IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

Thermal Analysis and PCB Layout Optimization of a DC-DC Buck Converter

Keerthana G, Subiksha S, Nithya Sri A, Saruchandhrikha H, Sri Vardhini T
Department of ECE
K.Ramakrishnan College of Technology (Anna University) Tiruchirappalli, India
kavithaa.ece@krct.ac.in, subiksha.s0907@gmail.com, nithyasri0489@gmail.com
saruchandhrikhah@gmail.com, tsrivardhini16706@gmail.com

Abstract: *A small DC-DC buck converter printed circuit board PCB is. Carefully improved to make it better at handling heat using a computer simulation called Finite Element Method FEM. As electronic systems need to be more reliable and have power in a small space keeping heat under control in power converters is a big problem. Old ways of designing usually find heat problems after making the real PCB, which causes hot areas, expensive changes and slow progress. Also regular FR-4 materials do not let heat move well which makes it hard to spread heat without help. In this study the main parts that make heat in the power part the switching MOSFET, the freewheeling diode and the power inductor are treated as heat sources on a FR-4 board. Initial heat simulations show differences in temperature not much heat moving across the board and serious heat problems under the parts that make the most heat. To fix these problems the PCB design is improved in three ways: making bigger areas of copper to help spread heat across the board putting special holes under the hot parts to help heat move up and down and moving parts apart so they do not make each other hotter. Simulation results show that the improved design lowers the temperature on the board makes the heat more even and makes it easier for heat to move away, from the parts. By doing this in the design the converter works better and needs fewer changes after it is built.*

Keywords: DC-DC buck converter, thermal analysis, PCB layout optimization, thermal vias, finite element method (FEM), heat spreading, FR-4 PCB.

I. INTRODUCTION

DC-DC buck converters are very common in power supply systems, embedded systems and consumer electronics. They play a role in reducing higher DC voltages to lower levels that sensitive circuits can use. As the need for more efficient electronic devices grows, the size of these power modules has become much smaller. During use components like the switching MOSFET, the freewheeling diode and the power inductor produce heat. This happens because of conduction losses and switching losses. If this heat is not handled properly it can build up. Cause serious problems. Much heat can reduce the performance of components lower the efficiency of power conversion and speed up the aging process. This can shorten the life of the circuit [1] [2]. To make these small designs possible standard FR-4 printed circuit boards (PCBs) are used everywhere. These boards are chosen because they are cheap, stable and easy to make in quantities. But the FR-4 material has a thermal conductivity—about 0.3 W/m·K. This makes it hard to move heat away from hot spots. Because of this the areas under high-power components often get very hot creating hotspots [3] . Traditionally PCB design focuses mostly on performance. The layout is shaped mainly to reduce inductance and keep signals clean. Thermal issues are usually ignored until later. As a result thermal performance is checked only after the board has been built and tested. This late-stage approach means designers have to build a prototype test it find the problems and then redesign it. This cycle takes time and money. It slows down the whole process of getting a product to market. To fix this we need to shift to a way of designing. We should start thinking about heat in the design phase. Predictive thermal analysis and layout optimization can help. This work suggests a step-by-step method that uses

Copyright to IJARSCT
www.ijarsct.co.in
DOI: 10.48175/IJARSCT-40757

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 6, Issue 2, November 2026 Impact Factor: 8.2

simulations to study how heat spreads on a buck converter PCB. The goal is to improve the layout before anything is built. Key strategies include using copper pour areas to spread heat sideways adding thermal vias in strategic patterns to move heat down to inner layers or the bottom of the board and placing components carefully to avoid heat buildup in one area. Using element method (FEM) tools we can simulate the temperature, in the board and spot thermal issues before making a physical board. This kind of electro-thermal co-optimization helps create a design that runs cool stays reliable and avoids redesigns.

II. POWER MODULE DESIGN AND PCB FABRICATION PROCESS

A. DC-DC Buck Converter Design

A DC–DC buck converter is a power circuit that converts a higher DC input voltage into a lower DC output voltage in an efficient way. A DC–DC buck converter is widely used in devices, embedded systems, battery-powered applications and power supply circuits. The main components of a DC–DC buck converter are a MOSFET switch, a diode or a synchronous MOSFET, an inductor, an input capacitor, an output capacitor and a control circuit. These components work together to give an output voltage that is suitable for the connected load.

The operating principle of a DC–DC buck converter is based on high-speed switching. When the MOSFET of a DC–DC buck converter is switched ON energy moves from the input source to the inductor and to the load. When the MOSFET of a DC–DC buck converter is switched OFF the inductor keeps supplying current to the load through the freewheeling diode or through a MOSFET. The inductor. Releases energy, while the capacitors lower voltage fluctuations and keep the output smoother. The output voltage of a DC–DC buck converter is controlled by changing the duty cycle of the switching signal.

