

Power Converter Module Package Process Specification for EN5310, EN5330, and EN5360 Converter Modules

Introduction

This applications note addresses the following topics:

- PCB Design Recommendations:
 - Pad Layout
 - Pad Design
 - Pad Plating
- Solder and Reflow:
 - Solder Paste Deposition
 - Package Placement
 - Solder Reflow
 - Solder Joint Inspection
- Rework and Repair:
 - Package Removal
 - Site Preparation
 - Solder Paste Deposition
 - Package Placement
 - Package Reflow
 - Solder Joint Inspection
- Summary and Figures

Reference Documents:

- EN5310 1A POL DC-DC Module Data Sheet
- EN5330 3A POL DC-DC Module Data Sheet
- EN5360 6A POL DC-DC Module Data Sheet
- IPC/JEDEC J-STD-020C

Enpirion's power converter packages are designed using module package technology that utilizes laminate substrates and mold caps with System in Package (SiP) construction to form a Land Grid Array (LGA) package. LGA module package technology is ideal for power conversion devices resulting in a highly reliable package that provides many advantages over conventional leaded packages, including a smaller package thickness profile and footprint, lower package thermal resistance, broader heat dissipation from the package, large lead size and pitch, and excellent lead co-planarity.

In order to ensure high performance and reliability of Enpirion's power converters, this application note provides the recommended process specifications for assembly including PCB design, soldering and reflow, and rework.

PCB DESIGN RECOMMENDATIONS

Please refer to the specific product data sheet for package dimensions, pin-out, and specific layout information. The PCB design recommendations presented here provide general design guidelines for power converter LGA module package assembly.

Pad Layout

Figure 1 shows a typical PCB pad layout for a power converter LGA module package. The dashed line represents the outline of the package body.

Recommendation: The pad lengths should extend beyond the package body by at least 0.20mm (8 mil) to allow for package placement tolerances and to provide additional solder wetting for solder joint reliability, inspection, and lead probing.

Pad Design

Solder Mask Pad Definition

In PCB design, the surface mount solder pads can be defined as either Solder Mask Defined (SMD) or Non-Solder Mask Defined (NSMD). The difference between these two solder mask pad definitions is in the proximity of the solder mask to the metal pad. In SMD pads the solder mask opening is smaller than the metal pad and overlaps the metal on all sides. The solder mask opening defines the solderable area of the pad. In NSMD pads the solder mask opening is larger than the metal pad and does not overlap the metal. The metal edge defines the solderable area of the pad. Figure 2 shows the details of SMD and NSMD pads.

Since the metal etching process in PCB manufacture has significantly tighter alignment and etch tolerances than the alignment registration of the solder masking process, which is typically ± 0.075 mm (3 mil), a more accurate solder pad land pattern can be obtained with NSMD pads. Likewise, with SMD pads, the solder mask that overlaps the metal pad introduces additional height above the metal surface that may affect solder joint adhesion and reliability.

Recommendation: The NSMD pad definitions should be used on the PCB pad layout for a power converter LGA module package. To avoid any potential overlap with the metal pad due to solder mask registration, the solder mask opening must be defined at least 0.075 mm (3 mil) larger than the metal pad on all sides.

Solder Pad Land Pattern

Each Enpirion power converter LGA module package has two rows of symmetric solder pads along the long edges of the module for signals and two larger solder pads for GND

and VOUT. (Note: Refer to the specific product data sheet for the solder pad land pattern.)

Recommendation: The PCB pad layout should conform exactly to the power converter module package pads as defined in the product data sheet. The 0.20 mm (8 mil) extended pad length defined in *Pad Layout* Section above should be applied. This will provide the highest solder joint reliability between the power converter module package and the PCB.

Design variation in the PCB pad layout is allowed for any optional probe pads adjacent to signal pads that extend outside the power converter module package footprint.

Recommendation: The probe pads should be designed with a small trace from the signal pad connecting the probe pads as shown in Figure 3. This trace must be at least 0.20 mm (8 mil) long and covered with solder mask to prevent solder from flowing onto the probe pad.

Pad Plating

A uniform metal plating thickness on the PCB will ensure reliable, high yield module package solder assemblies yield.

