

Study on PCB Designing Problems and their Solutions

Vishal Anand

Dept. of Electrical Engineering
National Institute of Technology,
Raipur, India

<https://orcid.org/0000-0002-6707-5387>

Varsha Singh

Dept. of Electrical Engineering
National Institute of Technology,
Raipur, India

<https://orcid.org/0000-0001-5835-5400>

Vinay Kumar Ladwal

Dept. of EEE
Birla Institute of Technology, Mesra,
Ranchi, India

<https://orcid.org/0000-0003-2471-019X>

Abstract—PCB designing is important for all kinds of power converters especially for usage in industrial applications. The parameters impacting the design and robustness of PCB are strain inductance associated with the circuit. For the design of control circuits, the primary requirement is precision. In order to make the PCB design reliable, reduction of strain inductance along with maintaining the simplicity, modularity, compactness and better cooling management is required. These parameters are key for designing PCBs with precision, performance, attainment of low power consumption and high-power converters at fast manufacturing speed. This paper also describes the methods to reduce signal lags which can be attained by equalizing trace lengths.

Keywords— PCB designing, vias, trace width, clearance, grounding, thermal considerations

I. INTRODUCTION

In the modern world, the power requirement is increasing exponentially. This creates space for research in circuitry where high current and high voltage can be handled efficiently without much interference, maintaining the reliability, precision, and reduction in the losses. EMI (Electromagnetic Interference) and EBG (Electromagnetic Bandgap) are important aspects discussed in [1] which impact the radiation, parasitic inductance of the package. This is studied by PEEC (partial element equivalent circuit) technique, here spate layers are used for shielding that is not economically fusible. Which can be replaced by an external metallic enclose that helps in EMI and EBG shielding. In [2] synchronous buck converter is studied using an electrothermal circuit design. Which uses the thermoelectric cooler to reduce the temperature by 3-5 degree C for the same electro thermal trace. The coupling between the inductor is complex and time-dependent, which is its drawback but losses associated with the device are optimized properly which adds an advantage in the device. In [3] parasitic effect of isolated resonance converter is described with proper isolation level between LV and HV side. This also describes parasitic capacitance and optimization of traces for high power density components which work at high frequency in the range of MHz's. This doesn't describes the continuous current mode operation. The windings can be made on a multi-layer PCB for motor designing. The major disadvantage of this technique is there is no proper core design that can be embedded for flux concentration, flux losses are considerably high and production of such design is highly expensive, difficult and time-consuming [4]. In [5] the gate driver for SiC power module decoupled capacitors and dc-link capacitors are described with a reduction in dV/dT and di/dT stress. This also focuses on power loop optimization and thermal management of high-power density

SiC module. In [6] high-frequency transformer is designed for a bi-directional onboard charging, considering loss and footprint optimization. This also focuses on the comparison between different core loss densities from different manufacturers. In [7] describes the cancellation of parasitic inductance for shunt filter capacitors.

This paper is organized as the placement of the component described in Section II. Routing and traces are described in Section III. Thermal management for PCB is described in Section IV followed by a conclusion in Section V

II. CONSIDERATION TAKEN WHILE PLACING COMPONENTS

Designing the schematic and producing netlist is pre place and route task. The next most important step is termed as place i.e. placing the footprints.

1. Placement of footprints is done in a way such that minimum path resistance and length is achieved between two connecting points on a PCB. For this rotating the footprint and flipping are done to adjust or move it to another side of the board respectively.
2. One important point to be taken care of is separating the components or spreading them on the basis of the number of ground planes present on the board. To assure proper grounding and isolation between ground planes.

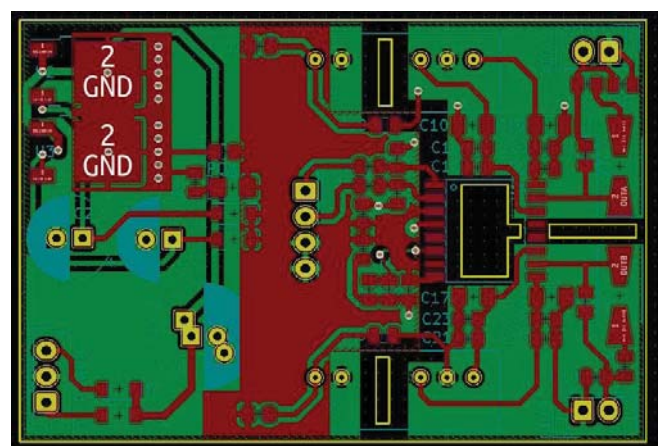


Fig 1. Components are separated on number of ground planes (green color polygon)