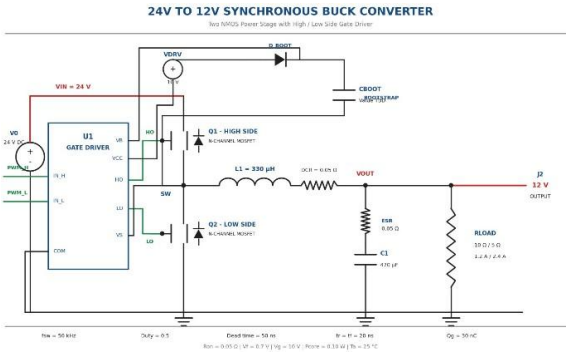


Fig 1. Circuit Diagram of DC-DC Buck Converter

In the proposed project Thermal Analysis and Optimization of a DC–DC Buck Converter in PCB a DC–DC buck converter design focuses on achieving voltage conversion while keeping heat generation low. The selection of components the PCB layout, the copper thickness and the thermal management techniques play an important role in the overall performance of a DC–DC buck converter. Excessive heat produced by switching devices and other power components can lower efficiency and reliability. Therefore thermal analysis is essential to find high-temperature regions and to better dissipate heat.

A DC–DC buck converter can be evaluated by studying the characteristics and the temperature distribution of the PCB of a DC–DC buck converter. Important parameters of a DC–DC buck converter include input voltage, output voltage, load current power losses and component temperature. By looking at these parameters suitable improvements can be made to component placement and to copper distribution. The goal is to develop a DC–DC buck converter with better thermal performance and, with efficient power conversion.

B. Selection of Power Electronics Components

Choosing the power electronic components is a crucial step when building a DC–DC buck converter. The key power electronic components are a MOSFET, a diode, an inductor, an input capacitor, an output capacitor and a control

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

circuit. Each of these power components has a clear job: they turn the input DC voltage into a smaller steady output voltage. To pick the power electronic components you must look at the required input voltage the desired output voltage, the load current the switching frequency and the power rating.

Here the MOSFET behaves like a switch that moves the energy from the input supply to the load. Choosing a MOSFET with ON-state resistance is important because it cuts down conduction losses and keeps heat low. The Schottky diode gives the inductor a path to flow when the MOSFET turns off in a buck converter. The inductor. Releases energy as it switches while the input and output capacitors smooth out voltage swings and make the supply more stable.

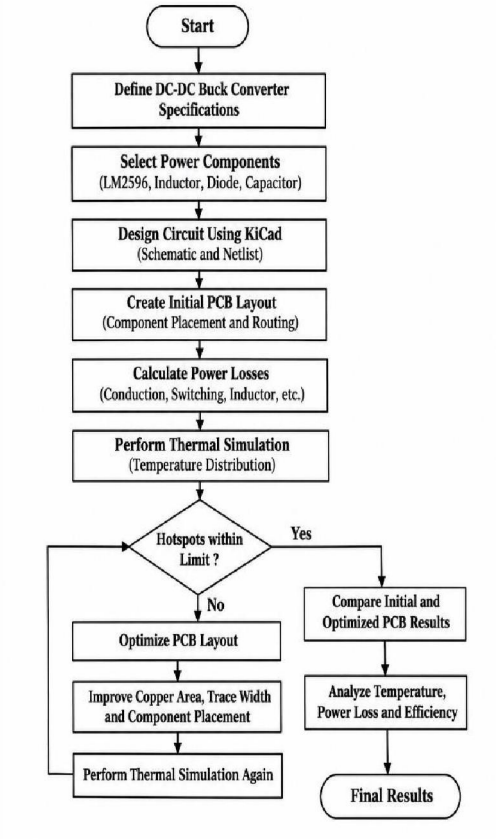


Fig. 2. Block Diagram of the Proposed System

The control circuit makes the PWM signal that tells the MOSFET when to turn on and off. The duty cycle of that PWM signal sets the output voltage when the converter is running normally. You must pick the ratings for the MOSFET, diode, inductor and capacitors so that everything stays safe and works well. Good component choice also cuts power loss. Stops the converter from overheating when it runs all the time. In the project I am doing Thermal Analysis and Optimization of a DC–DC Buck Converter on a PCB I focus on picking components that give electrical performance while staying cool. I look for parts that have the current rating, voltage rating, power dissipation and thermal behavior. By doing that I lower heat production make the PCB more reliable and set up a base, for the thermal study and improvement.

C. PCB Layout Design and Components Placement

PCB layout design is a step in building a DC–DC buck converter. PCB layout design changes how well the buck converter works, how power the buck converter loses and how heat spreads across the PCB. PCB layout design shows

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 6, Issue 2, November 2026 Impact Factor: 8.2

where the MOSFET, diode, inductor, capacitors and control circuit should be placed and how copper traces connect them.

When PCB layout design reduces parasitic inductance and resistance electromagnetic interference falls and the buck converter stays steadier. High-current parts need to be arranged so that current flows in paths. I make sure the input capacitor sits close to the switching stage so that voltage spikes from switching currents stay low. I place the MOSFET, diode and inductor carefully to shrink the high-current loop area. I also position the output capacitor near the output stage which helps reduce voltage ripple and keeps the output steadier. Placement of parts must also consider heat. Power parts such as MOSFETs and diodes can generate a lot of heat while running. I give space by using copper areas and building good thermal pathways, which spread heat across the PCB. I keep control circuits away from switching nodes when possible. I lay out signal traces so that they cut down on interference from high-current paths.

In this project PCB layout design lets us compare component placement with an arrangement. Tools such, as KiCad or Cadence Allegro help me make the layout and check trace routing and copper distribution. After improving the layout a thermal analysis can spot hotspots. Show whether the new placement really helps heat flow away.

D. Copper Layer Design and Thermal Management

Copper layer design is important for dealing with the heat that power electronic parts make. Copper moves heat well. It spreads it from areas across a bigger part of the PCB. In a DC–DC buck converter parts like MOSFETs, diodes and inductors make heat because of conduction losses switching losses and resistance in the windings. Using copper properly helps reduce spots and makes the circuit cooler.