Recommendation: The following plating guidelines for various PCB metal plating processes should be used:

- For an Electroless, Nickel-Immersion, Gold (Au) finish (ENIG), to prevent solder joint embrittlement, the thickness of the Au must be between 0.05 μm to 0.20 μm .
- A PCB with an Organic Solderability Preservative (OSP) coating is recommended as an alternative to an ENIG plating
- For a PCB with a Hot Air Solder Leveling (HASL) finish, the surface finish should be maintained within a 28 μm range.

SOLDERING AND REFLOW

Solder Paste Deposition

Since the Enpirion power converter module package is an LGA package, a stencil-printing process will be required for deposition of solder paste to the PCB for reflow of the module package to the PCB. The stencil-printing process requires the use of an aperture based metal stencil where solder is transferred through the apertures onto the solder pads of the PCB. To minimize solder voids and ensure maximum electrical and thermal connectivity of the package to the PCB, large pads, solder volume, solder leaching must be considered in the stencil design. The design and fabrication of the stencil determines the quality of the solder paste deposition onto the PCB and the

resulting solder joint after reflow. The primary stencil parameters are aperture size, thickness, and fabrication method.

Solder Stencil

Recommendation: The stencil aperture openings for all signal pads should have a 1:1 size ratio with the signal pads of the module package and the large internal pads (GND and VOUT) the stencil aperture openings should have a 0.95:1 size ratio with the pads on the module package. For pads longer than 4mm the stencil aperture openings should be divided into at least two or more smaller apertures with a 0.80:1 size ratio with the pads on the module package.

Recommendation: The stencil should be made from stainless steel and have a thickness of 0.127 μ m (5mil).

Recommendation: The stencil should be fabricated by chemical etching with electropolishing or by laser cutting. A tapered wall (up to 5°) on the apertures will facilitate solder paste release when the stencil is lifted from the PCB.

Solder Paste

Various types and grades of solder paste can be used for surface mounting the power converter module package. For leaded applications, a Sn-Pb solder can be used and for leadfree application a Sn-Ag (SA) or Sn-Ag-Cu (SAC) solder can be used.

Recommendation: Any Type 3 solder paste that is either water-soluble or no clean is acceptable.

Package Placement

The power converter module package is placeable onto the PCB using industry standard component pick-and-place systems that have a placement accuracy of ± 0.05 mm (± 2 mil).

Recommendation: For higher placement accuracy a component pick-and-place system with a top-and-bottom vision system should be used.

Recommendation: The module package should be released between 25—50 μ m (1—2 mil) into the solder paste.

Solder Reflow

The power converter module package may be surface mount soldered using standard IR or IR convection SMT reflow process. The power converter module package is qualified for a maximum of three (3) reflow cycles at 260°C peak reflow temperature according to the IPC/JEDEC J-STD-020C standard.

Recommendation: When using a solder paste with a no-clean flux a Ni (nitrogen) purge should be used during reflow.

The thermal reflow profile for component placement onto a PCB and the actual temperature of the component is dependent on various factors including: PCB thickness; PCB Cu (copper) weight; component density; component location; and size and thermal mass of other surrounding components. When using an IR reflow oven the location of other larger components can cause shadowing onto a specific component.

Recommendation: The thermal reflow profile for the power converter module package should be determined for each specific location on the PCB.

Solder Reflow Profile

Typical solder reflow profiles for both leaded and leadfree solders are shown in Figures 4 and 5, respectively. Specified time and temperature parameters are show below in Tables 1 and 2.

Table 1 Recommended Solder Reflow Profile Parameters for Leaded Solder

Leaded Solder	Reflow Parameters		
Reflow Profile Segment	Min	Recommended	Max
Preheat Temperature (°C)	100	140	150
Preheat Time (sec)	60	110	120
Ramp Up Rate (°C/sec)	1.5	2	3
Dwell Time above 183°C (sec)	50	75	85
Peak Temperature (°C)		215	240
Dwell Time at Peak (sec)	1	5	10
Ramp Down (°C/sec)	3	4	6

Table 2 Recommended Solder Reflow Profile Parameters for Lead-Free Solder

Lead-Free Solder	Reflow Parameters		
Reflow Profile Segment	Min	Recommended	Max
Preheat Temperature (°C)	150	175	200
Preheat Time (sec)	60	130	180
Ramp Up Rate (°C/sec)	2	3	4
Dwell Time above 217°C (sec)	60	90	120
Peak Temperature (°C)		250	265
Dwell Time at Peak (sec)	10	12	20
Ramp Down (°C/sec)	3	4	4

Solder Joint Inspection

An inspection of the solder joint between the solder pads of the power converter module package and the PCB should be performed.

