Preferred to keep 2mm it may vary.

Test point to component distance: It is the spacing between test point to component outline should be as per component height.

Incorrect mask clearance for test point: It is the solder mask clearance for the test point. The minimum clearance should be 75 μm according to IPC standard.

Solder pastes on test point: Basically solder paste layer for test point is required only for OSP PCB surface finishing for good contact of nail.

Solder mask dam: A solder mask dam in PCB (Printed Circuit Board) manufacturing refers to a protective layer of material applied over copper traces to prevent unintentional solder bridges during assembly. It forms a barrier between adjacent solder pads, ensuring proper electrical isolation and preventing short circuits.

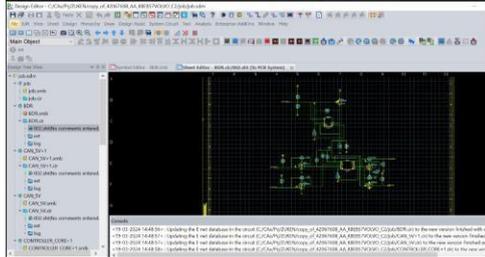
Preferred solder mask dam between pads 0.1mm. If needed, then solder mask clearance can be resized to maintain the solder dam 0.1mm.

BGA (Ball Grid Array) with component on opposite side: In double sided PCB (Printed Circuit Board) and in Multilayered PCB (Printed Circuit Board) if BGA (Ball Grid Array) is present then it is preferred to remove all the component from the opposite side of the BGA (Ball Grid Array). As BGA (Ball Grid Array) balls cannot be checked with X-ray inspection because 100% covered by another component.

Check point before Final package release for manufacturing.

Schematic DRC (Design Rule Check): To ensure that the design meets specific requirement regarding electrical, mechanical, and manufacturing consideration.

DRC (Design Rule Check) is important to list out the missing or any new component from the customer design. Schematic is required in BOM creation.



Parts Load: If any DRC error found the missing or new parts is Re-loaded in the library to get error free DRC.

Forward Annotation:

Load component position: All the components are load which is placed outside of the board design.

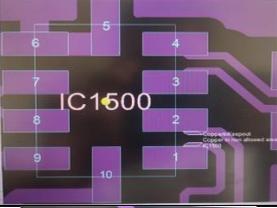
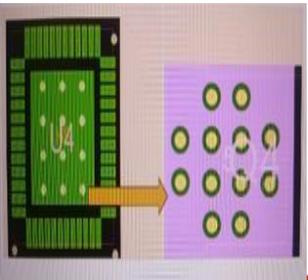
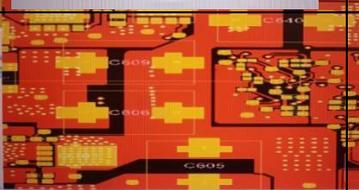
Saving component position: All the components are placed inside the board which is according to the customer design.

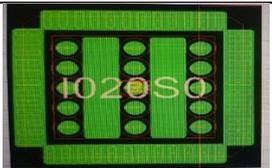
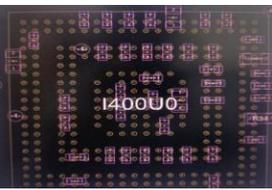
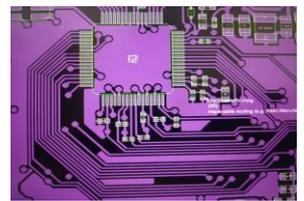
IDF Import:

Gerber import: Gerber file is created which consist of all the information related to routing in PCB (Printed Circuit Board). Whose output is called as Manufacturing package.

ODB (open data base) import: This ODB file is consist of X-Y Co-ordinated of the components. Which helps in creation of assembly drawing of the PCB (Printed Circuit Board).

Panel creation: Output Automation:

Design Check	Observation	Remark
Copper in keep-out		Routing trace connected to an IC pin shall not increase the overall width of the pad.
Improvable routing		Prefer to avoid 90-degree routing from the board.
Copper on NPTH (non-plated through hole)		Preferred to remove unwanted copper from all NPTH throughout the board.
Drill to drill distance for all plugged/closed vias		Minimum distance between two drills should be 400 um.
Missing pad for PTH & via		Provide copper pad for all PTH & via throughout the board.
Incorrect mask clearance for tab MOSFET & center pad vias		Preferred to remove the solder mask on each open hole of the MOSFET tab center pad to permit the plugin adhesion
Need split plane for solder paste layer		Preferred to provide split plane for the solder paste layer to avoid the risk of soldering issue.
Thermal via error		All power MOSFET should not have via on power pin as closed hole process
Alignments of RC component		
Restriction for selective soldering		Preferred to maintain 6mm clearance between the SMD component outline to