The thickness of the copper and the width of the lines should be right for the amount of current each path has. Wider lines and bigger copper areas make resistance, which lowers power loss from current moving. Copper planes also give ways for heat to move across the board.. How much copper you need depends on the current the board has how it is used and how much space is available.Thermal vias help move heat between copper layers in a PCB.

These vias connect copper areas on layers and create more ways for heat to move from hot components to other parts of the board. Good copper pours, thermal vias and smart placement of parts can all work together to spread heat better. The design must also follow spacing rules to prevent circuits, between different parts of the circuit.

E. PCB Fabrication and Assembly Process

PCB fabrication is how we build a printed circuit board from the circuit layout we designed. It starts with drawing the circuit schematic and then creating the PCB layout using design software. When that's done we prepare the files needed for manufacturing. The choices we make—like the board material, how thick the copper's how many layers it has its size and other fabrication details—depend on the electrical and thermal needs of the buck converter.

In the factory the PCB base material is first prepared. Then the copper layers are shaped to form the traces and pads we need. For boards with than one layer the layers are carefully lined up and pressed together. Holes are filled with copper to connect the layers where needed. After that the board gets a surface finish to help with soldering and protect the copper. We also check the board to make sure the copper patterns, holes, spacing and overall quality are correct.

Next comes assembly. We put the parts onto the board. Solder them to the pads. Small surface-mount parts are often placed with solder paste then heated in a reflow oven. Larger through-hole parts might be soldered with a wave-soldering machine or by hand. After everything is attached we check for problems like solder joints, parts in the wrong direction broken components or accidental electrical connections.

For this project "Thermal Analysis and Optimization of a DC–DC Buck Converter, in PCB " paying attention to fabrication and assembly is key. It makes sure the real board matches the design we planned. Once assembled we do tests—like checking for continuity measuring voltages and running function tests—before moving to testing. We use temperature sensors or thermal imaging to find spots and see how heat spreads across the board. These results help us tweak the layout improve copper layers or add cooling to make the buck converter run cooler and more efficiently.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

III. THERMAL ANALYSIS AND OPTIMIZATION METHODOLGY

A. Heat Generation in MOSFETs, Diodes, and Inductors

I notice that heat generation is a factor for the operation of a DC–DC buck converter. During operation electrical energy is turned into heat because of conduction losses switching losses and resistance in components. The MOSFET generates heat because of its resistance and switching transitions. The diode produces heat because of its voltage drop and current flow while the inductor generates heat because of the resistance of its winding and losses in its magnetic core.

The amount of heat generated depends on the operating voltage load current, switching frequency and characteristics of the selected components. Higher current can increase conduction losses while higher switching frequency can increase switching losses in the MOSFET. Similarly an inductor with winding resistance can produce greater copper losses during operation. These losses can cause the component temperatures to rise, especially when the PCB has limited heat-dissipation capability.

Excessive heat can affect component performance reduce efficiency and shorten the operating life of devices. Therefore selecting components with electrical ratings and low power losses is essential for thermal management. Proper PCB layout, copper area and suitable cooling methods can help reduce temperature rise and improve the reliability of the buck converter.

B. Heat Tranfer and Thermal Resistance

Heat transfer is the movement of energy from a high-temperature region to a lower-temperature region. In a DC–DC buck converter PCB heat generated by power components travels through component packages, solder joints, copper pads, PCB materials and surrounding air. Heat transfer mainly occurs through conduction within materials and through convection between the PCB surface and the surrounding air. Thermal radiation may also contribute to heat transfer depending on the operating temperature and surface conditions.

Thermal resistance describes how effectively heat can flow from one point to another. A lower thermal resistance generally allows heat to move easily away from a component reducing the temperature for a given power loss. Thermal resistance depends on factors such as thermal conductivity, thickness, contact area, component packaging and cooling conditions. Copper has high thermal conductivity whereas conventional FR-4 PCB substrate material has much lower thermal conductivity.

The temperature rise of a component depends on the power dissipated and the total thermal resistance between the component and its surroundings. A simplified relationship is:

$$\Delta T = P\text{loss} \times R\theta$$

where ΔT is the temperature rise, Ploss is the power dissipated as heat and $R\theta$is the thermal resistance in °C/W. This relationship gives a way to see how power losses and thermal resistance influence component temperature.

For the proposed project, heat transfer and thermal resistance are parameters, for evaluating the thermal behavior of the buck converter PCB. Copper areas, thermal vias and appropriate component placement can improve heat spreading. Reduce thermal resistance along suitable heat-flow paths. Thermal analysis helps identify where heat accumulates and supports the selection of methods to keep temperatures safe.

C. Identifications of Hotspots in the PCB

A hotspot is a place on a printed circuit board where the temperature's much higher than the areas around it. I notice that in a DC-DC buck converter hotspots usually appear near parts that use a lot of power like MOSFETs, diodes and inductors. The way temperature spreads on the PCB depends on how power each component loses the copper layout, the board material, the size of the traces how close the components are and how many cooling paths the PCB has.

I notice that you can find hotspots by using thermal simulation software or by measuring temperature in experiments. Simulation tools can predict how temperature spreads across the PCB from the board shape, material traits, component

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

losses and set limits. Experimental ways include infrared cameras, thermocouples and temperature sensors. These tools help you find spots and check if the parts stay inside their temperature limits.