Recommendation: Sample inspection monitoring at regular intervals during solder reflow should be performed of the solder joint between the solder pads of the power converter module package and the PCB.

The best visual inspection tool for inspection of the power converter module package solder joint on the PCB is a transmission X-ray, which can identify defects such as solder bridging, shorts, opens, and large voids (Note: small voids in large solder joints are not detrimental to the reliability of the solder joint). An additional visual inspection tool consists of side view inspection, such as 90° mirror projection, to determine component flatness and solder joint volume.

Recommendation: If possible, both transmission X-ray and a side view inspection process should be performed for visual solder joint inspection.

REWORK AND REPAIR

In the event that a power converter module package needs to be reworked (replaced) on the PCB, the process and procedures detailed in this section should be used. An LGA module package requires special process considerations for proper rework, particularly, precise process control for heat application for reflow of the module package solder joints.

Package Removal

If the power converter module package is being reworked due to failure, a heat profile may be used that has faster ramp rates and shorter time at reflow. When a power converter module package will be reused after rework, a heat profile that is near identical to the installation profile must be used. The heat should be applied locally to the module package to be reworked to prevent solder reflow of other adjacent packages or components. Special reheat equipment with nozzles to direct the heat to the module package should be available.

Removal of the power converter module package from the PCB requires heating the solder joints above the liquidus temperature of the installation solder.

Recommendation: The PCB should be baked at 125°C for four (4) hours prior to rework to remove any moisture in the PCB assembly to prevent moisture induced cracking or delamination of the PCB.

Reheat Equipment

To provide local heating only to the module package to be reworked special reheat equipment should be used. These include a hot air or hot gas reflow system with a gas focusing nozzle that is sized to the module package dimensions and a preheater, such as a convection heater, hotplate, or IR heater. To lift the module package from the PCB equipment fitted with a vacuum nozzle should be used. Also, a vacuum pen may be used to lift the module package.

Module Package Reheat and Lift

Proper preheating of the PCB and localized reheat of the module package are necessary for successful removal of the module package. Removing the module package while only applying heat onto the top surface of the module package will result in damaged or lifted land pads on the PCB and will overheat the rework site exceeding recommended heating ramps for the PCB.

Recommendation: The rework site of the PCB around the module package should be preheated from below to at least 100°C with a convection heater, hotplate, or IR heater to better control the module package removal process.

Recommendation: If an adjacent package or component is within 1.27 mm (50mil) of the module package, a custom nozzle must be used to direct heat only onto the module package and prevent solder reflow of the adjacent package or component.

All module package solder joints must be liquidus before the module package can be lifted from the PCB. An automated vacuum nozzle or a vacuum pen should be used to lift the module package from the PCB.

Recommendation: The module package should be lifted straight from the PCB without any tilting. It is beneficial to use equipment with an automated vacuum nozzle to lift the module package. A hand-held vacuum pen may result in tilting.

To ensure that the correct heat profile is used for module package removal, thermocouples should be mounted on the PCB near the module package for temperature monitoring.

Site Preparation

After the module package has been lifted from the PCB, any residual solder on the land patterns must be removed to prepare the rework site for new module package reflow.

Solder Removal Equipment

To remove any residual solder from the PCB land patterns conductive desoldering tools, Teflon tipped vacuum wands, and Cu (copper) de-soldering braids (solder wicks) may

be used. In removing residual solder, best results are obtained with the use of a low-temperature, blade style conductive desoldering tool that matches the footprint of the module package in conjunction with a de-soldering braid. The use of a de-soldering braid with a hot iron tip against the braid can be used.

Recommendation: When using a de-solder braid and hot iron tip, high temperatures and pressures on the PCB should be avoided as temperatures above 200°C combined with excessive pressure can damage land pads and the solder mask on the PCB.