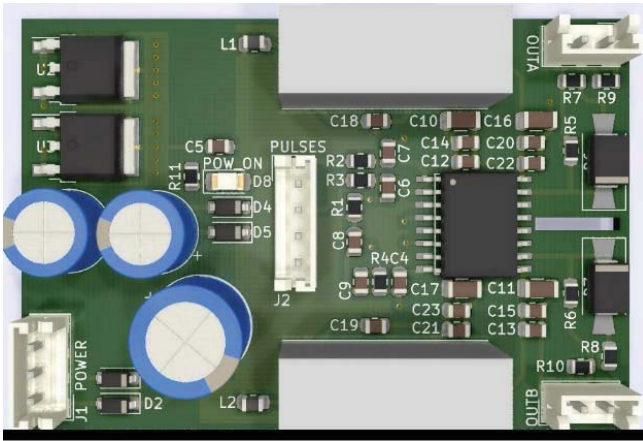


Fig 2. 3D view of the SiC MOSFET gate driver

Fig 1 and Fig 2 illustrate the importance of placing components in such a way that the trace/routing work is easy and all the ground planes are nicely isolated. In this design, there are three separate ground planes isolated from each other. The biggest ground plane on the left side is for the primary supply of the SiC gate driver which is common with the input signals. There is two separate ground planes on the right side for each SiC MOSFET. The second major consideration that is made in the design is high voltage side is kept on one side of the PCB to achieve good isolation between the 5V and 12V traces. The four cut-outs made in PCB are the last consideration for the design that is administered to achieve excellent isolation between all possible ground planes and traces voltages. The capacitors are placed in a sequence as they have an individual common rail. Such placement makes tracing easier as well as a minimum resistance path is achieved. Thus, it is an effective design eliminating the chance of stray capacitance and parasitic inductance.

3. When high voltage components and high current carrying components are placed, make sure to leave enough clearance between them for proper isolation in case of high voltage components and to have enough space for wide traces in case of high current-carrying components. Also done to make sure proper cooling in case of high current-carrying components.

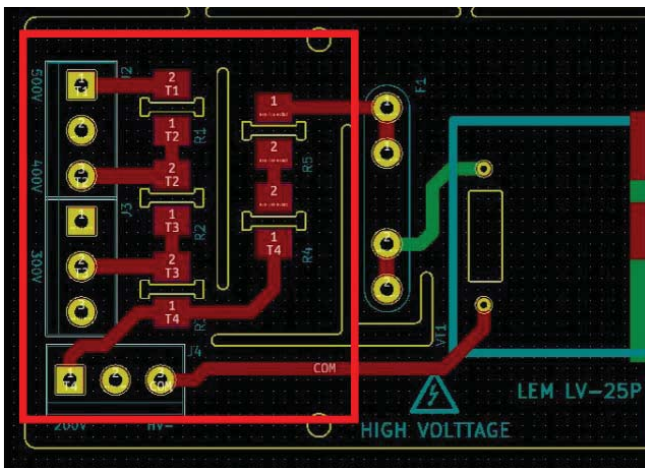


Fig 3. High Voltage isolation by adding cutouts on PCB (100V separation)

Fig 3 illustrates the design of a voltage sensing circuit where the maximum drop across the individual SMD resistors is 100V, which is labeled in the red box. The cut-outs made in the design ensures isolation between two terminals as well as isolation between individual resistors pads. This method prevents accidental sparks that may occur bridging the contacts.

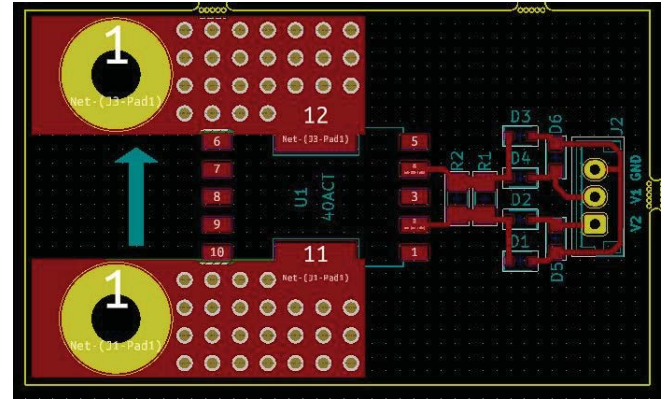


Fig 4. High Current traces design (40A)

Fig 4 illustrates the high current trace. It is done to ensure the width of the trace for a particular design. It is calculated based on the maximum current that can flow through the designed circuit. An important observation that can be seen in the design is a lot of holes present on the traces. This method helps to achieve passive cooling through the ambient airflow.