I notice that the highest PCB temperature, the temperature difference between components and how big the hot area is give clues, about how the PCB stays cool. A big temperature difference may show that heat is not spread evenly or that the thermal paths are not enough. Spotting these regions early in the design lets engineers decide if the parts need moving if the copper needs changing or if more cooling is needed.

In this project we use identification to check how the DC-DC buck converter PCB stays cool. By looking at the temperature spread we can see which parts and which PCB spots get the hottest. Those hot spots help us choose the ways to improve the design. We will only give the hotspot temperatures and spots after we finish the simulation or the real measurements.

D. Thermal Optimization Using Copper Area and the Thermal Vias

The concept of thermal optimization includes modifying the layout in such a way that the heat is evacuated from the PCB faster. This results in lower temperatures. One of the ways to do so is to use copper pour around components dissipating high power. Copper distributes the heat on the surface of the board. At the same time, it lowers the electrical resistance of the paths through which the current passes. Properly designed copper pours and planes contribute to the performance in addition to their cooling effect.

Thermal vias are special holes in the board connecting copper of the top and bottom layers. These vias provide pathways for dissipation of the heat through the copper connected with the component to the copper of other layers allowing for distribution of the heat over a wider surface. The number of vias, their sizes, positions, and manufacturing technology all influence the efficiency of thermal vias.

Another aspect of optimization includes positioning of the components and design of the copper traces. Components dissipating high power should be positioned in such a way that heat dissipation is good and the connections remain electrical. Copper traces with larger width have less resistance and reduce losses in the conduction. Proper positioning prevents heating of adjacent components. Some applications may benefit from adding heatsinks or air flow.

In the proposed project, the thermal optimization is planned to be done using changes in copper pour patterns, copper trace widths and the position of the thermal vias on the board. The initial and the optimized designs can be tested under operating conditions. The parameters such as the temperature, heat distribution, and locations of the hotspots will allow evaluation of the modifications.

E. Comparison of Conventional and Optimized PCB Layouts

Conventional PCB layout serves as a basis for evaluating the effectiveness of the optimization process. I have noticed that conventional PCB layout tends to suffer from heat concentrations when high-power components are used. The conventional PCB layout may have limited thermal pathways because of ordinary component placement and lack of copper areas. When high-power components produce heat, it may become concentrated in certain regions especially when the copper area is not enough and the thermal pathway to the environment is limited. This situation may increase the temperatures of the components, thus affecting the reliability of the converter.

The optimization process allows improving heat distribution through the adjustment of component placement, copper areas, trace width and locations of thermal vias. Optimized PCB layout allows arranging high-power components in such a way to provide the pathways for the electrical current and better thermal pathways. Additional copper areas and designed vias will allow better heat distribution and proper routing will eliminate unnecessary parasitic effects and electrical losses.

In order to compare the PCB layout and the optimized PCB layout fairly, these two designs have to be evaluated with identical input voltage, output load, switching frequency, component specification, PCB material and environmental conditions. Key parameters for comparison are the temperature of the components, temperature distribution, locations

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

of hotspots and power losses. If the thermal simulations are used in evaluation, then it is necessary to provide identical material properties and boundary conditions for both designs to evaluate their difference.

For this particular project, it is possible to compare the conventional PCB layout and optimized PCB layout in order to evaluate the efficiency of the suggested thermal optimizations and determine whether they actually decrease the temperature and help heat dissipation. The result of such comparison can be represented in the form of pictures, temperature plots and comparison table. The percentage reduction in the temperature can be calculated after experiment or simulation and not assumed beforehand.

IV. SIMULATION SETUP AND RESULTS

A. PCB Design and Simulation Setup

The PCB design and simulation setup play a role in checking how well a DC–DC buck converter handles heat. The process starts with making the circuit diagram and the PCB layout using tools like KiCad or Cadence Allegro. The layout has parts like the MOSFET, the diode, the inductor, the input capacitor, the output capacitor and the control circuit. The positions of these parts the copper lines, the copper areas and the size of the board are decided based on the heat needs of the converter. Careful placement of the components. The way the current flows are important to reduce power loss and help the heat go away better.

Once the PCB layout is done a model for analysis can be made using software that works with PCB heat studies. The model needs to have the size of the PCB the thickness of the copper, the material of the board the types of the components how heat they make and how the cooling happens around them. The MOSFET, the diode and the inductor are the parts that make heat because of the energy they lose while working. The right conditions, like the temperature around the board and the air moving past it must be set to get results from the simulation. The setup should copy the life working conditions as close as possible.

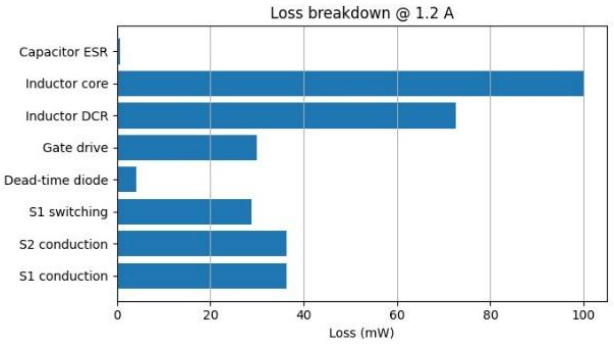


Fig. 3. Power Loss Breakdown at 1.2 A

To see how well the heat is managed two versions of the PCB can be made: an one and an improved one. The normal one is the base design. The improved one has changes like copper area, better part placement, wider lines for electricity or more thermal vias where needed. Both versions must use the way of working, the same parts and the same heat conditions. This way it is possible to see if the changes help spread the heat better and lower the temperature. Because the results depend on how good the model's what assumptions are made the settings, for the simulation must be written down clearly.