Any unfilled vias in the PCB will become filled with solder with the improper use of desoldering tools. A de-solder braid is best used to remove residual solder around unfilled vias.

Recommendation: When using a de-solder braid and hot iron tip around unfilled vias in the PCB, the de-solder braid should remain stationary against the PCB surface and the hot tip moved across the back of the braid. As the solder reflows it will 'wick' into the braid.

The land pattern pads on the PCB must be clean of all solder or flux residue prior to module package remount.

Solder Paste Deposition

To properly re-attach a power converter module package onto the rework site of the PCB, solder paste must be reapplied to the land pattern of the PCB. A stencil-printing process is best used for precise application of the solder paste. However, since the PCB is populated with other surface mount components that may be in close proximity to the rework site, a miniature (spot) stencil that fits within the rework site will be required. This mini-stencil should be 127µm (5 mil) thick and made from stainless steel. It is important that stencil be carefully registered and aligned to the land pattern for accurate printing of the solder paste. Also, attention should be used when screening the solder paste to insure a uniform solder thickness through the stencil.

If the rework site is small with other surface mount components in too close proximity, a component stencil can be used to apply the solder paste directly to the power converter module package. Alignment fixturing can be used for aligning the module package to the stencil.

An alternative to stencil-printing solder paste on either the rework site of the PCB or the module package is to use dispensing equipment to apply dots of solder paste to the separate land pattern pads on the PCB.

Package Placement

Power converter module package placement requires high precision to place the module package accurately onto the land pattern of the PCB.

Recommendation: Module package placement should be performed with automated pick-and-place equipment that can provide high precision for accurate placement onto the land pattern of the PCB. The use of manual pick-and-place is not recommended, as the necessary placement precision will not be obtained.

A vision overlay system (VOS) with a minimum magnification of 70X should be used for accurate placement of the module package. Z-axis placement precision of the VOS should be 50 μ m (2 mil) or better. The module package should be placed accurately on the land pattern pads. A slight (25 μ m (1 mil)) misalignment can be tolerated, as the solder will provide self-aligning during reflow.

Package Reflow

As with removal of the power converter module package, it is necessary to provide sufficient heat only to the local rework site of the PCB to reflow the replacement module package. Preheating the bottom side of the PCB at the rework site is important for successful module package installation. Preheating will prevent the PCB from warping, twisting, or flexing during reflow. The PCB should be preheated to 100°C prior to application of localized heating to the rework site for solder reflow.

Reheat Equipment

To provide local heating only to the rework site of the PCB special reheat equipment should be used. These include a hot air or hot gas reflow system with a gas focusing nozzle that is sized to the module package dimensions and a preheater, such as a convection heater, hotplate, or IR heater. To place the module package onto the PCB equipment fitted with a vacuum nozzle should be used.

Recommendation: The reflow profile used to originally surface mount the packages and components on the PCB should be used to reflow the replacement module package.

When the solder reaches liquidus temperature, the solder will reflow and the module package will self-align.

Solder Joint Inspection

An inspection of the solder joint between the solder pads of the power converter module package and the PCB should be performed.

The best visual inspection tool for inspection of the module package solder joint on the PCB is a transmission X-ray, which can identify defects such as solder bridging, shorts, opens, and large voids (Note: small voids in large solder joints are not detrimental to the reliability of the solder joint). An additional visual inspection tool consists of side view inspection, such as 90° mirror projection, to determine component flatness and solder joint volume.

Recommendation: If possible, both transmission X-ray and a side view inspection process should be performed for visual solder joint inspection.

