Assembling eGaN® FETs and Integrated Circuits

EFFICIENT POWER CONVERSION

When customers express wishes for better packaging of power semiconductors – transistor, diode, or integrated circuit – their requests fall into six categories [1]:

1. *Can the package be smaller?*
2. *Can package inductance be reduced?*
3. *Can the product be made with lower conduction losses?*
4. *Can the package be more thermally efficient?*
5. *Can the product have a lower price?*
6. *Can the package be more reliable?*

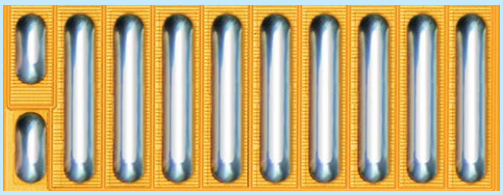


Figure 1(a): Mounting side of EPC2001C [2].

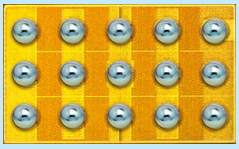


Figure 1(b): Mounting side of EPC2045.

eGaN® FETs and integrated circuits from EPC have taken a very different approach to packaging power semiconductors – we have ditched the package altogether and thus improved all six of the above requests at the same time. EPC's innovative wafer level, Land Grid Array (LGA) and Ball Grid Array (BGA) packaging has enabled a new state-of-the-art in power density [2]. Figure 1 shows a photograph of the mounting side of an EPC2001C, which has a land pitch of 0.4 mm and an EPC2045 with a ball pitch of 0.5 mm.

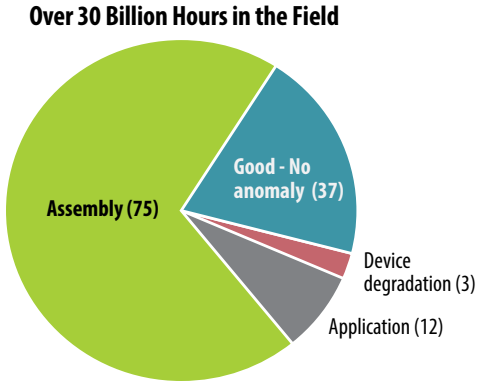


Figure 2: After more than 30 billion hours in actual applications, eGaN devices have experienced only three (3) device related failures. The largest cause of field failure is poor assembly.

Extensive testing has proven that eGaN FET and ICs are reliable [3] when designed into application circuits correctly. In fact, between 2010 and 2017 there have been a total of only 127 device failures out of a total of more than 30 billion hours in actual use in the field, 75 of the failures were the result of poor assembly technique or poor printed circuit board (PCB) design practices [3]. The graph of figure 2 shows a breakdown of those 127 failures.

Ensuring high reliability and to extract maximum performance from eGaN devices, it is important to follow some simple PCB design and assembly guidelines, which are presented in this application note along with examples of what can go wrong if those guidelines are not followed.

NOTE: EPC should be contacted to assist in the development of any stencil thickness/solder combination that deviates from those recommended here. **EPC WLCSP die are not compatible with wave soldering process technique.** Contact EPC via email at info@epc-co.com.

Overview

For a reliable, high yielding assembly, LGA or BGA eGaN devices must:

1. Have the correct PCB solder mask defined (SMD) footprint for each solder bump to ensure proper containment of solder on a clean PCB surface.
2. Have the correct solder volume and reflow process to provide sufficient height for proper rinsing of any solder flux from between the lands, but not excessive solder, where the joint becomes unstable and tilts or collapses during reflow.
3. Have all flux rinsed from between lands and be dry before applying power.
4. Use underfill if the assembly will be exposed to moisture during use.

Figure 3 shows a side view of a properly mounted device.

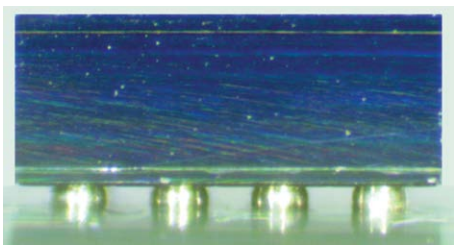


Figure 3: Properly mounted eGaN device (side view).

Printed circuit board design considerations for eGaN FETs and ICs

Power designers may not be as familiar with fine pitch devices as their digital circuit counterparts. This section will cover the many design aspects designers need to consider when designing a footprint for eGaN devices.

The quality of the solder bump interfacing the eGaN device to the PCB is crucial for a reliable electrical, thermal, and mechanical connection. The factors that define solder bump quality include symmetry, volume, height, and finish. These factors may be overlooked by a designer who is focused primarily on layout, thus the device manufacturer must take on the responsibility for providing clear and simple guidelines in their product datasheets. Those guidelines include the recommended footprint (copper dimension and solder mask opening) and solder paste stencil designs.

1. Solder mask defined pads

For eGaN devices, a solder mask defined (SMD) footprint for the LGA and BGA bumps is recommended as shown in figure 4 (right). Figure 4 (left) shows a non-solder mask defined (NSMD) footprint pad that is typically used for PCB deigns. In our investigations of failures at customers' assembly facilities, we have found instances where PCB manufacturers modified Gerber files to accommodate their internal manufacturing design guidelines developed for much larger packages thus over-riding the design provided.

A design review prior to final board release would highlight this problem prior to incurring scrap or rework expenses in assembly. Figure 5 shows how the SMD pad works to reduce mechanical stress by ensuring a symmetrical bump after soldering. In contrast, using an NSMD footprint can result in an asymmetrical solder bump since 100% perfect registration between the copper and mask layers in not likely. In the case of the SMD footprint, immunity is ensured within the manufacturing tolerances of the PCB.

