

APPLICATION NOTE

SMT Assembly/Rework Guidelines for MCM-L Packages



REVISION HISTORY

Revision	Date	Description
A	August 2001	Initial Release
B	January 17, 2002	Revise: Sections 2.1, 2.2, 2.3, 3.1, 3.2, 3.3, 3.5, 4.0, 4.1, 4.2, 4.4, 5.0, 5.1, 5.3, 5.4,
C	July 18, 2002	Revise: Sections 2.2, 3.1, 4.3, 4.4, 5.0, 5.1; Figures 3-1, 4-2, 4-3; Table 4-2,
D	July 25, 2003	New Format Revise: Sections Introduction, Thermal-Mechanical Characteristics, PCB Design Guidelines, SMT Assembly Guidelines, Rework Guidelines; Figure 2. Add: Figures 4, 7, 9, 15, 16, 18, 19, 20 through 33; Tables 4, 5,
E	May 21, 2004	Revise: Table 1 Add: page 9 item 7, Figure 27; Revision History
F	January 12, 2006	Revise: New format. Figures 13-17, text revisions in all sections.
G	October 19, 2006	Revise: Table 6
H	March 8, 2010	Update Review
I	July 21, 2010	Revise: Text changes throughout; Figures
J	June 1, 2012	Revise: Removed PCB Design Guidelines section, reorganized
K	July 20, 2015	Revise: Revised to reflect current practices

Copyright © 2001–2004, 2006, 2010, 2012, 2015, Skyworks Solutions Inc. All Rights Reserved.

Information in this document is provided in connection with Skyworks Solutions, Inc. (“Skyworks”) products or services. These materials, including the information contained herein, are provided by Skyworks as a service to its customers and may be used for informational purposes only by the customer. Skyworks assumes no responsibility for errors or omissions in these materials or the information contained herein. Skyworks may change its documentation, products, services, specifications or product descriptions at any time, without notice. Skyworks makes no commitment to update the materials or information and shall have no responsibility whatsoever for conflicts, incompatibilities, or other difficulties arising from any future changes.

No license, whether express, implied, by estoppel or otherwise, is granted to any intellectual property rights by this document. Skyworks assumes no liability for any materials, products or information provided hereunder, including the sale, distribution, reproduction or use of Skyworks products, information or materials, except as may be provided in Skyworks Terms and Conditions of Sale.

THE MATERIALS, PRODUCTS AND INFORMATION ARE PROVIDED “AS IS” WITHOUT WARRANTY OF ANY KIND, WHETHER EXPRESS, IMPLIED, STATUTORY, OR OTHERWISE, INCLUDING FITNESS FOR A PARTICULAR PURPOSE OR USE, MERCHANTABILITY, PERFORMANCE, QUALITY OR NON—INFRINGEMENT OF ANY INTELLECTUAL PROPERTY RIGHT; ALL SUCH WARRANTIES ARE HEREBY EXPRESSLY DISCLAIMED. SKYWORKS DOES NOT WARRANT THE ACCURACY OR COMPLETENESS OF THE INFORMATION, TEXT, GRAPHICS OR OTHER ITEMS CONTAINED WITHIN THESE MATERIALS. SKYWORKS SHALL NOT BE LIABLE FOR ANY DAMAGES, INCLUDING BUT NOT LIMITED TO ANY SPECIAL, INDIRECT, INCIDENTAL, STATUTORY, OR CONSEQUENTIAL DAMAGES, INCLUDING WITHOUT LIMITATION, LOST REVENUES OR LOST PROFITS THAT MAY RESULT FROM THE USE OF THE MATERIALS OR INFORMATION, WHETHER OR NOT THE RECIPIENT OF MATERIALS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Skyworks products are not intended for use in medical, lifesaving or life—sustaining applications, or other equipment in which the failure of the Skyworks products could lead to personal injury, death, physical or environmental damage. Skyworks customers using or selling Skyworks products for use in such applications do so at their own risk and agree to fully indemnify Skyworks for any damages resulting from such improper use or sale.

Customers are responsible for their products and applications using Skyworks products, which may deviate from published specifications as a result of design defects, errors, or operation of products outside of published parameters or design specifications. Customers should include design and operating safeguards to minimize these and other risks. Skyworks assumes no liability for applications assistance, customer product design, or damage to any equipment resulting from the use of Skyworks products outside of stated published specifications or parameters.

