PCB Design Guidelines to Maximize Cooling of eGaN® FETs

Adolfo Herrera, Engineer, EPC

GaN FETs, like any other transistor, dissipate power during switching, which manifests in the form of heat [1] and can damage the device if specific limits are exceeded. Heat management is thus an important part of converter design. In some applications, thermal management strategies are limited to just those that can be utilized using only the PCB. Several effective design techniques can be implemented at the board level to maximize heat dissipation of the devices.

This application note presents simple thermal management guidelines maximizing heat conduction from the GaN FETs to the environment using just the PCB and optimizes thermal performance without the use of a heatsink. In addition, a case study is presented with simple and effective thermal management solutions for the cooling of a development board with two active GaN FETs.

Thermal Overview

Power electronic devices in operation dissipate energy causing self-heating. The heat generated flows from the heat source, referred to as the device junction, towards colder bodies through two main parallel paths – to the printed circuit board (PCB) and to the case or backside of the device. The heat flux for each path is a result of the temperature difference and the thermal resistances encountered along the path, as shown in the formula:

$$R_{\theta JX} = \frac{T_J - T_X}{P} \qquad \text{Eq. 1}$$

Where,

$R_{\theta JX}$, alternatively, θ_{JX} (°C/W or K/W) = thermal resistance from junction to a reference location X

T_J (°C or K) = device junction temperature in steady state conditions

T_X (°C or K) = temperature of reference location (board (B), case (C), or ambient (A))

P (W) = power dissipated in the device

The first thermal resistances encountered by the heat flux are internal to the device, from the junction to the board ($R_{\theta JB}$) and from the junction to the case ($R_{\theta JC}$). Both depend on the device construction and the thermal conductivity of the materials used. The second thermal resistances encountered are outside of the FET and represent the thermal system, such as the PCB, heatsink, and environment around the board. These thermal resistances are the board to ambient ($R_{\theta BA}$) and case to ambient ($R_{\theta CA}$) respectively and are usually the dominant ones in the overall junction to ambient thermal resistance ($R_{\theta JA}$) path. Therefore, the design of the PCB plays a key role in the effectiveness of thermal strategy used. The impact of several PCB design parameters on $R_{\theta JA}$ will be discussed in the following sections and reported for either still air ($R_{\theta JA}$) or moving air ($R_{\theta JMA}$).

Figure 1(a) shows how heat flows from the two GaN devices mounted on a PCB in a half-bridge configuration. This setup serves as the basis for examining PCB-only cooling. The equivalent thermal circuit of this configuration is shown in Figure 1(b). The collective thermal resistances in this system reduce to a single equivalent thermal resistance of $R_{\theta JA}$. In Figure 1(b): R_{PCBT-A} and R_{PCBB-A} represent the resistance from the top and bottom layers of the PCB to ambient respectively; $R_{12,PCB}$ represents heat exchange laterally within the layers of the PCB; and $R_{z,PCB}$ represents the thermal resistance transversely from the top to the bottom layer of the PCB.

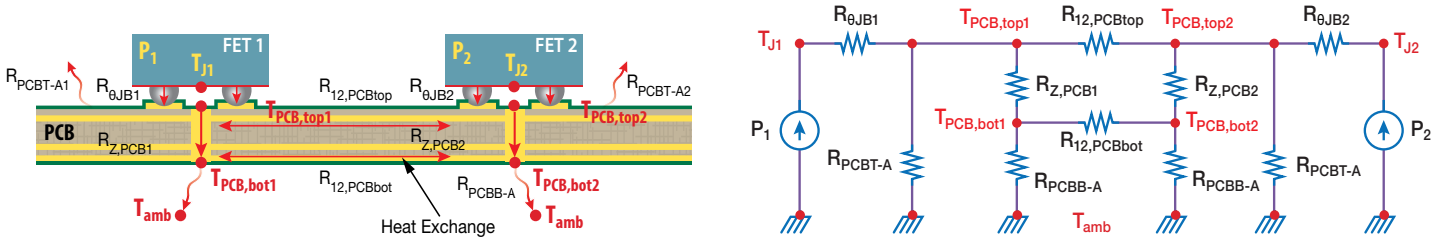


Figure 1. (a) Cross-section view of GaN devices mounted on a PCB in a half-bridge configuration without a heatsink. (b) Equivalent thermal circuit for GaN devices in a half-bridge configuration without a heatsink

Designing a PCB for maximum heat dissipation

The various PCB design characteristics that have thermal characteristics are board area, copper area, copper thickness, vias, and device proximity. Each independent characteristic has a thermal performance and an impact on the electrical system and will be discussed next.