		connector hole or any through hole component drill on soldering side for the selective soldering.
Shape and position of fiducial marks.		Minimum of 3 fiducial / reference mark is preferred for proper recognition and position alignment of the panel/PCB.
Test point to test point distance		Spacing between one test point center to another test point center as 2.1mm throughout the board.
Test point to component distance		Spacing between test point to component outline should be as per height of the component.
Incorrect mask clearance for the test point.		Preferred to provide 50um min for the all test point though out the board.
Solder paste on test point		Solder paste on test point
Solder mask dam		Preferred solder mask dam between pads should be of 100um
BGA with component on opposite side		Preferred to clean design by removing the component opposed to the BGA as BGA balls cannot be checked with the X-ray inspection because of 100% coverage by another component.
Conductor width		Copper trace width or plane width should be according to Impedance, base cu thickness.

V. RESULTS

Sl. No.	Additional Questions Not Addressed Above	Requirement (P-Proposed, R-Remark, G-Guideline)	Customer feedback	Agreement
81	Component spacing ADI	P- Please correct the attached error list in respective sheet according to 'Space' under the label 'Distance formula: F.T.S.A x n'	OK, Customer will correct and attach updated info. Labels to be made by Customer	
82	Component spacing ADI	P- Preferred to follow the Continental standard guidelines. Customer Part No. Range of Board Part - Range with 47' open in the center. Range of Board part and PCB Clearance from Copper part. P- Range of Board Part - Range of Copper part. P- Range of Board Part - Range of Copper part. P- The PCB reduction for Free parts Components of Design will be provided.	Required Components cannot be investigated by other methods (e.g. C.T)	
83	Component spacing ADI	P- Preferred basic clearance for 100µm. To be reviewed according to Continental guidelines. Please correct the errors brought to the board. Case 2 - Name to be corrected by Customer.	Customer will correct and attach updated info. Labels to be made by Customer	
84	Missing - Incorrect mask clearance for PCB part	P- Preferred basic clearance for 100µm. Please correct the errors brought to the board. Case 1 - Can be corrected by PCB Suppliers/Manufacturers? Case 2 - Name to be corrected by Customer.	Required Components cannot be investigated by other methods (e.g. C.T)	
85	Drill Size	P- Preferred to review the drill Size through the board.	OK, error will be corrected	
86	Component Check (Issues) : Example	P- To confirm to check All Components solder issues in Cont. CAD for due to safety margin (Component + safety margin) around the component. Case 1 - Please confirm the Components guidelines with Continental Guidelines given in attachment page. Case 2 - Please check the CDR file. Extra margin of component can be provided.	Required Components cannot be investigated by other methods (e.g. C.T)	
87	Component spacing	P- Preferred to improve the routing (Error list is provided in attachment sheet)	Required Components cannot be investigated by other methods (e.g. C.T)	
88	Component spacing	P- Minimum annular ring for inner layer should be 15µm. Please correct the errors according to Continental guidelines. Case 1 - Annular Ring (Outer layer) = 15µm. Case 2 - Annular Ring (Inner layer) = 17µm. Case 3 - Annular Ring (Outer layer) = 20µm. Case 4 - Annular Ring (Inner layer) = 17µm.	Required Components cannot be investigated by other methods (e.g. C.T)	
89	Drill Size (Open area - Not Error)	Remark: Please confirm to give all the tolerances?	Required Components cannot be investigated by other methods (e.g. C.T)	
90	Missing drill	P- Please provide the drill for all Through-hole components. Need more detail on this point. Whether it has some standard?	Required Components cannot be investigated by other methods (e.g. C.T)	
91	Coverage: Exposed Copper / Connect Symbol	Remark: These are the issues with the request to the Customer modification can be done by suppliers. (See Customer confirmation)	Required Components cannot be investigated by other methods (e.g. C.T)	
92	Same net spacing	Remark: These are the issues with the request to the Customer modification can be done by suppliers. (See Customer confirmation)	Required Components cannot be investigated by other methods (e.g. C.T)	
93	Drill	Remark: These are the issues with the request to the Customer modification can be done by suppliers. (See Customer confirmation)	Required Components cannot be investigated by other methods (e.g. C.T)	
94	Alignment of IC component	P- Please follow the Continental standard guidelines given in attachment.	Required Components cannot be investigated by other methods (e.g. C.T)	
95	Restrictions for selective soldering	P- Please maintain the 2 mm clearance between SMD components relative to Connector hole or any through-hole component drill on soldering pads for selective soldering. NPTH should be in selective soldering or Dip soldering area.	Customer request provided as 0.5 mm. Please correct the errors brought to the board. Case 1 - Name to be corrected by Customer. Case 2 - Name to be corrected by Customer.	
96	Tolerance error	P- It is required to maintain the face ratio of 50% to avoid Tolerance effect of Dip Components for SMD soldering Process. P- correct the signal.	Required Components cannot be investigated by other methods (e.g. C.T)	
97	PCB Masking	P- It is required for product feasibility & PCB supplier feasibility. Position & Size can be confirmed after sourcing Process. Please correct.	Required Components cannot be investigated by other methods (e.g. C.T)	
98	PTH & NPTH Tolerance	P- PTH & NPTH preferred balance should be as per Continental Guidelines. P- spacing between One net point center to another net point center as 2.1 mm. Cont. Guidelines follow. Through hole board - Please correct the errors according to the Continental guidelines.	Required Components cannot be investigated by other methods (e.g. C.T)	
99	Tempnet to Tempnet Distance	Remark: Need correct CDR file to do the detailed analysis of Top point to other Top point spacing and distance between net point to other height components. To make it 70µm suitable for auto-assembly in CAD. The part name that needs to be used in CDR file should be 'temp_netpoint'. net package name must be 'PTH_net' or 'NPTH_net'. Please refer the below snap shot. Case 1 - Please correct the attached guidelines. Case 2 - Please check the CDR file.	Required Components cannot be investigated by other methods (e.g. C.T)	
100	Solder mask Drill	P- Preferred Solder mask Drill between Pads should be 100µm. Please correct through-hole board. I checked that Solder mask clearance can be reduced to maintain the solder size as 100µm.	Required Components cannot be investigated by other methods (e.g. C.T)	
101	PTH definition for connectors for plating process	P- Please follow the Continental standard guidelines given in attachment.	Required Components cannot be investigated by other methods (e.g. C.T)	