4. When its turn-off RF components where they have their antenna's to be traced on the PCB, define a no routing zone wherever the antenna is placed or consider not tracing across that or defining any ground plane on the other-side/near to the antenna design.
5. In case of SMT/THT parts that heat up place them in a way to provide direct connection of their ground planes for heat dissipation to prevent melting solder while operating conditions. Add uncovered vias for cooling to the plane if required.

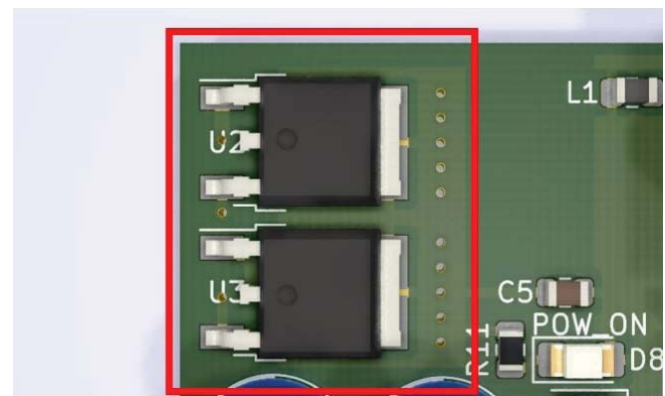


Fig 5. Adding small heatsink on PCB for SMT components

6. Parts that dissipate heat and requires a heat-sink should be placed towards the edges of the board so that heat-sink stays to one side of PCB, finally, at the end of the assembly they can be attached to the heat-

sink easily and heat-sink can be screwed with the board as well to have robust design. Selecting a proper footprint matters the most for proper alignment of part with the heat sink.

Fig 5 illustrates that the addition of a protective heatsink for the SMD components while the operation. In design two LM7805 SMD (TO-252-2package) produce a considerable amount of heat during operation. These linear regulators dissipate some heat when they are not operation. While in operation the heat dissipation increases significantly very high and chances of melting solder increases. In order to prevent the melting of solder under the linear regulator packages, the path is created to dump heat dissipated to the largest plane as soon as possible.

7. Try to keep the maximum number of components on the same side of PCB to ensure fast and easy production. This step has importance on the basis of manufacturing process whether the designed PCB is being produced in mass and assembled using dots, on the other hand, the PCB is hand soldered. This can be violated only when PCB is densely populated with different sized parts and has a restricted dimension or area to work with.
8. Finally, fine-tuning is always the best option. Adjusting components and packing them as close as possible is done when they have a common trace going through them.

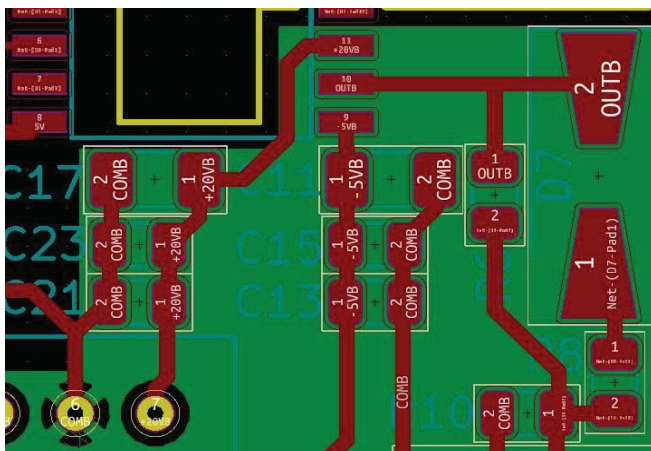


Fig 6. Common component placed in one side

Fig 6 illustrates the placement of components next to each other that shares a common rail for easy routing and least path length.

III. CONSIDERATION TAKEN WHILE ROUTING

As proper placement of footprints is finished, the next step in PCB designing is called routing i.e. tracing out copper tracks between two points to establish a connection between them. Routing is executed on the basis of whether the trace is for just signals or is for carrying a considerable amount of power. For an effective routing, following steps are to be followed:

1. The width of PCB is an important consideration for the low voltage trace. The maximum amount of current that can be drawn from the PCB is key parameter and that may lead to melting of solder when the temperature rises above critical value. In order to decide this critical value, it is necessary to decide consider the trace using a differential pair. The expression for trace width calculation are as follows:

Internal Trace:

$$I = 0.024 * dT^{0.44} * A^{0.725}$$

External Trace:

$$I = 0.048 * dT^{0.44} * A^{0.725}$$

where:

I = max current in Amps

dT = temp rise above ambient in °C.