PCB Area for Improved Cooling

One of the main characteristics that influences heat transfer to the ambient is the area of the PCB. A larger board area allows for more heat transfer between the PCB and the environment as there is more area for convection to occur. This relationship is shown in Figure 2, where two EPC2218 [2] devices dissipating 1 W each are simulated for varying PCB surface areas. Maximum board temperature sharply rises as the PCB gets smaller, but there are diminishing returns in cooling as board size increases past a certain threshold. The area of the GaN device itself also has an impact on heat transfer. Larger devices have more solder bumps connecting to the PCB, resulting in more heat flow from the device into the copper on the board. Figure 3 shows how temperature rise changes for different device sizes on the same board area of 2.56 cm^2 and the same power dissipation of 1 W total. There is also a nonlinear trend in the relation between these two factors as the decrease in temperature rise as the devices continue to increase in size plateaus very quickly.

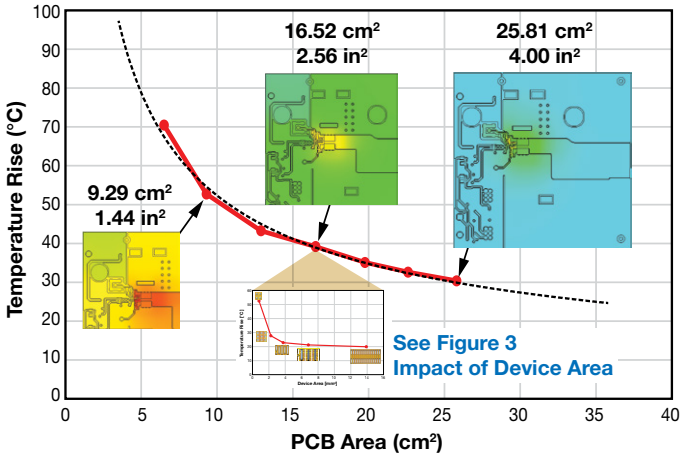


Figure 2. Effect of board size on temperature rise

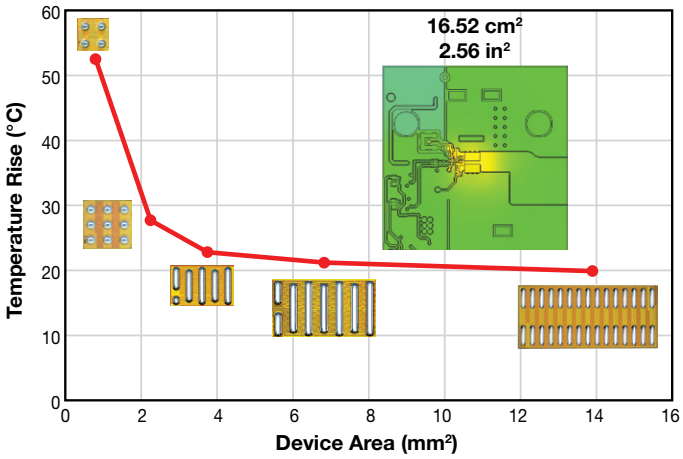


Figure 3. Effect of device size on temperature rise for a fixed board area

A 4 in^2, 4-copper layer board will serve the baseline on which the PCB-only cooling design techniques are examined.

Impact of Copper Trace Width and Plane Area on Thermal Performance

Copper is the dominant path for heat to spread from the FET and to transfer to the ambient. This thermal heat spreading effect is primarily affected by the trace geometry, area, and thickness of the copper layers [3]. The most impactful layer for heat-spreading is that to which the device is soldered to. The width of the copper traces that the part is connected to as well as the total area of the copper must be considered for keeping thermal resistance as low as possible. Example layout designs of the top layer for an EPC PQFN device are shown in Figure 4. Figure 4(a) shows a design that would result in a higher thermal resistance throughout the top layer of the board due to the narrow traces and lack of large thermal dissipating area, raising the thermal resistance from junction to ambient. The thin traces that the solder bars of the device rest on in combination with the disconnected copper planes of small area would serve to trap the heat dissipated from the device in the pad area, resulting in a penalty to thermal performance. Figure 4(b) shows the design of a layer that maximizes the area of the copper planes around the device with wider traces connected to large areas of copper, which in turn keeps the thermal resistance from junction to ambient low.