Figures

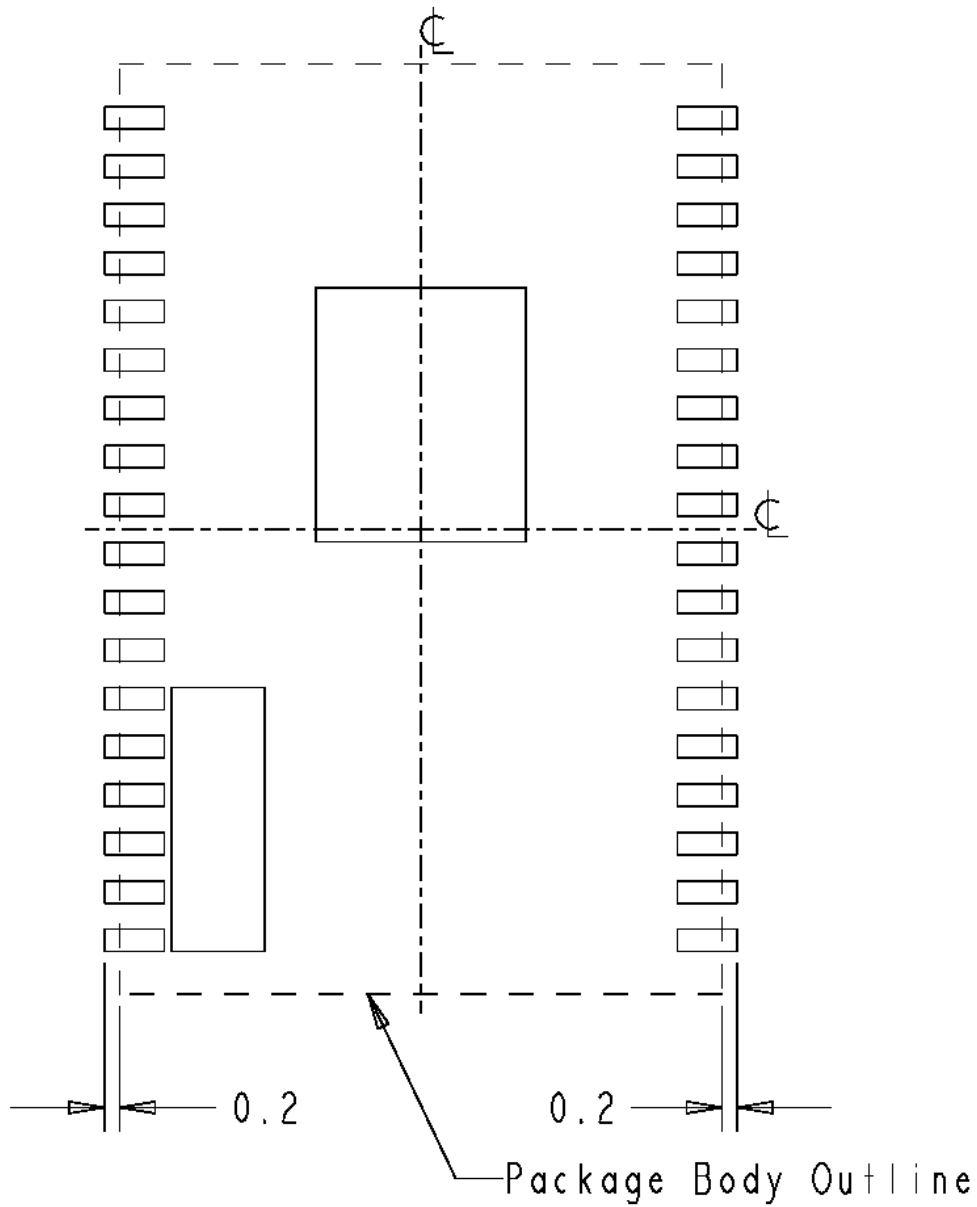


Figure 1: PCB pad lengths should be extended 0.20mm beyond package body outline