A = trace width * thickness of the copper layer in mils

Considering *IPC-2221(A)*

Here A (area) is calculated by width multiplied by the thickness of the copper layer.

2. In case of design of trace for high potential difference it is important to take care of minimum separation distance. The voltage isolation between two traces for high voltage isolation must follow a minimum separation distance (clearance) that is decided by calculation of potential difference between them.

$$\text{Clearance in Inches} = 0.023'' + (0.0002'' \times V)$$

$$1 \text{ mil} = \frac{1}{1000} \text{ inches}$$

3. For high speed and/or precision signal traces equalizing the length of each traces in a set of 'n' number of channel traces is done to attain equal physical specification for each trace. Hence, assuring there will be no considerable delay or drop in the path.
4. Differential pairs are mostly low power signal traces where the primary requirement is to have a minimum and identical trace length between sending and receiving end to get quality signal transfer capabilities. When there are a large number of differential pairs, for example, the connections of an LCD controller and HDMI decoder on a PCB, setting the length of each connection to be equal is required. When the distance between two such points is less and the length of trace required is more, it is necessary to make curved and or zigzag-shaped traces to match the length of each track. Usually done in high-speed digital circuits or even in mixed-signal circuits design.

It is assumed $C=0.725$

$$b=0.44$$

$$I=1 \text{ Ampere}$$

$$dT=10^{\circ}\text{C}$$

$$H=1 \text{ ounce}=1.4 \text{ mils}$$

$$Area (mils^2) = \frac{1 \text{ Ampere}}{(K \times dT(^{\circ}C)^2)^{\frac{1}{c}}}$$

$$= \frac{1}{(0.048 \times 10^{0.44})^{\frac{1}{0.725}}} = 16.296$$

But, $Area = H \times W$ $\forall H=1.4$
Thus, $W = 11.64 \text{ mils} = 0.3 \text{ mm}$

Clearance Calculation

$$\begin{aligned} \text{Spacing} &= 0.6 + (0.005 \times V_{pk}) \\ &= 0.6 + (0.005 \times 500V) \\ &= 0.6 + 2.5 = 3.1 \text{ mm} = 0.123 \text{ inches} \end{aligned}$$

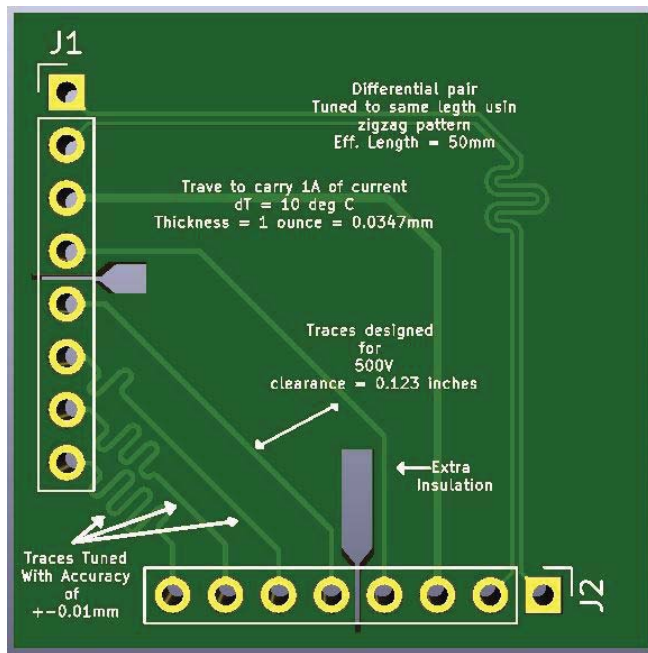


Fig. 7. Fine tuning techniques

In fig the of high-speed digital circuits where time is an important constraint, it is necessary to have equal trace length for all the available connections on a bus. This helps to achieve no delay in signal propagation between the individual connection on the bus. Example: 8-bit bus. All buses must have data reaching at the same time. This is necessary to avoid delay and false triggering. For voltage isolation of 500V, the calculated clearance between the trace is 0.123 inches. When the available space for clearance is less, cutouts are added to achieve isolations on PCB design. For carrying one ampere current the width of the trace is coming to be 0.0347mm after calculation. When a differential pair tracing is done, two important consideration is taken care. Firstly, the individual length of both traces should be the same. Secondly, traces should be placed with a separation of at least 7 mils (0.187mm).