Impact of Copper thickness

Using thicker copper traces for low electrical resistance also benefits thermal resistance and provides a high thermal-conductance medium at each layer in the PCB. The effectiveness of increasing the thickness of copper layers is shown in Table 1 for two EPC2218 devices, each dissipating 1 W of power. Increasing the copper thickness from 1 oz to 2 oz results in a decrease in $R_{\theta JA}$ of 21%. Further increases in thickness beyond 2 oz copper though results in diminishing returns in $R_{\theta JA}$ reduction. Copper thickness greater than 2 oz also presents challenges in etching fine

features into the PCB. The fine pitch of eGaN FETs restricts designing boards to copper thickness with a recommended maximum of 2 oz [4].

When airflow is considered as a cooling method, the decrease in $R_{\theta JMA}$ is lower relative to that of the natural convection condition. In addition, increasing the number of layers in the PCB also increases the heat spreading capability of the PCB itself, but as buried layers, they have very little area of contact with the surrounding environment and therefore a limited impact on the reduction of thermal resistance to ambient. Nevertheless, the number of layers has a determinant influence on the electrical resistance of the board and thus should be designed based on electrical performance rather than on thermal considerations.

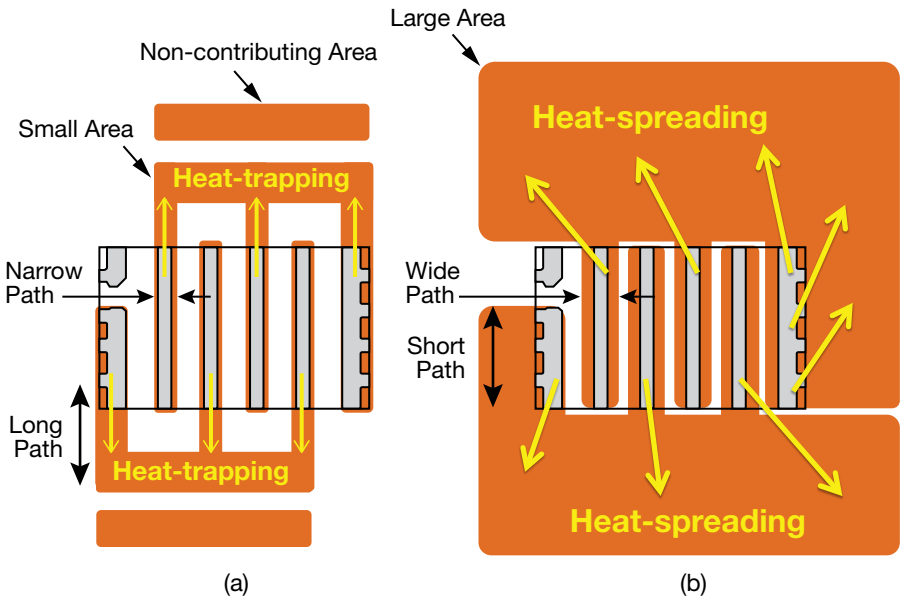


Figure 4. Illustrations of copper layouts for a PQFN GaN device with (a) poor thermal design and (b) recommended thermal design

Table 1. Temperature rise of 2 EPC2218 each dissipating 1 W, for different copper layer thicknesses under two cooling conditions using baseline PCB

Copper Thickness	ΔT Still Air	ΔT 400 LFM
1 oz / 35 µm	64°C (baseline)	42°C (baseline)
1.5 oz / 53 µm	57°C (-13%)	39.5°C (-6%)
2 oz / 70 µm	53°C (-21%)	37.5°C (-12%)
3 oz / 105 µm	51°C (-25%)	35.5°C (-18%)

Using the Bottom Side of the PCB for Additional Heat-Spreading

The insulating dielectric layers of a PCB have low thermal conductivity which results in a very high thermal resistance from the device to the copper layer on the opposite side of the PCB. However, the inner and bottom layers of the PCB can still be designed into the thermal solution by connecting them to the top layer with highly conductive "heat pipes." By including these heat pipes in the board, paths of heat conduction are formed to the bottom layer so that it can be more effectively used to transfer heat to the ambient. Copper plated vias are the predominant PCB design elements that can fulfill this purpose. To obtain the greatest benefit from thermal vias, they must be placed either directly beneath the device or adjacent to the device and as close as possible to the device.