B. Temperature Distribution Analysis

Temperature distribution analysis is performed in order to evaluate the temperature distribution on the PCB while the buck converter operates. Heat generation is the result of the power dissipation of various elements that occur depending on their electrical parameters and environmental conditions. Thus, the MOSFET is affected by heat caused by conduction and switching losses, while heat generation of the diode is determined by its voltage and current. In

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

addition, the inductor heats up due to the heat generated by the resistance of its windings and core losses. Temperature distribution is one of the factors that affect thermal performance evaluation of the PCB.

Thermal simulation can present temperature distribution as a color map. According to the thermal map, cooler areas are usually painted blue or green, while warm zones are marked with yellow, orange, and red. Temperature distribution analysis allows evaluating temperature generation and distribution. It depends on the amount of copper on the PCB, its material, positioning of the components, the number and structure of thermal vias, and ambient temperature. Even temperature distribution implies efficient heat spreading; however, it is also necessary to pay attention to temperature limits of the components and real power losses.

In the proposed project, temperature distribution analysis will be performed for the PCB and improved PCB layout. The first option will allow getting the initial data concerning natural temperature distribution while the second one will present how temperature distribution changes with the improvement of the copper layout and paths of heat dissipation. Thermal maps will help to define if there are even distribution of heat and reduced hot spots. The results will be interpreted on the basis of temperature values from the simulation and not only color representation of the maps.

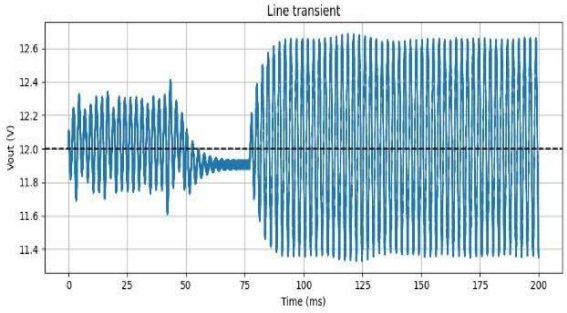


Fig. 4. Line Transient Response of DC-DC Buck Converter

Temperature distribution analysis also helps in defining the necessity of additional modifications related to cooling of the board. In case of localization of heat in one zone, it is possible to change the copper layers of the board, component placement, and thermal vias positioning. The results may also indicate on the possibility of using a heatsink or increasing air flow. Thus, temperature distribution analysis provides information regarding the thermal characteristics of the converter and possible improvements.

C. Maximum Temperature and Hotspot Identification

Maximum temperature is one of the important factors used to check how well a DC–DC buck converter PCB works in terms of heat. It shows the temperature that a part or area reaches when the PCB is working as it should. A hotspot is a place where the temperature's much higher than the areas around it. Hotspots can be close to the MOSFET, the diode, the inductor or other parts that create a lot of heat. Finding these spots helps see if the PCB design can spread the heat properly.

The maximum temperature can be found using simulation or real measurements. In simulation the software guesses the temperature of parts of the PCB based on how much heat each component creates the materials used, the shape of the board and how it cools. The result is a map that shows the hottest area and the temperature there.In experiments an infrared camera or thermocouples can be used to check temperatures at points. Measurements help check if the simulation is correct and show how the real heat behavior is different from what was expected.

In this project the focus is on finding the areas that get the hottest in the PCB layout. Once these hotspots are found they can be studied to see why the heat builds up. Possible reasons might be that some components create much heat there is not enough copper to carry the heat away or there are not enough ways for the heat to move. After making changes to improve the heat handling the same areas can be checked again to see if the highest temperature went down.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

The places where the heat is strongest should also be compared because improving the design might move where the heat is strongest on the board.

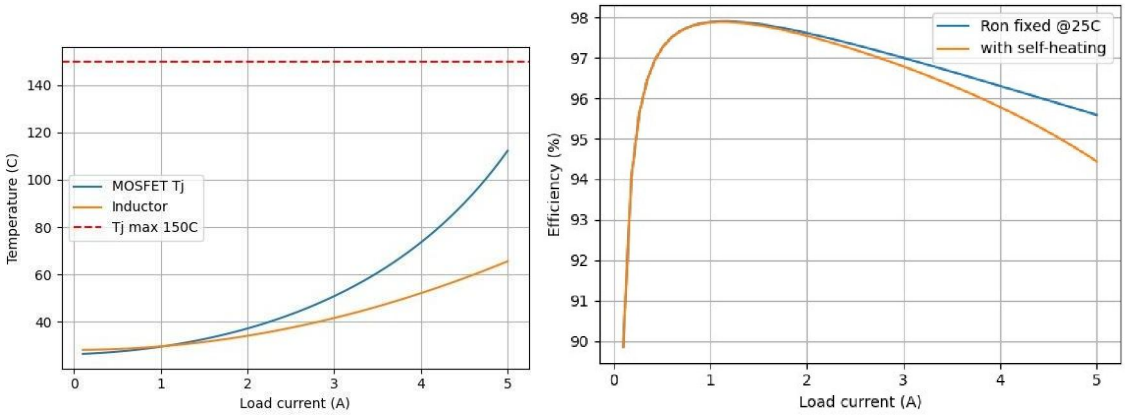


Fig. 5. Thermal Analysis of DC-DC Buck Converter

The highest temperature needs to be checked against the limits that the component makers say are safe. A lower maximum temperature can mean heat control but it does not always mean the PCB works better electrically or lasts longer. Other things, like the temperature inside the components how often the temperature changes the stress on the solder joints and the temperature, around the PCB should also be taken into account. The highest temperature and the places where heat builds up from tests or simulations should be written down and used to back up the findings of the project.