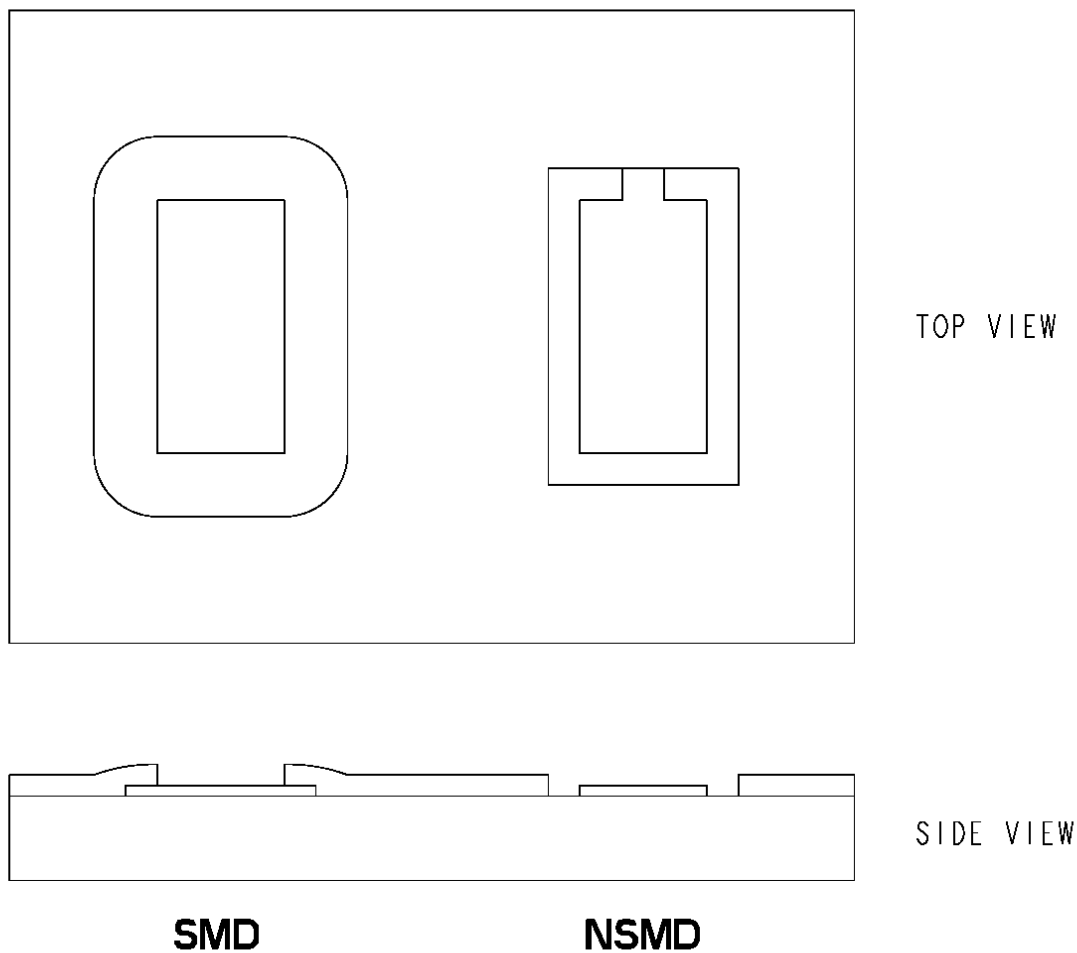


Figure 2: PCB Pad SMD and NSMD definitions