2. Solder mask quality

Not all solder masks are the same, and it is important to know what to look for when specifying the solder mask to yield a high-quality PCB with the thickness and consistency being the most important. If the solder mask is too thick, it will be difficult to properly dispense the solder paste as the distance into which the paste needs to be pressed becomes larger.

If the consistency of the solder mask is not uniform, it can lead to bumps that prevent the stencil from being seated properly against the board. Solder mask defects, or excessive solder mask openings, can result in reduced bump height and lead to cracking and/or die tilt or even open circuits in extreme cases. The resultant deformed bumps will cool with increased mechanical stress that can accelerate thermally induced failures.

Suitable solder masks for PCBs employing eGaN devices fall under IPC-SM-840 class T, such as Taiyu 4000HFX L.P.I, PSR-2000/LF02/CA-25, or equivalents. It is important to specifically state in the PCB fabrication files that the solder mask not be enlarged or modified by the PCB manufacturer.

Laser Direct Imaging (LDI) should be used to register the solder mask to a tolerance as specified in the master drawing or 2 mils with respect to the copper layers. Finally, the solder mask should not be clipped. Clipping places greater emphasis on the designer to ensure that the layout software has the correct design rules setup and that the footprint is correctly designed.

3. Solder bump volume

The height of the solder bump between the board and the device is also critical for mechanical stress. The height of the solder bump has been determined to influence the balance between reliability, electrical, and thermal performance. If the bump height is too low, the device will experience high thermally induced mechanical stresses that will result in solder ball or under bump metal fatigue. Devices seated too high will experience higher electrical and thermal stresses (due to reduced thermal extraction effects from the PCB) [3].

Optimal height is different for each device. For LGA and BGA devices, additional options such as various solder types must be part of the design decisions. Figure 6 shows how different solder types can yield different stencil designs. It should be noted that if a customer desires to use the same solder mask for both Type 3 and Type 4 solder, EPC can work with the customer to provide a recommendation that can possibly work for both solder types if possible. Such recommendations will also be shown in the datasheet.

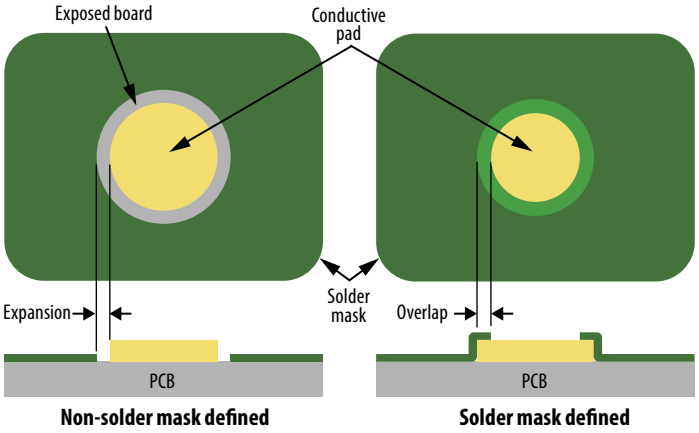


Figure 4: Solder mask defined versus non-solder mask defined pad.

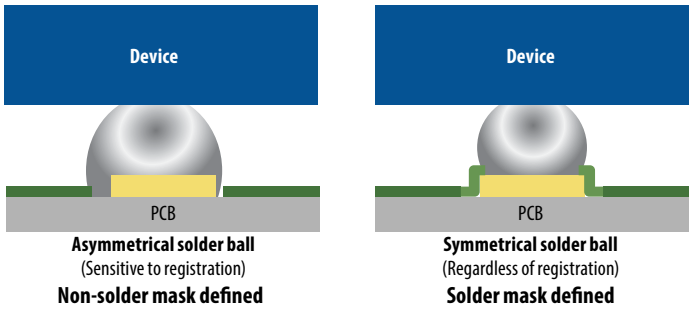


Figure 5: Effect of copper to solder mask layer registration on the solder ball symmetry.

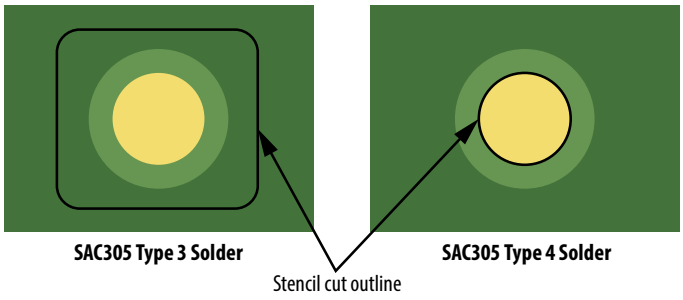


Figure 6. Impact of solder type on stencil design.

4. Solder pad finish

Many designers opt to use a hot air solder leveling finish (HASL) for their boards, which deposits solder on the pads. These solder deposits yield small amounts of solder on the pads that add to the amount of solder that will be dispensed via the stencil. This added solder will be included in the bump solder volume, ultimately affecting the finished bump volume and hence its quality.

In addition, the HASL process is imprecise and typically yields uneven solder on the pads of varying quantities, as can be seen in figure 7 (right). This can lead to die tilt or open solder joints. It is recommended to use an electroless nickel immersion gold (ENIG) pad finish that yields a very uniform and flat pad as shown in figure 7 (left).

For the ENIG finish, a typical nickel (Ni) thickness of 150 micro inches, and a gold (Au) thickness of 3 - 5 micro inches in accordance with IPC-4552 are recommended.