Figure 2: Feedback report from customer

free solder alloys under chloride-containing thin electrolyte layers International Journal of Electrochemical Science 13, 9942 (2018).

9. O.N. Kamil, O.F. Rifdi, and A.F. Che, Electrochemical migration and corrosion behaviours of SAC305 reinforced by NiO, Fe₂O₃, TiO₂ nanoparticles in NaCl solution. IOP Conference Series: Materials Science and Engineering 701, 012 (2019).
10. B.K. Liao, H. Wang, L. Kang, S. Wan, X.D. Quan, and X.K. Zhong, Electrochemical migration behavior of low-temperature-sintered Ag nanoparticle paste using water-drop method. Journal of Materials Science: Materials in Electronics 32, 5680 (2021).
11. G.Q. Lu, C.Y. Yan, Y.H. Mei, and X. Li, Dependence of electrochemical migration of sintered nanosilver on chloride. Materials Chemistry and Physics 151, 18 (2015).
12. M.S. Hong, and J.G. Kim, Method for mitigating electrochemical migration on printed circuit boards. Journal of Electronic Materials 48, 5012 (2019).
13. H.L. Huang, X.M. Guo, F.R. Bu, and G.L. Huang, Corrosion behavior of immersion silver printed circuit board copper under a thin electrolyte layer. Engineering Failure Analysis 117, 14 (2020).
14. P. Yi, K. Xiao, C.F. Dong, S.W. Zou, and X.G. Li, Effects of mould on electrochemical migration behaviour of immersion silver finished printed circuit board. Bioelectrochemistry 119, 203 (2017).
15. K. Xiao, P. Yi, C.F. Dong, S.W. Zou, and X.G. Li, Role of mold in electrochemical migration of copper-clad laminate and electroless nickel/immersion gold printed circuit boards. Materials Letters. 210, 283 (2018).
16. P. Yi, K. Xiao, K.K. Ding, C.F. Dong, and X.G. Li, Electrochemical migration behavior of copper-clad laminate and electroless nickel/immersion gold printed circuit boards under thin electrolyte layers. Materials 10, 137 (2018).
17. X.F. He, M.H. Azarian, and M.G. Pecht, Evaluation of electrochemical migration on printed circuit boards with lead-free and tin-lead solder. Journal of Electronic Materials. 40, 1921 (2011).
18. B.I. Noh, J.W. Yoon, W.S. Hong, and S.B. Jung, Evaluation of electrochemical migration on flexible printed circuit boards with different surface finishes. Journal of Electronic Materials 38, 902 (2019).
19. W.S. Hong, and C. Oh, Lifetime prediction of electrochemical ion migration with various surface finishes of printed circuit boards. Journal of Electronic Materials 49, 48 (2020).
20. D. Bušek, K. Dušek, J. Kulhavý. Dendritic growth and its dependence on various conditions. 2018 41st International Spring Seminar on Electronics Technology (ISSE). 1 (2018).