D. Comparison of Thermal Perfomance Before and After Optimization

A comparison of the performances of conventional and optimized PCB layouts allows estimating the efficiency of design improvements. Conventional PCB serves as reference design, and optimized PCB has copper area increased, improved components' placement, widened copper traces, and additional thermal vias where required. Both designs have to be analyzed using the same input voltage, output load, switching frequency, components' specifications, ambient temperature, and cooling system. These factors should be constant because only PCB design improvements could cause differences between the results of analysis.

Comparison could include such parameters as PCB temperature, temperature rise relative to ambient, appearance of hot spots, temperature distribution, and estimated power losses. Thermal map would show the heat distribution in each layout, and comparison table would give numeric information about the results. Higher maximum temperature or more uniform temperature distribution in optimized PCB would indicate an improvement of the thermal paths, however, the result depends on the design, components losses, and cooling system so any improvements should not be expected before the analysis was made.

The percentage of temperature drop can be calculated using the measured or simulated temperature in both designs. Assuming that *T*conv is the temperature of the conventional PCB, and *T*optis the temperature of the optimized PCB, the temperature reduction can be calculated as follows:

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

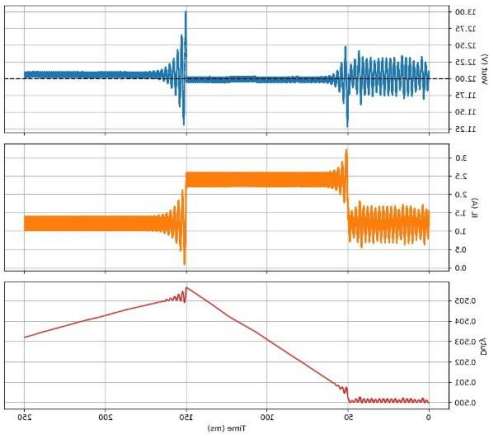


Fig. 6. Load Transient Response of Buck Converter

Temperature Reduction (%) = (Tconv − Topt)/ Tconv × 100

However, this formula gives the comparison of absolute temperatures and for comparison of thermal improvement it would be more useful to calculate the percentage reduction of temperature rise relative to ambient. This temperature is obtained by subtracting the temperature from the maximum temperature of each design. The chosen method of comparison has to be specified in the report.

The results of the comparison allow determining the design improvements that provide the thermal benefit. Additional copper area in PCB would stay if the temperature rises were reduced. In case when hot spots still exist near some components additional improvements of components' placement, thermal vias, or cooling system would be required. Any percentages of improvement should be based on simulation or measurement. If the analysis is not performed yet, the comparison should be presented as evaluation.

E. Discussion of Results

The discussion will assess the relevance of the thermal analysis of the converter board to the aims of the project. To assess the performance of the PCB design, one needs to analyze the temperature distribution, maximum temperature, location of hot spots, and temperature rise. One has to define which elements cause the greatest amount of thermal load, as well as the ways in which the PCB design affects heat spreading. One needs to consider the interplay between power losses, copper distribution, thermal resistance, and cooling conditions employed in the thermal analysis.

This can result in hot spots being formed on the PCB. If heat is concentrated in certain areas and the PCB doesn't have a heat dissipation method, these areas will become hot spots. This will cause a decrease in thermal performance.

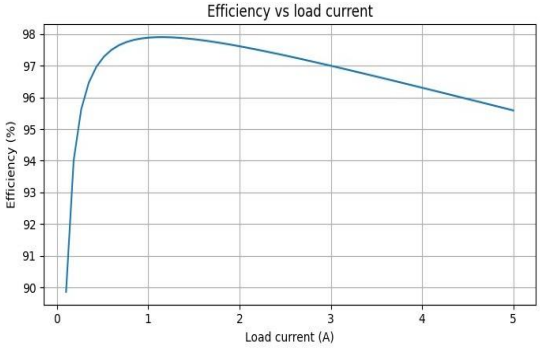


Fig. 7. Efficiency Variation with Load Current

If the optimized PCB shows lower temperature rise as compared to the conventional PCB, then the improvement in temperature rise can be attributed to more effective heat spreading, reduced electrical resistance, or better heat transfer

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 6, Issue 2, November 2026 Impact Factor: 8.2

routes. The improved temperature distribution means that heat is effectively spread away from the local areas of high temperature. However, one has to pay attention to the issue that the results should be analyzed assuming the equality of the conditions under which the boards have been tested. The changes in component losses, ambient temperature, air flow, and other factors can change the results and should not be considered as an effect of PCB layout only.

One also has to assess the results in accordance with the practical requirements of designing the PCB. The converter board should have an appropriate electrical clearance, current carrying capability, reliable solder joints, and adequate temperatures. Increasing the copper area and adding thermal vias can contribute to the more effective heat spreading; however, these changes have to satisfy manufacturing capabilities, physical characteristics of the board, and electrical requirements of the design.