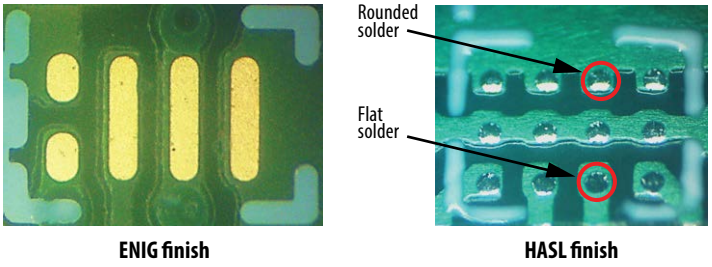


Figure 7: ENIG finish versus HASL finish showing uneven solder heights.

5. Silkscreen

While silkscreen has traditionally played a minor role in PCBs, it can be a part of the reliability function because it does not have zero thickness and, as a result, can impede the flow of flux during reflow process.

If the flux flow is impeded during reflow, it can lead to die tilt and flux residue. The silkscreen is also used to properly register the die during the assembly process, and many designers will tend to design a fully enclosed silkscreen pattern outlining the device as shown in figure 8 (right). This can lead to damming when the flux cannot flow out from under the die during reflow, and is particularly acute with a thick silkscreen that extends around all or most of the die. Flux damming can result in uncured flux being present under the die and can lead to thermal and electrical dendrite formation (See troubleshooting section below).

Simply opening the silkscreen walls at various locations as shown in figure 8 (left) can prevent damming. Silkscreen can also be a source of pad contamination and care should be taken to avoid this. The silk screen should also be kept as thin as possible, as far below 1mil that the vendor can do (probably in the 0.7mil range).

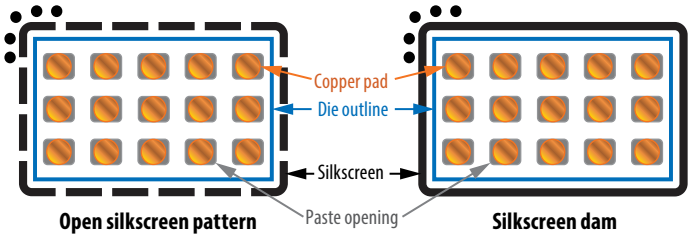


Figure 8: Device land pattern with open device outline silkscreen (left) and dammed-closed silkscreen device outline (right).

6. Vias

Vias form an integral part of the PCB design for eGaN devices due to their small size and electrical performance requirements [4]. Via dimensions are at the discretion of the designer who needs to be aware of several limitations depending on how the via is used.

The basic via is a vertical connection between the layers of the PCB and is made up of a hole with annular ring of copper. Manufacturing restrictions limit hole sizes to a minimum of 6 mils and the minimum annular ring dimension of 5 mils. Designers may recognize that this already exceeds the dimensions of some eGaN device bump spacing and a compromise in one direction may be made. Vias that are near the die should always be tented (covered with the solder mask) to prevent solder from wicking into the hole during the reflow process, and to prevent voltage clearance issues due to exposed copper in proximity to the die.

In some cases, it may be necessary to place a via under a device pad. If this is required, the via must be filled and capped to prevent the solder required for optimal bump height from draining into the hole during reflow. Capping is required to prevent the filler from outgassing under the solder bump. This via should be tented in the layout design software so that the pad solder mask opening, should it overlap the via, will determine which part of the via to cover or expose. The finished via must have essentially the same height as the remainder of the pad. This will ensure proper solder paste dispensing.

Some designs may require many vias, which are used to distribute the current and increase the total current transferred to another layer. Despite the vias being connected to the same electrical node, manufacturing restrictions require at least 10 mil wall-to-wall spacing for vias to prevent weaknesses in the board and ensure a uniform board thickness finish. Tented vias near, or within the device, can lead to high spots that cause the stencil to sit up higher and may lead to solder over-deposit during manufacturing. When using vias close to or within the device, make sure the PCB manufacturer is aware of your stencil seating concerns.

Some designs may have minimum via hole size restrictions, which means placing vias under the pads of the device is not possible and must be placed next to the device such as the example design shown in figure 9 (top). Figure 9 (bottom) shows a design without the via in pad restrictions where the photo clearly shows that the FET pads have been plated over and the via can no longer be seen.

7. Layer registration

The many layers that make up the PCB need to be aligned (registered) with each other. This ensures a proper functional board and is specified by providing a registration tolerance. Layers of the same type are typically specified differently from layers of different types. Layers typically registered with each other are copper-to-copper, copper-to-solder mask, copper-to-hole, and copper-to-silkscreen (not typically specified but is helpful for die placement).

Most layers can be specified with a registration tolerance of 2 mils with respect to a copper layer, except holes, which should use 3 mils minimum.

8. Layer stack-up

A stack-up defines the thickness of each of the layers that make up the PCB. PCB's for eGaN FETs and ICs typically use copper thicknesses between 1 and 2 oz (35 & 70 µm), depending on the design and the current density required. The insulating layers are typically made using materials such as FR4 or FR370-HR. Substrates with higher glass transition ratings (T_g of at least 180°C) are preferred for higher reliability.

The balance between manufacturability and electrical properties drives the thicknesses of the insulating layers. The optimal layout for 100 V devices specifies a 5-mil core thickness between layers 1 (e.g. top) and 2. Due to symmetry requirements, this will also force layers 3 and 4 to be 5-mils thick with the prepreg layer being adjusted to meet the final board thickness (typically 1.5 mm or 62 mils). For higher voltage devices, and to ensure proper "creepage" requirements, a minimum core thickness of 12 mils is recommended. The prepreg layer will then adjust to approximately 25 mils. Figure 10 shows a typical PCB stack-up.