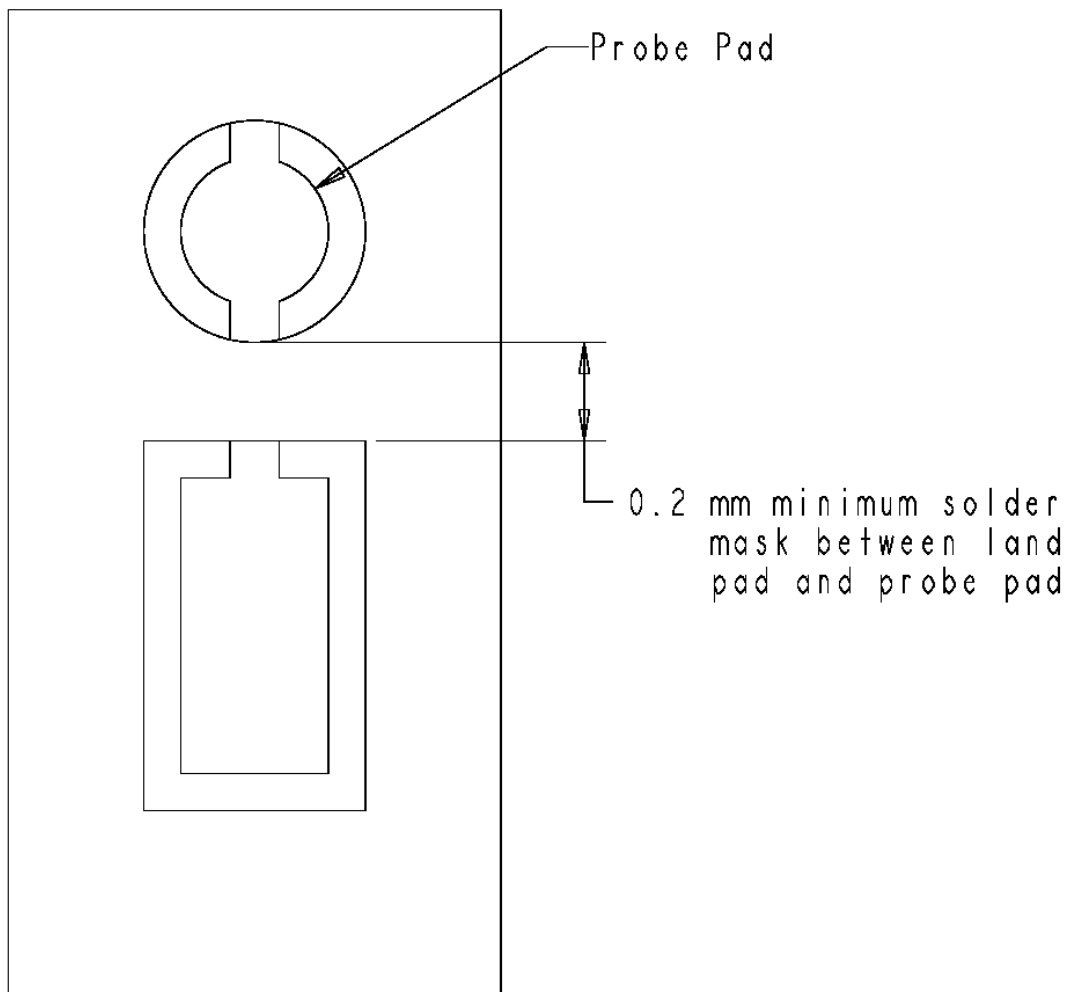


Figure 3: PCB pad design for probe pads

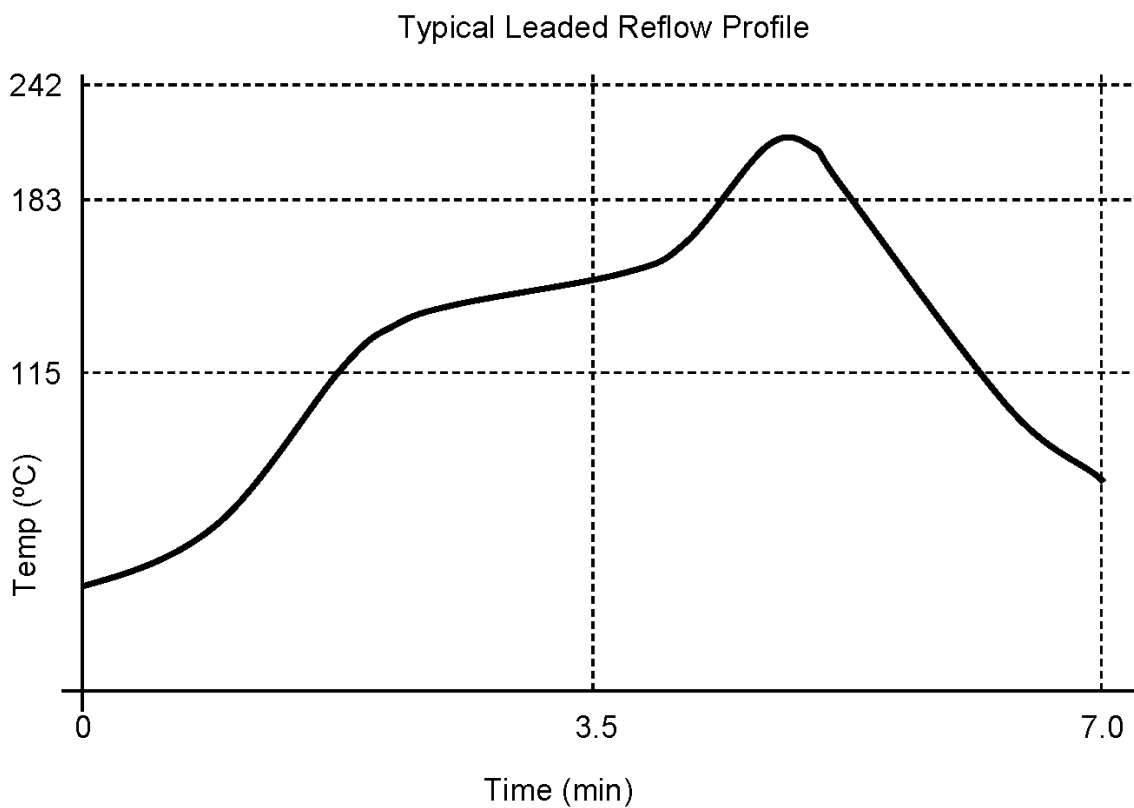