In conclusion, one has to admit that the results obtained during the simulation are affected by the assumptions about material properties, component losses, and cooling conditions. The experimental testing can verify the predictions and detect thermal phenomena not taken into account in the model.

V. DESIGN LIMITATION AND PRACTICAL CONSIDERATION

A. Components & Materials

The thermal performance of the DC-DC buck converter PCB will depend heavily on the component and materials properties. Small changes in component properties can change the thermal performance of the PCB. Each component has thermal properties:

• MOSFET : The MOSFET will generate heat due to conduction and switching losses. This can increase the temperature of the PCB. Often, the MOSFET becomes a hot spot on the PCB.

• Diode : The diode can also generate heat due to the voltage drop and conduction. This can create an increase in load on the PCB. This can also create a hot spot.

• Inductor : The inductor can generate heat due to resistive losses in the windings and magnetic losses. This can create an increase in load on the PCB. This can create a hot spot.

The PCB material will also affect the thermal performance. The conductivity of the material is key. The material used is the FR-4 substrate. This material has a degree of conductivity, however, it is not as conductive as metal. This will cause limited heat transfer. Without sufficient copper area or features for heat dissipation, the high powered components could cause heat build up. This increases the temperature of the PCB.

When designing the PCB, the copper thickness, trace size, component placement and PCB substrate should be considered. Thicker copper traces and copper areas can reduce the resistance. This will help spread the heat. Thicker traces will help spread the heat.Thermal vias can help transfer heat between copper layers. However, the effectiveness will depend on the power loss from the components, PCB design and cooling conditions.

When designing, the material properties and component data sheet limitations should be considered to keep the converter within the required temperatures. The thermal limits given by the data sheets will help guide the design.The PCB shouldn't be optimised for heat simply by increasing the amount of copper. This should take into consideration the component ratings, size of the PCB, electrical requirements and limitations of the materials chosen. The PCB should be optimised to balance the copper with the limitations.

B. Thermal Simulation Assumptions

Thermal simulations are useful for seeing the temperature distribution and thermal dissipation of the DC-DC buck converter PCB. This will show how the thermal profile changes under certain conditions and where hot spots are located.The accuracy of the simulation will depend on the assumptions and parameters chosen. The assumptions will be based on the thermal conductivity, copper thickness and ambient temperature.

To compare the conventional and optimised PCBs, both should be simulated under the operation conditions. The input voltage, output load, switching frequency, ambient temperature, component models and cooling conditions should be the same. This will help identify if the changes in layout caused the differences in thermal performance.The thermal

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 2, November 2026
Impact Factor: 8.2

analysis should consider the temperatures reached by the components, the temperature distribution and the increase in temperature from ambient. This will show where the highest thermal load is on the PCB.Thermal simulations should be considered as estimations. The model can be simplified, component properties can vary and cooling conditions can vary. I always verify with measurements.

C. PCB Manufacturing Limitations

The PCB should be optimised to keep thermal performance whilst meeting the manufacturing and electrical design criteria.The trace widths and copper thickness should be chosen based on the current flowing through the conductor and acceptable temperature rise. The trace widths and copper thickness should be chosen to ensure the thermal rise is within limits. If the trace widths are too small this can increase the resistance. This can increase the heat and cause larger copper areas. This can increase the size of the board. The layout should also allow for adequate clearance between areas. The trace widths should be designed to keep the load low on the PCB whilst keeping the board size reasonable. Thermal vias can help transfer heat between copper layers. However, you must consider how many thermal vias are used, size and positioning. This should be achievable by the manufacturer. The component placement should allow for wiring, soldering and visibility. The PCB should be designed to allow for solder joints and keep sensitive components away from hot areas. The vias should be planned to ensure good thermal transfer on the PCB and achievable by the manufacturer. The optimal PCB layout should balance thermal performance, wiring, easy manufacture, correct size and reliable assembly. A PCB that looks good in the simulation may not work if it goes outside of the manufacturers limits or is difficult to place components on. These guidelines should be considered at all stages of the design. The final board will be thermal tolerant and achievable.

D. Testing

Testing is needed to prove the thermal values obtained from the simulation and to see the real performance. Whilst simulations can give hints as to where heat will build up, real life can throw up some surprises.Test the board and new board in the same way. Using the same power input, load, switching speed, room temperature and cooling fan. Temperature can be taken using thermocouples, sensors or infrared camera. Take temperature readings from the important hot spots such as the MOSFET, Diode, Inductor and any other components that could get hot. Record the component, temperature rise from the room and hot spot locations. Compare these to the simulation. Consider the difference in temperature between the board and new board to see if improvements have been made.If the temperatures don’t match the simulation, look into why. This could be due to higher power loss from components.

VI. CONCLUSION

Thermal analysis and optimization of a DC-DC buck converter PCB help to understand how the PCB behaves under heat and to improve how heat is let out. During operation MOSFET, diode and inductor generate heat because of losses. Thermal simulation is used to find hotspots to look at the temperature and to see how temperature spreads across the PCB before the PCB is made. In this project the initial PCB layout and the optimized PCB layout are compared while the conditions are the same. Increasing copper area using traces adding thermal vias and improving where the components sit can reduce heat buildup. These changes help management. Overall this project shows how thermal simulation and thermal analysis, with PCB layout optimization can help make a DC-DC buck converter more reliable. It also shows how to cut problems before the PCB is built.