9. Board flatness

A lesser-known PCB specification is board flatness. It is still an important specification because a board with excessive bow can prevent the stencil from being properly seated on the board, potentially leading to pads not being properly dispensed with solder paste. An array with maximum horizontal or vertical dimensions of 200 mm (8000 mils) should be specified with a flatness to be within 40 mm per meter (7.5 mil per inch).

10. Fiducials

Fiducials are used for component registration during the placement process of assembly. Due to the fine pitch of eGaN devices, it is typically required to add fiducials to the PCB. There are two types of fiducials, global and local. Global fiducials are used to align a PCB array and local fiducials are used on a single board. Most designs will require local fiducials to accommodate eGaN devices.

A typical board should have at least three (3) fiducials with at least two (2) aligned vertically and two (2) aligned horizontally. Fiducials should be placed as close to the board edge and as far apart from each other as possible. This placement improves the registration over longer distances. If the eGaN devices are located more to one side of the board, then the fiducials should be located in close proximity to one another. A 40-mil diameter fiducial should be sufficient for most assemblers.

11. Board or array size limits

PCBs and PCB arrays using eGaN devices should be limited in size. Larger boards are more difficult to register for precise assembly of the eGaN devices with their small feature size. Boards and arrays should be limited to 200 mm on each side. For arrays, it is recommended to rotate the boards in an attempt to locate all of the eGaN devices of the various boards as close to each other as possible. This allows larger boards to be used, then the registration location can still be located as close as possible to the eGaN devices.

Assembly process for eGaN devices

The discussion up to this point has focused on the PCB design and its manufacturability. In this section, the assembly process that goes hand in hand with the PCB will be presented.

12. Solder paste choice

EPC currently uses Kester NXG1 Type 3 SAC305 and Kester NP505-HR SAC305

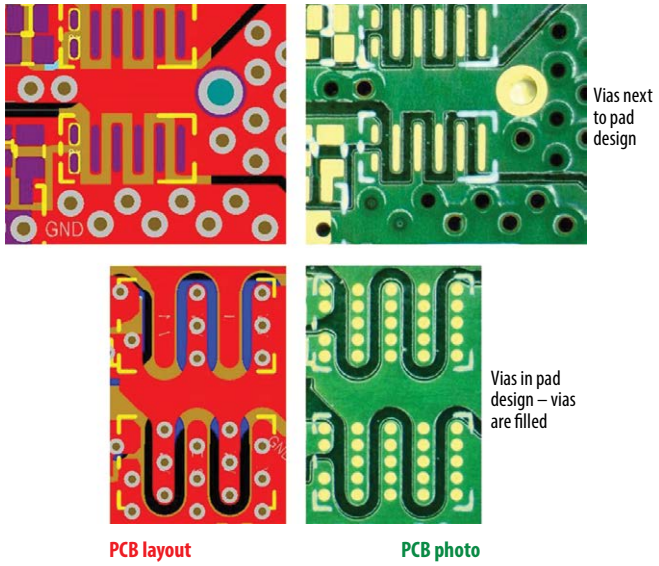


Figure 9: Layout design with vias next to the FETs (top) and within the pads of the FET (bottom).

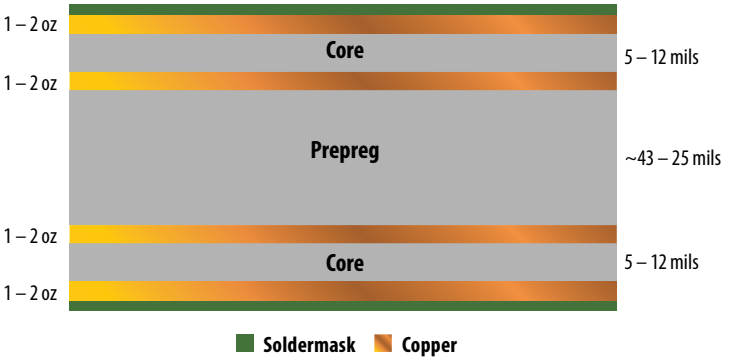


Figure 10: PCB stack-up for a typical PCB suitable for eGaN devices.

Type 4 solder pastes for soldering eGaN devices. Both pastes are no-clean flux with 88.5% metal.

To prevent the formation of thermal and electrical dendrites, it is recommended to clean the flux from the board even if no-clean flux is used. EPC uses Kyzen Aquanox® A4625 chemical in a Nu/Clean AquaBatch XL standard system manufactured by Technical Devices Company to remove the no clean flux.

If a no-clean flux is used and it is not rinsed off, a post-reflow bake for a minimum of 60 minutes at 150°C is recommended. This ensures that the no-clean flux is properly cured and helps prevents dendrite formation.

If a water rinseable flux is used, the eGaN device needs to be rinsed on all four sides to ensure proper flux removal. A tilted device can obstruct the flow of the rinse and cause flux to remain trapped under the die. For this reason, using a no rinse solder flux with low ionic content and then rinsing the no rinse flux is recommended.

13. Stencil design

A laser-cut stainless steel stencil of 100-µm thickness is recommended. A smooth wall laser-cut stencil is more likely to release the desired dispense volume. Type 3 solder paste requires a larger opening than Type 4 solder,

and recommendations are available for both in reference [15], and for each die configuration. In the case that a stamped stencil must be used, it may be necessary to enlarge the opening slightly to compensate for proper solder release volume.