Placing vias under the device bumps comes with additional design considerations such as diameter limits, tenting requirements, and assembly requirements. Two main parameters define a via; hole diameter and annular ring diameter. The copper-plated vias' hole diameter and annular ring diameter must be smaller than the width of the pad to which it connects. The annular ring diameter is also tied to the minimum hole diameter that can be physically drilled into a PCB of 7.8 mils (200 µm.) Thus, for a via with a hole diameter of 7.8 mils, the minimum manufacturable annular ring diameter is 13.8 mils (350 µm) and apply to PCB designs up to 2 oz copper. Typical class 2 vias plating thickness is 0.787 mils (30 µm.) Since the via is placed in the bump area of the device that is soldered to the board, solder can drain down the via hole during reflow. To prevent this from happening, vias under the bump must be filled and subsequently plated over with copper. In addition, the via must be "covered" with a solder mask to prevent altering the solder-mask design of the device land-pattern which affects the quality of the solder joint. According to the IPC4761 standard [5], the type of via that meets these design requirements is the Type VII via. An illustration of a Type VII via is shown in Figure 5.

For board designs that make use of only adjacent vias rather than vias under the bump, Type VII vias are no longer required but must still be covered with solder-mask to prevent solder bleed during reflow.

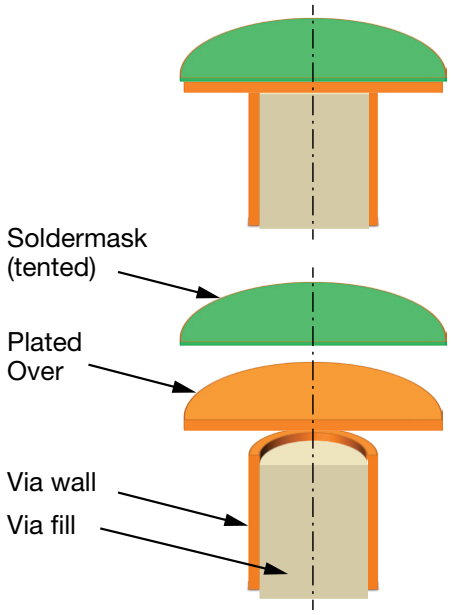


Figure 5. Thermal via construction based on IPC4761 Type VII vias - Via-in-Pad-Plated-Over (VIPPO)

Figure 6 shows the impact on thermal resistance from junction to moving air ($R_{\theta JMA}$) for two EPC2619 devices each dissipating 1 W in 400 LFM airflow [6] when using vias to transfer heat from the top layer to the bottom layer. The first row of images shows various via layout designs; from left to right with no vias (left), adjacent to the devices vias (center) and vias under the bump (right). Vias under the bump should be staggered rather than placed in straight lines as a square matrix can weaken the integrity of the PCB [7]. The next two rows show the effect the various via layouts on temperature of the devices and surrounding board volume. Using the no via case as a baseline, the adjacent via layout shown in Figure 6(b) results in a 22% decrease in $R_{\theta JMA}$ while the under-bump design in Figure 6(c) results in a decrease of 33% of $R_{\theta JMA}$. The cross-section view in the bottom row of images shows how the vias facilitate heat transfer through the dielectric of the PCB to the opposite side of the board.

Joule Heating Effect on Device Heat Flux

Heat is not only generated within the GaN device. The flow of current through the copper in the board, particularly in thinner connections with increased current density, also results in power losses that generate heat. The heat that is produced through electrical current flow is known as joule heating. Because of the small size of GaN devices, there is a higher concentration of connections around the device which can lead to higher current densities. Joule heating surrounding the parts can be reduced by designing parallel connections, strategic use of vias and shortening concentrated paths of current. Figure 7 shows a FEA simulation result for joule heat concentrations in the top layer of an EPC90123 demo board carrying 5 A [8]. Figure 7(a) shows how joule heat concentrates around the edges and corners of copper traces. Figure 7(b) shows how joule heating can be decreased substantially by designing wider pads where the current is forced to neck through. In the regions of the highest current densities, the joule heating decreases by about three to five times as much when the copper pad width is doubled. The impact of vias to carry current is also shown in Figure 7(c); the traces that have vias show lower joule heat concentrations as some current can flow into the buried layers, reducing the heating in the top layer.