IV. THERMAL CONSIDERATION

Thermal relief and proper heat sinking techniques in PCB designing.

1. When in a footprint the pad that is used to connect the components to the PCB by soldering if it doesn't require high current to flow through that connection. In such cases, the pad is connected to a plane using a thin cross-shaped connection the techniques are

called as thermal relieving. Helps in final production as it requires less time to melt the solder at the thermally relived point.

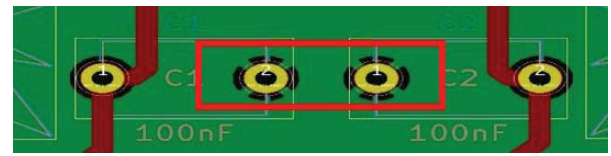


Fig 8. Thermal relieving footprint.

Fig 8 discusses a technique that helps in fast soldering which is helpful in the production of PCB. It can be observed that the soldering pads inside the red box are connected to the ground using the cross-shaped connection. This technique is known as thermal relieving. (The cross-shaped connection helps in fast soldering as compared to solid connection points where heat dissipation will be more as compared to this technique)

2. In case of an SMD component on PCB tends to produce reasonable heat is present. The heat-sink part of the component is connected to the large plane on the PCB to dissipate heat as much as possible so that during operation it doesn't melt the solder and create fault by disconnecting. In extreme cases, largely uncovered vias are added to the plane to increase cooling. In the end, adding an external heat-sink is never a bad option.
3. For high current carrying tracks in order to protect PCB from heating up there are many ways used in multiple ways stated below:
 - i. Adding uncovered vias to cool the PCB naturally (passive cooling).

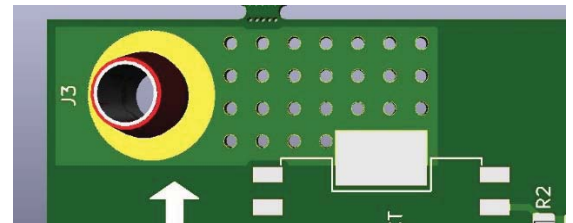


Fig 9. Trace cooling by adding through vias.

Fig 9 shows passive cooling attained by using holes.

- ii. Copper thickness of the PCB is increased to increase the cross-sectional area of the conductor usually measured in ounces of copper deposited.
- iii. As the first two steps may become a little costlier for the production point of view. In most cases on such tracks, the solder mask is removed later no while assembly a flood of solder is poured over the track to increase the cross-sectional area.

4. Last few steps for turning PCB

- i. To increase voltage insulation considering the separation added PCB cuts are made to increase the breakdown voltage required to establish an arc between the tracks. Finally increasing safety.

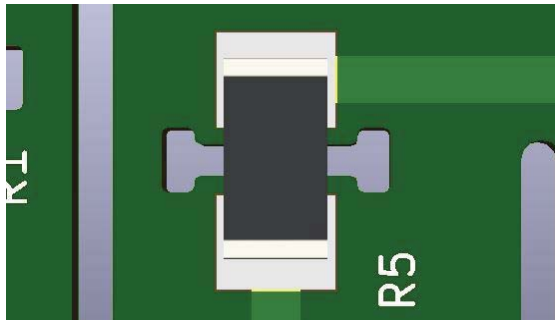


Fig 10 High voltage isolation (100V)

Fig 10 describes the proper trace width for achieving high voltage isolation between thermal pads.

- ii. For faster production and reducing copper wastage in etching, solid planes are used that are either defined as positive or ground on the PCB for proper power distribution. In such cases, if the weight of PCB is a factor to be considered planes are kept in repeating patterns to reduce the weight of PCB.

All the design of PCB is carried out in KiCAD open-source platform.

V. CONCLUSION

The design rules if followed properly helps in eliminating unnecessary stray capacitance, and parasitic inductance. Proper isolation and active noise cancellation can be attained by proper placement of components and cut-outs. Symmetrical placement of components helps in eliminating unnecessary loop inductance by achieving least path resistance. Thus, the PCB design is a very crucial aspect in dealing with high power density circuits and making effective usage of design.