REFERENCES

[5] T. C. Neugebauer and D. J. Perreault, “Computer-Aided Optimization of DC/DC Converters for Automotive Applications,” IEEE Transactions on Power Electronics, vol. 18, no. 3, pp. 775–783, May 2003, doi: 10.1109/TPEL.2003.810866.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 6, Issue 2, November 2026 Impact Factor: 8.2

[6] C. Gao, H. Liu, J. Huang and S. Diao, "Steady-State Thermal Analysis and Layout Optimization of DC/DC Converter," 2014 Prognostics and System Health Management Conference, Zhangjiajie, China, 2014, pp. 405–409, doi: 10.1109/PHM.2014.6988203.
[7] Y. Liu, M. Huang, H. Wang, X. Zha, J. Gong and J. Sun, "Reliability-Oriented Optimization of the LC Filter in a Buck DC–DC Converter," IEEE Transactions on Power Electronics, vol. 32, no. 8, pp. 6323–6337, Aug. 2017, doi: 10.1109/TPEL.2016.2619690.
[8] E. A. Jones et al., "Thermal Characterization and Design for a High Density GaN-Based Power Stage," 2018 IEEE 6th Workshop on Wide Bandgap Power Devices and Applications (WiPDA), 2018, pp. 295–302, doi: 10.1109/WiPDA.2018.8569136.
[9] R. Caillaud, C. Buttay, R. Mrad, J. Le Leslé, F. Morel, N. Degrenne and S. Mollov, "Thermal Considerations of a Power Converter With Components Embedded in Printed Circuit Boards," IEEE Transactions on Components, Packaging and Manufacturing Technology, vol. 10, no. 2, pp. 230–239, Feb. 2020, doi: 10.1109/TCPMT.2019.2939969.
[10] W. Belloumi et al., "Automatic PCB Layout Optimization of a DC–DC Converter Through Genetic Algorithm Regarding EMC Constraints," IEEE Access, vol. 9, pp. 149870–149882, 2021, doi: 10.1109/ACCESS.2021.3124935.
[11] K. Booth, H. Subramanyan, J. Liu and S. M. Lukic, "Parallel Frameworks for Robust Optimization of Medium-Frequency Transformers," IEEE Journal of Emerging and Selected Topics in Power Electronics, vol. 9, no. 4, pp. 5097–5112, Aug. 2021, doi: 10.1109/JESTPE.2020.3042527.
[12] H. T. Le, Y. Nour, F. Jensen, A. Knott, A. Han and Z. Ouyang, "Silicon Power Interposer Technology (PIT) for Integrated Power Converter," IEEE Transactions on Power Electronics, vol. 38, no. 6, pp. 6755–6758, June 2023, doi: 10.1109/TPEL.2023.3240357.
[13] K. Sado, J. Hannum and K. Booth, "Digital Twin Modeling of Power Electronic Converters," 2023 IEEE Electric Ship Technologies Symposium (ESTS), Alexandria, VA, USA, 2023, pp. 86–90, doi: 10.1109/ESTS56571.2023.10220465.
[14] X. Tian, N. Jia, D. DeVoto, P. Paret, H. Bai, L. M. Tolbert and H. Cui, "PCB-on-DBC GaN Power Module Design With High-Density Integration and Double-Sided Cooling," IEEE Transactions on Power Electronics, vol. 39, no. 1, pp. 507–516, Jan. 2024, doi: 10.1109/TPEL.2023.3311440.
[15] A. Hussain, D. Perez, D. Cucak and K. Booth, "PCB-Based Inductor Design for 1-kW, 1-MHz Buck Converter," 2024 IEEE Applied Power Electronics Conference and Exposition (APEC), Long Beach, CA, USA, 2024, pp. 3199–3203, doi: 10.1109/APEC48139.2024.10509065.
[16] K. Sado, J. Peskar, S. Ionita, J. Hannum, A. Downey and K. Booth, "Real-time Electro-thermal Simulations for Power Electronic Converters," 2024 IEEE Applied Power Electronics Conference and Exposition (APEC), Long Beach, CA, USA, 2024, pp. 2616–2623, doi: 10.1109/APEC48139.2024.10509396.
[17] A. Hussain, K. Sado, D. Perez and K. Booth, "FEA-Driven Solutions to Minimize Driving Loop Inductance and EMI in MHz PCB Designs," 2024 IEEE Sixth International Conference on DC Microgrids (ICDCM), Columbia, SC, USA, 2024, pp. 1–4, doi: 10.1109/ICDCM60322.2024.10665225.
[18] A. Hussain, K. Sado, D. Perez and K. Booth, "Beyond-Nominal Operation of GaN-Based Converters for High-Power Density Applications," 2024 IEEE Transportation Electrification Conference and Expo (ITEC), Chicago, IL, USA, 2024, pp. 1–5, doi: 10.1109/ITEC60657.2024.10598924.
[19] J. G. Berger, C. A. Rojas, A. H. Wilson-Veas, R. A. Bugueño, A. Peralta, S. Salinas, J. Marín, H. Young and L. Callegaro, "On the Experimental Verification of Electrothermal Modeling of GaN-HEMT-Based DC–DC Converters," IEEE Transactions on Power Electronics, vol. 40, no. 4, pp. 5518–5530, Apr. 2025, doi: 10.1109/TPEL.2024.3514492.