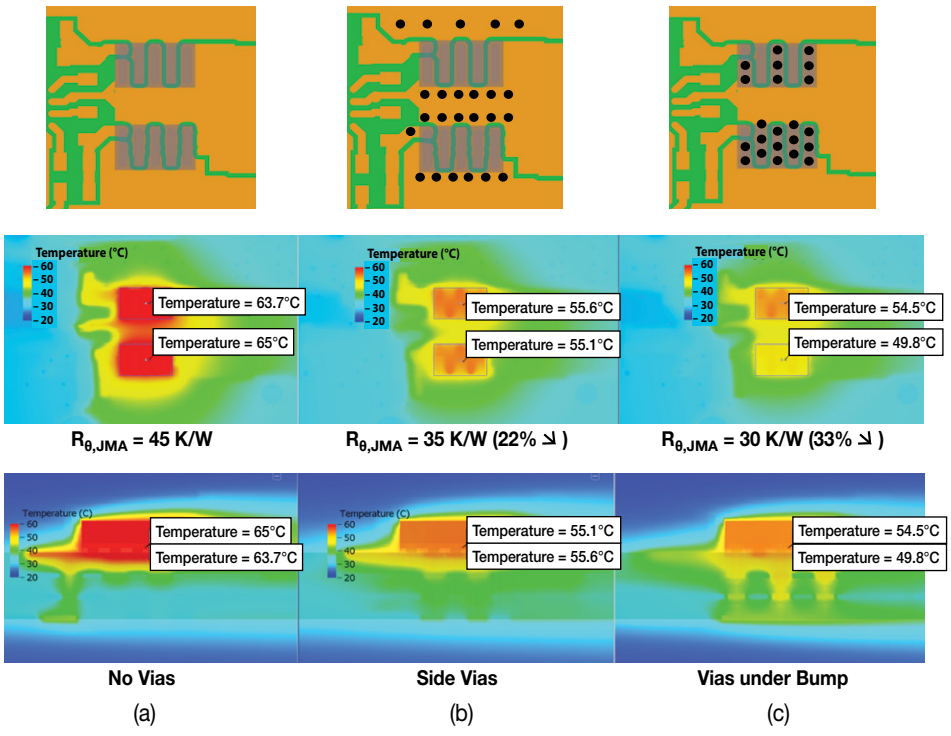


Figure 6. PCB thermal performance comparison for various via configurations

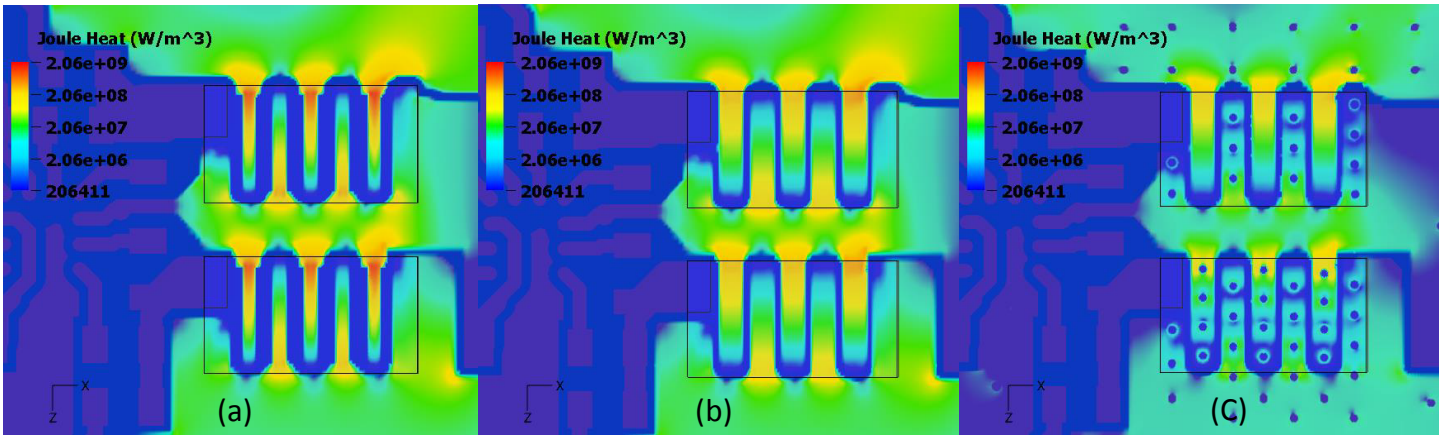


Figure 7. Joule heating in the top layer of the EPC90123 demo board with (a) narrow copper traces to attach EPC2218, (b) wide copper traces, and (c) wide copper traces with vias to parallel flow. Image produced in CelsiusEC FEA simulation. [8]

Layout Configuration

Typical GaN FET layouts [9, 10] require close proximity between the two devices in a half bridge configuration to ensure lowest loop inductance [11]. Heat-generating components, such as two FETs in a half-bridge configuration, on a PCB exchange heat flux when placed in proximity with each other which increases $R_{\theta JA}$ for each device. To approximate the effects of heat-flux exchange, a simple