REFERENCES

- [1] P. Zuo, Y. Li, Y. Xu, H. Zheng and E. Li, "Near-Field Radiation Estimation and Its Reduction Using a Novel EBG for PCB," in *IEEE Transactions on Components, Packaging and Manufacturing Technology*, vol. 9, no. 2, pp. 329-335, Feb. 2019.
- [2] E. M. Dede *et al.*, "Electrothermal Circuit Design With Heat Flow Control—Synchronous Buck Converter Case Study," in *IEEE Transactions on Components, Packaging and Manufacturing Technology*, vol. 8, no. 2, pp. 226-235, Feb. 2018.
- [3] X. Zhao, C. Chen, J. Lai and O. Yu, "Circuit Design Considerations for Reducing Parasitic Effects on GaN-Based 1-MHz High-Power-Density High-Step-Up/Down Isolated Resonant Converters," in *IEEE Journal of Emerging and Selected Topics in Power Electronics*, vol. 7, no. 2, pp. 695-705, June 2019.
- [4] F. Marignetti, G. Volpe, S. M. Mirimani and C. Cecati, "Electromagnetic Design and Modeling of a Two-Phase Axial-Flux Printed Circuit Board Motor," in *IEEE Transactions on Industrial Electronics*, vol. 65, no. 1, pp. 67-76, Jan. 2018.
- [5] C. Chen, Z. Huang, L. Chen, Y. Tan, Y. Kang and F. Luo, "Flexible PCB-Based 3-D Integrated SiC Half-Bridge Power Module With Three-Sided Cooling Using Ultralow Inductive Hybrid Packaging Structure," in *IEEE Transactions on Power Electronics*, vol. 34, no. 6, pp. 5579-5593, June 2019.
- [6] B. Li, Q. Li and F. C. Lee, "High-Frequency PCB Winding Transformer With Integrated Inductors for a Bi-Directional Resonant Converter," in *IEEE Transactions on Power Electronics*, vol. 34, no. 7, pp. 6123-6135, July 2019.
- [7] A. J. McDowell and T. H. Hubing, "A Compact Implementation of Parasitic Inductance Cancellation for Shunt Capacitor Filters on Multilayer PCBs," in *IEEE Transactions on Electromagnetic Compatibility*, vol. 57, no. 2, pp. 257-263, April 2015.
- [8] V. Anand, S. Singh, A. Alam and A. Gautam, "State-Space Stator-Flux Model of 3-phase Induction Motor using LabView," *2018 International Conference on Current Trends towards Converging Technologies (ICCTCT)*, Coimbatore, 2018, pp. 1-5.
- [9] V. Anand, A. Alam, S. Singh and A. Gautam, "Hardware Implementation of Real-Time Flux, Torque, Sensorless Speed Estimation of a 3KW Three-Phase Induction Machine using LabVIEW," *2018 International Conference on Current Trends towards Converging Technologies (ICCTCT)*, Coimbatore, 2018, pp. 1-6.
- [10] V. Anand, V. Singh and V. Anand, "Design and Analysis of MLI with reduced number of switches and PV panel as a source," *2019 IEEE International Conference on Sustainable Energy Technologies and Systems (ICSETS)*, Bhubaneswar, India, 2019, pp. 308-312.
- [11] Y. Wu, Z. Hao, M. Tao, X. Wang and J. Hong, "A Simple and Accurate Method for Extracting Super Wideband Electrical Properties of the Printed Circuit Board," in *IEEE Access*, vol. 7, pp. 57321-57331, 2019.
- [12] L. Taylor, X. Margueron, Y. Le Menach and P. Le Moigne, "Numerical modelling of PCB planar inductors: impact of 3D modelling on high-frequency copper loss evaluation," in *IET Power Electronics*, vol. 10, no. 14, pp. 1966-1974, 17 11 2017.
- [13] Y. S. Cao *et al.*, "Inductance Extraction for PCB Prelayout Power Integrity Using PMSR Method," in *IEEE Transactions on Electromagnetic Compatibility*, vol. 59, no. 4, pp. 1339-1346, Aug. 2017.
- [14] C. Panchal and J. Lu, "High Frequency Planar Transformer (HFPT) for Universal Contact-Less Battery Charging Platform," in *IEEE Transactions on Magnetics*, vol. 47, no. 10, pp. 2764-2767, Oct. 2011.
- [15] G. Wu, Y. Chen and T. Wu, "Design and Implementation of a Novel Hybrid Photonic Crystal Power/Ground Layer for Broadband Power Noise Suppression," in *IEEE Transactions on Advanced Packaging*, vol. 33, no. 1, pp. 206-211, Feb. 2010.