14. Reflow profile

Figure 11 shows the recommended reflow profiles for eGaN devices based on the solder paste manufacturers' recommendations for the pastes. The vendor-recommended reflow profiles should always be followed for the paste being used.

Troubleshooting PCB design and assembly problems

There are many issues that can arise if proper PCB design rules not followed. These issues can be exacerbated by poor assembly techniques. This section presents the many issues we have encountered and explains their origins.

Die Placement Pressure

Die damage due to over pressure can occur. EPC recommends limiting maximum backside pressure to 50 psi or less. Please see **Reliability Report Phase 11** for more information.

15. Electrical dendrites

Electrical dendrites are considered ionic contamination formed when the flux is exposed to an electric voltage and forms conductive crystals [5]. The higher the voltage the faster dendrites can form. Electrical dendrites can quickly lead to failures because during their formation they can generate a lot of heat in addition to creating short circuits. Figure 12 shows an example of dendrite formation around a solder bar of an eGaN FET.

16. Thermal dendrites

Thermal dendrites are a relatively new discovery and are not to be confused with electrical dendrites, although they may appear similar. They are caused by flux cracks formed during furnace cool down. Solder, which has not cooled can diffuse into the flux cracks. These solder filaments can significantly reduce the electrical distance between bumps and can breakdown during operation. Flux cracks can remain after the assembly process even when using no-clean flux. Figure 13 shows a flux crack on a PCB formed in the absence of a die.

Thermal dendrites are slightly different from traditional thermal dendrites seen in metal melts where the presence of a nucleation source can lead to "arms" growing from the particles in a super-cooled liquid forming dendrites [6,7,8]. This type of dendrite can be prevented or removed by cleaning the no-clean flux.

Examples of thermal dendrites are also shown in figure 14.

17. Poor solder adhesion

Contaminated solder pads can lead to poor solder adhesion, voiding, and large un-wetted areas. Most contamination arises from a poor PCB fabrication process where solder mask residue is left on the pads, for example from contaminated cleaning liquids. Solder mask bleed and silk screen residue are also common sources of pad contamination.

Figure 15 shows an example of a clean pad finish (left) alongside an extremely contaminated pad finish (right). Poor solder adhesion can lead to open circuits, and in high-current capable devices, will lead to excessive current densities as currents are forced into undefined directions that will ultimately cause the device to fail. Figure 16 shows an example of a solder wetting issue.

18. Solder bump cracking

Solder bumps crack mainly due to thermally induced mechanical stresses. If the solder solidifies under stress during the assembly process, it can lead to accelerated failure as the solidified stresses are added to those induced by thermal expansion and contraction.

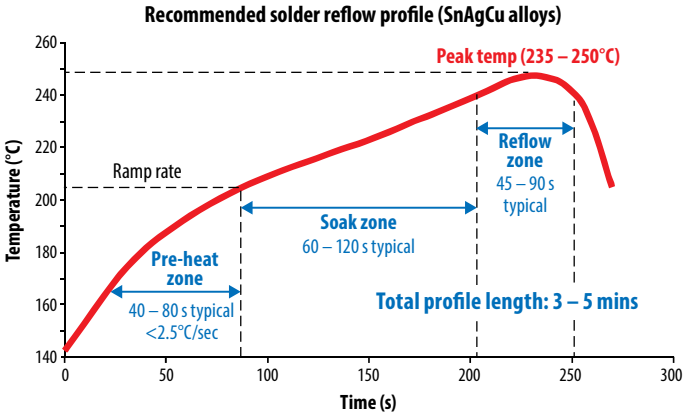


Figure 11: Recommended solder reflow profile.

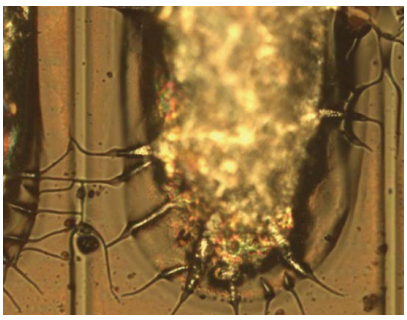


Figure 12: eGaN FET showing electrical dendrite formation after exposure to residual flux.

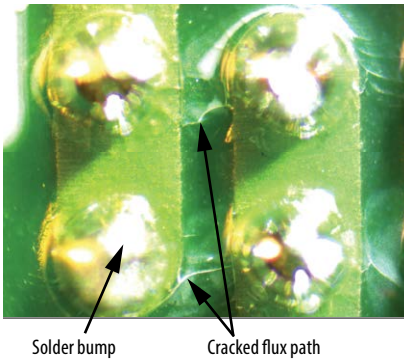


Figure 13: Example of thermal dendrite formation path in cracked residual flux without a die present.

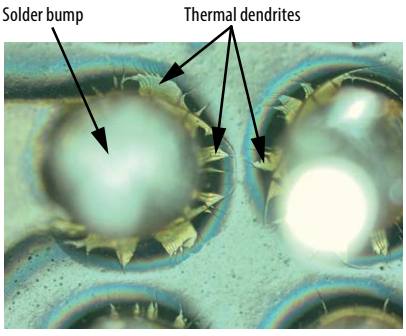


Figure 14: eGaN IC showing thermal dendrite formation in cracked residual flux.

A well-known industry analytical model [9] for strain on solder joints during thermal cycling is:

$$\varepsilon = \frac{(\Delta\alpha \cdot \Delta T \cdot DNP)}{h} \quad (1)$$

where:

ε = Strain on solder joint

Δα = Difference in coefficient of thermal expansion between die and PCB [°C^{-1}]

ΔT = Cyclic temperature swing [°C]

DNP = Distance from neutral point (stress centroid based on die size & solder bump/bar locations) [m]

h = Solder joint standoff height [m]

Equation 1 suggests a very tall solder bump height will reduce the thermally induced stress to near zero, which is not always practical. The balance between thermally induced stresses and reliability is typically determined by the manufacturer of the devices and given in product datasheets. In the case of eGaN devices, the solder mask-defined pad further reduces the stresses induced during the reflow process.