superposition model can be used to analyze the combined effects of their temperature rise. However, because there are nonlinear dependencies on temperature there will be a small error in the final temperature rise calculated. Figure 8 shows the superposition principle of two heat sources in simulation, note that the simulations are done in an environment with an ambient temperature of 20°C. With only the high-side FET dissipating 1 W of heat flux, its temperature rise is 32.9°C; and with addition of the low-side FET dissipating 1 W of heat flux, co-heating raises the high-side FET by 22.4°C; adding the two temperature rises gives an estimated temperature rise of 55.3°C, which is close to the temperature rise of 52.3°C shown in Figure 8(c).

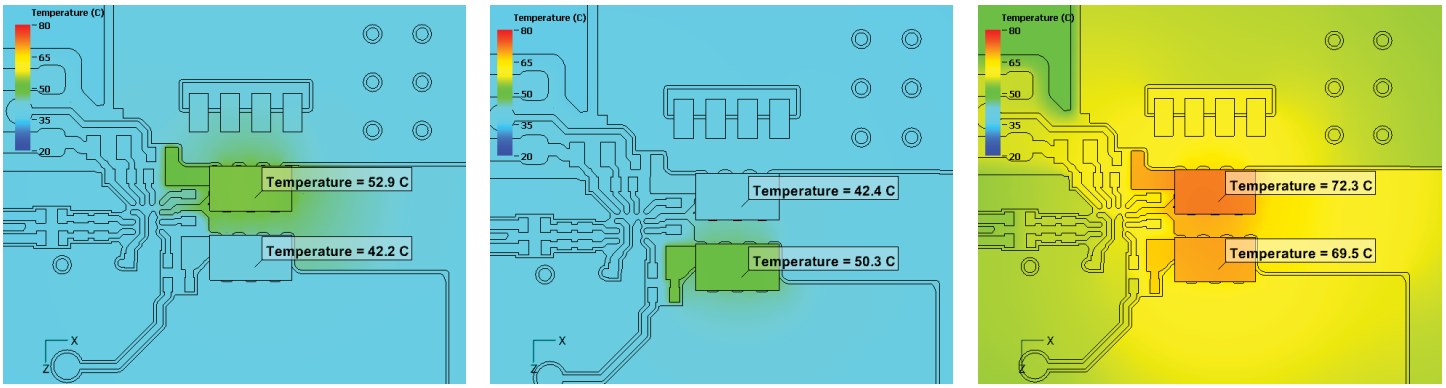
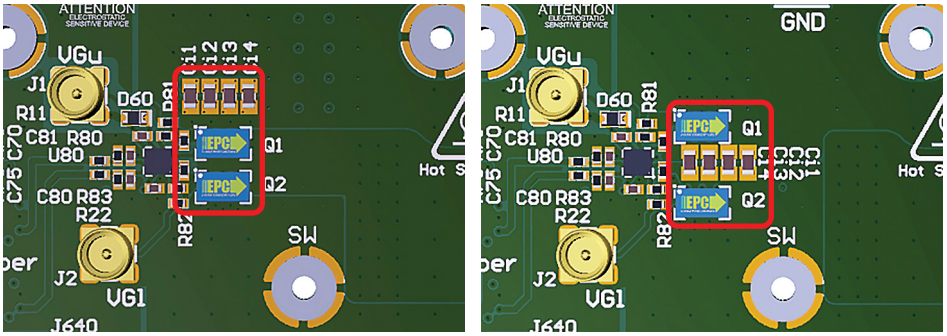