Figure 4: Typical Leaded Reflow Profile

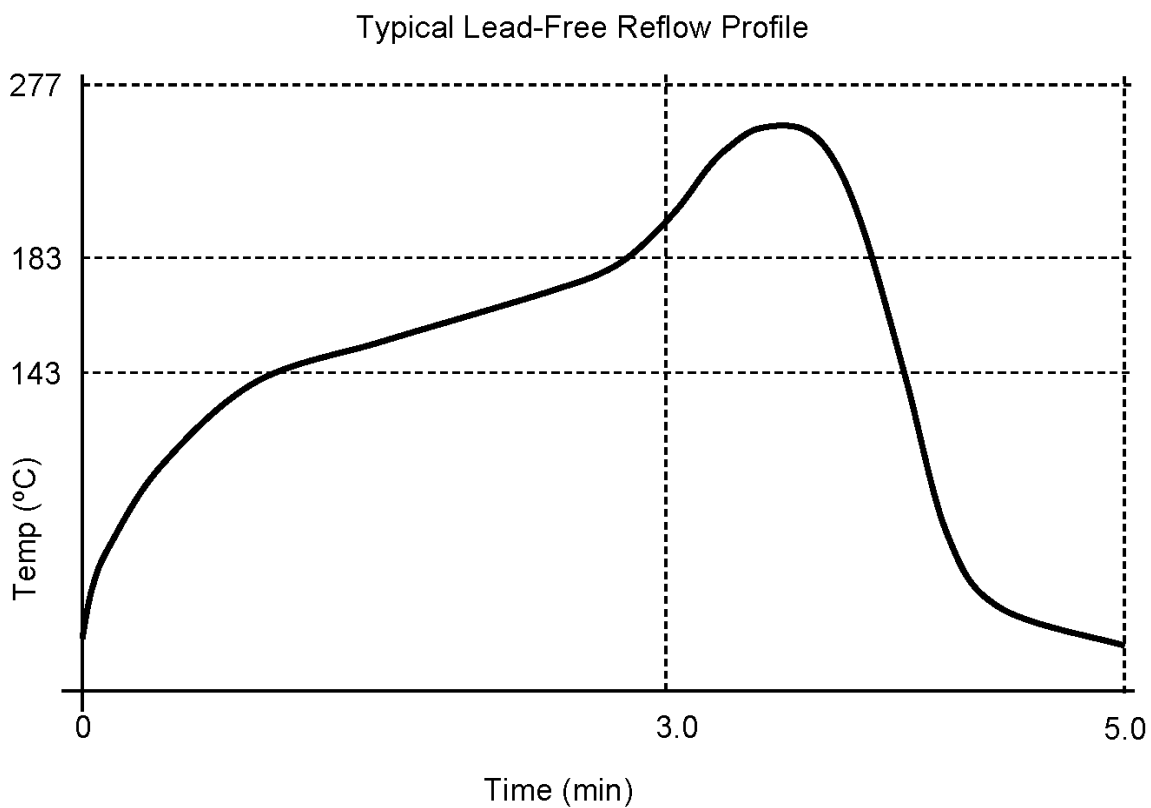


Figure 5: Typical Lead-Free Reflow Profile