An example of solder bump cracking is shown in figure 17.

19. Solder voids

Solder voids are open volumes within the solder as shown in figure 18. Solder voids can have various causes including poor solder adhesion to the pad, outgassing from contaminated pads during reflow, insufficient device standoff height [10], and incorrect solder profile.

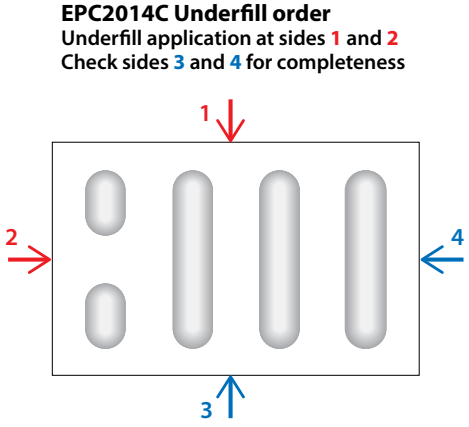
Voids reduce the contact area between the device and PCB pad and induce uneven mechanical and thermally induced stresses within the solder bump. Over time, these voids can grow and lead to failure.

20. Die tilt

Die tilt can be caused due to several reasons such as poor solder adhesion, uneven solder paste dispensing, excessive vibration during reflow, non-optimized temperature profile, and oversized solder mask or oversized solder stencil apertures. Die tilt is detrimental to device reliability, as it causes uneven thermal mechanical stress across the die. It may also be an indication of short or open circuit bumps. Figure 19 shows an example of a tilted eGaN device.

21. Underfill

Underfill should be used in applications where the board is exposed to moisture. Moisture and other contaminants may provide an environment that allows dendrite growth. For 150°C-capable EPC devices, some available underfills are Hysol FP4531, Namics U8437-2, Namics 8410-406B, and Henkels Loctite Eccobond UF 1173. The die surface should be free of any flux residue before applying the underfill per the vendor recommendations. Bump layout directionality must be taken into consideration when choosing the sides to apply the underfill.



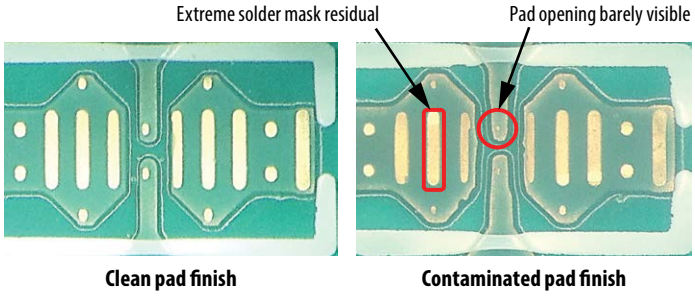


Figure 15: Examples of a contaminated pad versus clean pad.

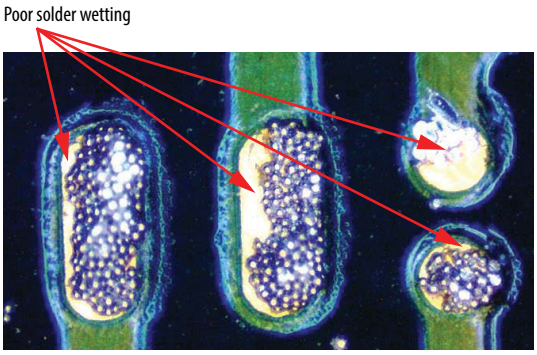


Figure 16: Example of poor solder wetting.

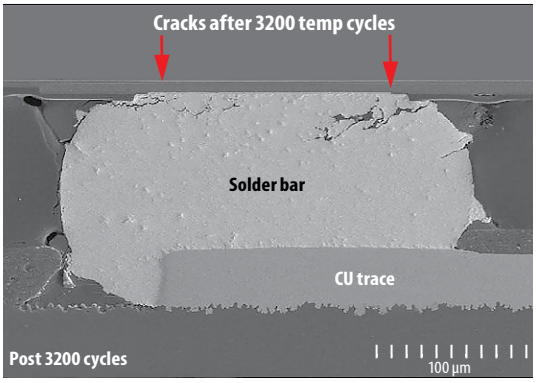


Figure 17: Cross section x-ray of an eGaN IC solder bump showing bump cracking.

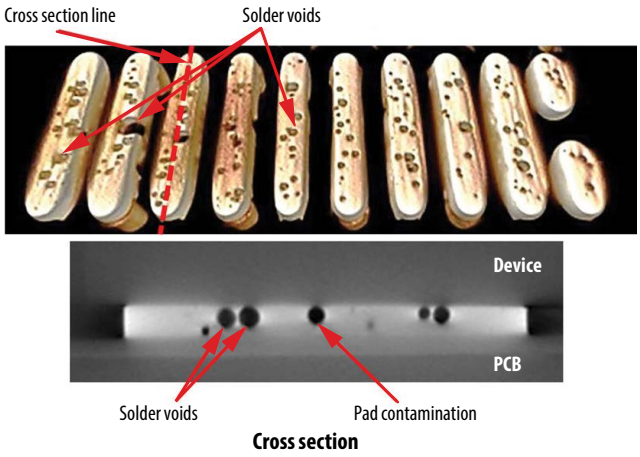


Figure 18: 3D-X-ray of an eGaN FET showing excessive solder bump voids (top) Contaminant on the pad to solder interface leads to voids formation (bottom).