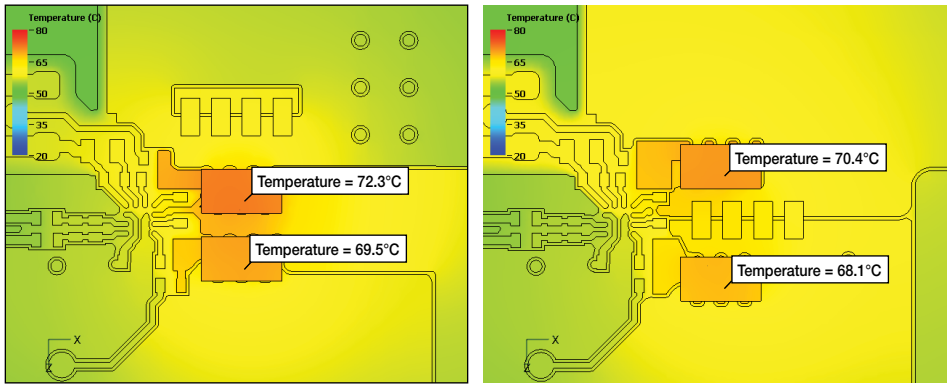
Figure 8. (a): Only high-side FET dissipating 1 W. (b): Only low-side FET dissipating 1 W. (c): Both FETs dissipating 1W

Figure 9 shows two half-bridge topology boards each with different bus capacitor layout configurations intended for examining the effects of devices separation on co-heating. Figure 9(a) shows a traditional adjacent bus capacitor layout where both devices are placed as close as possible and Figure 9(b) shows a center capacitor layout where the bus capacitors are placed between the devices. The objective in separating the devices with the bus capacitors is to reduce the magnitude of heat flux exchange between the two parts, which should also reduce $R_{\theta JA}$. Figure 10 shows the differences in temperature rise for two FETs in a half-bridge configuration; adopting a center bus capacitor layout reduces $R_{\theta JA}$ by about 5% for both FETs. The two layout configurations can be achieved with similar loop inductance [12] and thus have no impact on electrical performance.



(a) (b)

Figure 9. (a): Adjacent bus capacitor layout. (b): Center bus capacitor layout

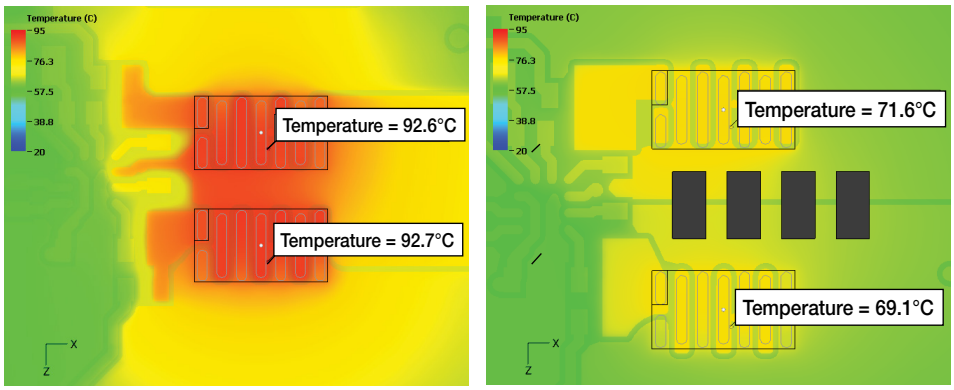


(a) (b)

Figure 10. (a): Adjacent bus capacitor temperatures. (b): Center bus capacitor temperatures

Impact of Combined Techniques

When combining the techniques outlined in this app note, it is possible to reduce $R_{\theta JA}$ by close to 30% when compared to the baseline with none of the techniques used. Figure 11 shows a simulation of two boards and the temperature of each FET in a half bridge. The board in Figure 11(a) has 0.5 oz copper, 4 layers, no vias near the FETs, and both FETs placed close to each other. The board in Figure 11(b) has 2 oz copper, 4 layers, vias under the bump and close to the FETs, and the FETs separated using the center bus capacitor layout configuration. Adopting these techniques to the board layout not only can reduce $R_{\theta JA}$ by over 35%, but also comes at little to no additional cost in manufacturing of the PCB. Table 2 shows the effectiveness of each technique discussed in this application note when tested on several EPC demo boards, isolating the different PCB configurations as was done in FEA simulation. The most impactful PCB design consideration for reducing $R_{\theta JA}$ is including vias under the bump, followed by increasing copper thickness to 2 oz.



(a) (b)

Figure 11. (a): Baseline thermal simulation results. (b): Thermal results using all the available techniques

Table 2. Summary of PCB cooling techniques and their effect on $R_{\theta JA}$

Configuration	Design change	ΔT	$R_{\theta JA}$
Change in copper thickness	1/2 oz Cu	53°C Baseline	46°C/W Baseline
	2 oz Cu	44.6°C (-19%)	37.3°C/W (-23%)
Change in via design	Min vias	60.4°C Baseline	48.9°C/W Baseline
	Adjacent vias	49.6°C (-18%)	40.4°C/W (-21%)
	Vias under bump	43.8°C (-27%)	36.9°C/W (-32%)
Change in layout configuration	Adjacent FETs	46.6°C Baseline	40.6°C/W Baseline
	Spaced out FETs	44.6°C (-5%)	37.3°C/W (-9%)

Conclusion

An overview of PCB only thermal management strategies was presented that can be implemented to improve GaN FET cooling in those applications where heatsinking or external heat spreading cannot be used. These guidelines recommend using thicker copper layers, thermal vias under or near the devices, and separating the FETs to reduce co-heating. Combining these techniques results in a reduction of $R_{\theta JA}$ by over 35%, with little to no increase in overall system cost, while reducing peak operating temperatures by over 20°C with natural convection cooling. Overall efficiency of the converter increases too as the devices operated at lower temperature for the same operating conditions.

References

[1] A. Lidow, M. De Rooij, J. Strydom, D. Reusch, and J. Glaser, GaN Transistors for Efficient Power Conversion, 3rd ed. John Wiley & Sons, 2019. ISBN: 978-1119594147.

[2] Efficient Power Conversion Corporation EPC2218 eGaN FET datasheet. [Online] Available: https://epc-co.com/epc/Portals/0/epc/documents/datasheets/EPC2218_datasheet.pdf

[3] AN-2020 Thermal Design by Insight, Not Hindsight," Texas Instruments Application Report, Apr. 2013. [Online]. Available: https://www.ti.com/lit/an/snva419c/snva419c.pdf.

[4] IPC-6012E: "Qualification and Performance Specification for Rigid Printed Boards," IPC, March 2020

[5] IPC-4761, "Design Guide for Protection of Printed Board Via Structures," IPC, July 2006.

[6] Efficient Power Conversion Corporation EPC2619 eGaN FET datasheet. [Online] Available: https://epc-co.com/epc/Portals/0/epc/documents/datasheets/EPC2619_datasheet.pdf

[7] Y. Shen, H. Wang, F. Blaabjerg, H. Zhao and T. Long, "Thermal Modeling and Design Optimization of PCB Vias and Pads," in IEEE Transactions on Power Electronics, vol. 35, no. 1, pp. 882-900, Jan. 2020.

[8] Efficient Power Conversion Corporation, "EPC90123: 100 V, 25 A Half Bridge Development Board," [Online] Available: https://epc-co.com/epc/products/evaluation-boards/epc90123

[9] Efficient Power Conversion Corporation, "EPC9097: 100 V, 20 A Half-Bridge Development Board," [Online] Available: https://epc-co.com/epc/products/demo-boards/epc9097

[10] Efficient Power Conversion Corporation, "EPC90123: 100 V, 25 A Half-Bridge Development Board," [Online] Available: https://epc-co.com/epc/products/demo-boards/epc90123

[11] D. Reusch, J. Strydom, "Understanding the Effect of PCB Layout on Circuit Performance in a High Frequency Gallium Nitride Based Point of Load Converter," Applied Power Electronics Conference, APEC 2013, pp. 649–655, 16–21 March 2013.

[12] J. S. Glaser, A. Helou, "PCB Layout for Chip-Scale Package GaN FETs Optimizes Both Electrical and Thermal Performance," Applied Power Electronics Conference, APEC 2022, pp. 991–998, March 2022.