22. Design example

Figure 20 is a drawing of a cross section of an EPC2001C with an LGA format (it is also valid for BGA footprints) on the NC257-2 SAC305 [13] Lead Free, No Clean Solder Paste mounted on the PCB with a two-ounce copper topside metal layer. The cross-sectional area will be used as a proxy for solder volume. Per the datasheet, the solder bump radius is 100 µm giving a cross sectional area of 15.7 nm^2. Using NC257-2 SAC305 lead free, no clean, type 3 solder paste with an 88.5% metal load, and 100-µm stencil thickness, the solder mask width comes to approximately 180 µm.

23. Inspection

eGaN devices are mechanically robust and have demonstrated high yield in volume assembly. Damage, however, can still occur if several standard precautions are not taken to ensure adequate solder reflow, reduce excessive die tilt, and avoid residual uncured solder flux.

Even though eGaN devices have been designed such that the reflowed solder is visible to the unaided eye, the best way to determine if devices have been properly reflowed is by producing X-ray images. Figures 21 and 22 show X-ray images of an EPC2019 assembled with a solder stencil process. Figure 21 shows an image with voids and uneven shaped joints indicating potential solder volume or reflow issue. As can be seen in figure 22, minimal voiding and consistent joints represent giving high reliability and excellent thermal and electrical characteristics.

24. Rinsing

If the assembly process uses a solder with a flux that requires rinsing, die tilt can obstruct the flow of the rinse and cause flux to be trapped under the die. This residual flux can cause rapid formation of dendrites (figure 12), which will cause early device failure. For this reason, using a no rinse solder flux with low ionic content and then rinsing the no rinse flux is recommended. Some EPC devices require rinsing in a specific manner to properly clean under the die. Attention should be given to properly rinse the part to remove the flux. The example shown in figure 23 is an EPC2001C type die that requires rinsing on a minimum of three sides to adequately clean the flux.

Figure 19: Side view of an eGaN device with severe die tilt after soldering.

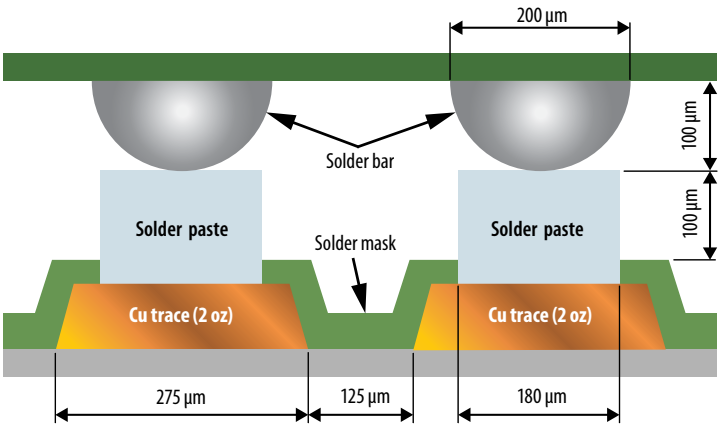


Figure 20: Cross section of a representative PCB mount using EPC2001C (Pre-reflow).

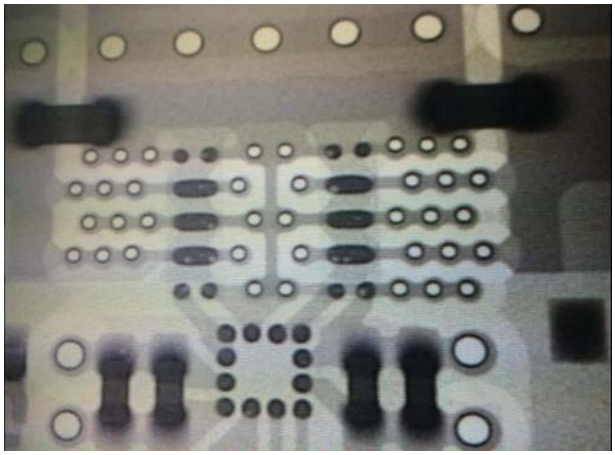


Figure 21: X-ray of a board showing the EPC2019 and the LM5113 driver. The image shows voiding in the solder bars and uneven looking joint shapes in the EPC part and driver. This is example of possible solder paste volume or reflow issue.

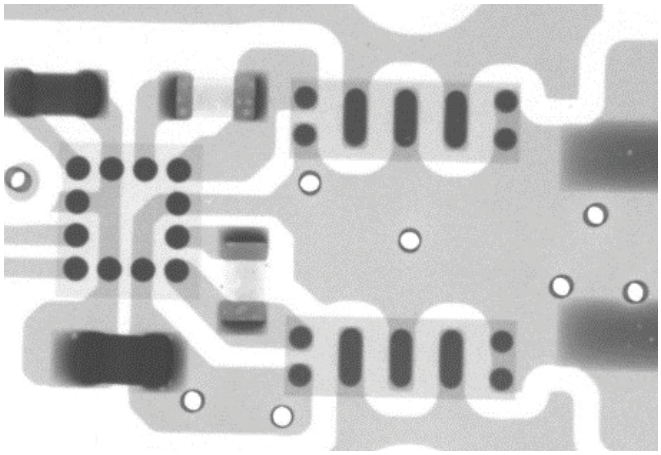


Figure 22: X-ray of a properly reflowed EPC2019 and LM5113 driver. No obvious voids in the EPC part or driver. Joints are a solid dark color and joint shapes are very consistent in size and even looking.

About LGA and BGA packages

Devices are lead and halogen free. The RoHS compliant LGA and BGA package uses a Sn/Ag/Cu solder with a composition of 95.5% Sn, 4% Ag, 0.5% Cu or a Sn/Ag solder with composition of 97.5% Sn, 2.5% Ag. All lead-free products are moisture sensitivity level 1 (MSL1 260°C), the highest commercial semiconductor level.

Normal manufacturing ESD precautions should be taken when handling EPC eGaN FETs and ICs. Recommended warehouse storage conditions for tape and reel: temperature 20°C to 28°C, Humidity 40% to 60%.

Quick-Start Engineering Lab Assembly

EPC's eGaN devices can be mounted directly onto PC boards without added solder by using a tacky flux to hold the part in place while reflowing the solder. An example of an acceptable Lead Free (PbF) process uses Kester TSF6502 no-rinse flux. Quick reference die attach and removal instructions, as well as videos, are available on the assembly page of the EPC website (http://epc-co.com/epc/DesignSupportbr/Applications/AssemblyBasics.aspx).

Summary

LGA and BGA packages provide the low inductance, small size, and excellent thermal performance necessary to take full advantage of the capabilities of eGaN technology. With proper manufacturing techniques, assemblies using eGaN devices will have high yield and a long, reliable working life.

The LGA and BGA device footprint must be solder mask defined. Solder mask designs should be according to the recommended land pattern on the datasheet.

Correct solder volume and reflow profile will help ensure mounted die are level with enough standoff for proper rinsing. Rinsing in all directions and drying are required to remove residue that would otherwise enable dendrite growth.

The reflow temperature profile must be adjusted to ensure complete reflow and to help avoid die tilt.

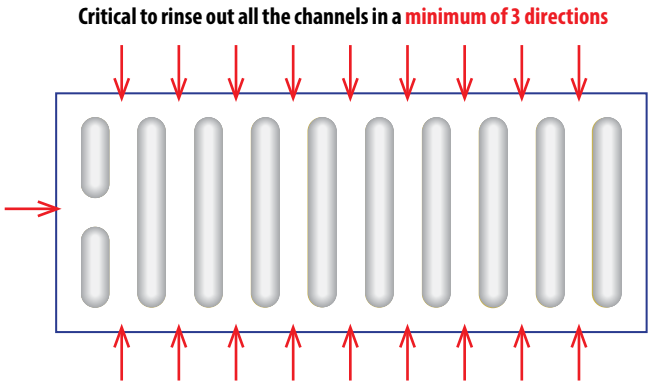


Figure 23: Example of EPC2001C die and minimum rinsing required to properly clean the flux.

REFERENCES

1. A. Lidow, "Six Reasons to Rethink Power Semiconductor Packaging," https://www.linkedin.com/pulse/six-reasons-rethink-power-semiconductor-packaging-alex-lidow
2. A. Lidow, J. Strydom, M. de Rooij, D. Reusch, *GaN Transistors for Efficient Power Conversion. Second Edition*, Wiley, ISBN 978-1-118-84476-2.
3. C. Jakubiec, R. Strittmatter, C. Zhou, "EPC eGaN® FETs Reliability Testing: Phase 8," 2016, [Online] Available: http://epc-co.com/epc/DesignSupport/eGaNFETReliability/ReliabilityReportPhase8.aspx
4. 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.
5. R. Ambat, M. S. Jellesen, D. Minzari, U. Rathinavelu, M. A. K. Johnsen, P. Westermann, P. Møller, "Solder Flux Residues and Electrochemical Migration Failures of Electronic Devices," Proceedings of the Eurocorr 2009, Paper No. 8141, 6-10 September 2009.
6. "Development of Thermal Dendrites," 366-2012-Lecture7, Course: MATE 366, School: Drexel University.
7. V. Pines, A. Chait, M Zlatkowski, "Thermal diffusion dominated dendritic growth – an analysis of the wall proximity effect," Journal of Crystal Growth 167 (1996) 383–386
8. R.F. Sekerka, S.R. Coriell, G.B. McFadden, "The effect of container size on dendritic growth in microgravity," Journal of Crystal Growth 171 (1997) 303–306
9. Denis Barbini, Ph.D. & Michael Meilunas, "Reliability of Lead-Free LGAs and BGAs: Effects of Solder Joint Size, Cyclic Strain and Microstructure," IPC APEX EXPO Conference Proceedings.
10. Muffadal Mukadam, Michael Meilunas, Peter Borgesen, Ph.D., K. Srihari, Ph.D., "Assembly and Reliability Issues Associated With Leadless Chip Scale Packages," October 2, 2006.
11. http://www.mv-group.biz/images/loctite.pdf?nconline=208d580d695356b06651cd3ccceaabfc
12. http://www.shinetsu.co.jp/encap-mat/e/product/k_s/smc/index.html
13. NC257-2 SAC305 Datasheet - http://www.aimsolder.com/sites/default/files/nc257_sac305_solder_paste_tds.pdf
14. M. de Rooij, A. Nakata, "Designing Manufacturable and Reliable Printed Circuit Boards Employing Chip Scale eGaN® FETs," International Exhibition and Conference for Power Electronics, Intelligent Motion, Renewable Energy and Energy Management (PCIM Europe), May 2017, pp 1399–1406.
15. A. Lidow, J. Strydom, M. de Rooij, Y. Ma, *GaN Transistors for Efficient Power Conversion. First Edition*, Chapter 10, Power Conversion Publications, ISBN 978-0-615-56925-3.

Revised July, 2021