Skyworks and the Skyworks symbol are trademarks or registered trademarks of Skyworks Solutions, Inc., in the United States and other countries. Third—party brands and names are for identification purposes only, and are the property of their respective owners. Additional information, including relevant terms and conditions, posted at www.skyworksin.com, are incorporated by reference.

Table of Contents

INTRODUCTION	1
Power Amplifier MCM-L Packages	2
Large Integrated MCM-L Packages	2
THERMAL-MECHANICAL CHARACTERISTICS	3
Thermal Considerations	3
Mechanical Considerations	4
Thermal Vias	5
Inner and Bottom Copper Pad	5
SURFACE MOUNT TECHNOLOGY (SMT) ASSEMBLY GUIDELINES	6
Stencil Design	6
Solder Paste	11
Moisture Sensitivity	11
SMT Reflow Profile	11
REWORK GUIDELINES	13
MCM-L Module Removal	13
PCB Pad Redress	13
Solder Application	14
New Package Placement	14
Post-placement Reflow	14

List of Figures

Figure 1. Active and Passive Components in a Skyworks MCM-L RF Module.....1

Figure 2. Top and Bottom Views of a Typical Skyworks 2 x 2.5 mm, 10-Pad MCM-L RF Power Amplifier Module.....2

Figure 3. Top and Bottom Views of Skyworks 8 x 9 mm 60-pad MCM-L integrated Module3

Figure 4. Heat Dissipation of a Mounted MCM-L RF Package through the PCB to the Ambient Air3

Figure 5. Optimal Bottom Copper Pad with Thermal Via Array Tented by Solder Mask.....5

Figure 6. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design 2 x 2.5 mm, 10-Pad MCM-L Integrated Module.....6

Figure 7. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design 8 x 9 mm, 60-Pad MCM-L Integrated Module 7

Figure 8. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design 8.15 x 5.60 mm, 42-Pad MCM-L Integrated Module with Conformal Shielding8

Figure 9. Typical Footprint and Solder Pad Configuration 10 x 10 mm, 68-Pad MCM-L Integrated Module9

Figure 10. Solder Coverage at Center Ground Pad – Mounted 10 x 10 mm, 68-Pad MCM-L Integrated Module.....9

Figure 11. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design 4 x 4 mm 10-Pad MCM-L Module.....10

Figure 12. Typical SMT Reflow Profile for Maximum Temperature = 260 °C12

List of Tables

Table 1. Typical SMT Reflow Profile for Maximum Temperature = 260 °C..... 12

INTRODUCTION

The Skyworks Multi-Chip Module-Laminate (MCM-L) package is a system solution that combines both active and passive functions in a compact, efficient module. MCM-L technology is used in most Skyworks RF modules and related applications.

The Power Amplifier (PA) module usually consumes a high level of current in an RF system resulting in high heat dissipation and severe thermal-mechanical stress conditions for the package. Special attention to Printed Circuit Board (PCB) design and the assembly of RF modules onto system boards is extremely critical to achieving optimal heat dissipation, stress condition, and operating reliability.

This Application Note provides thermal and mechanical background information, specific guidelines for PCB layout design, Surface Mount Technology (SMT) assembly processes, and other related instructions and solutions for effectively mounting Skyworks MCM-L packages onto system boards.

Figure 1 (left panel) shows a complete circuit system in an MCM-L package prior to overmold. Figure 1 (right panel) shows the cross-sectional construction of a typical MCM-L package based on a 6-layer laminate substrate. Compared to conventional solutions which use individual surface-mount semiconductors and a number of passives assembled directly onto the PCB, the Skyworks MCM-L package solution offers several advantages.

- Optimal RF matching circuitry through active and passive components
- Localized high density interconnection to achieve a small and efficient package
- Fewer Bills of Material (BOM) for system application
- Simpler system circuit design and shorter development cycle time

Skyworks MCM-L packages allow customers to meet the electrical, mechanical, and thermal performance requirements in a variety of RF applications. The package application for the RF module consists of a GaAs PA, GaAs/SOI switch, Si controller etc. The active GaAs die is mounted and wire bonded onto a copper die pad. The copper die pad connects through thermal vias to the inner metal layers of the laminate substrate and to the exposed metal center pad of the package (Figure 1). This unique metal structure ensures stable ground paths, low RF inductance, and effective heat dissipation for the MCM-L package. All are critical factors for achieving high performance (see Figure 2).

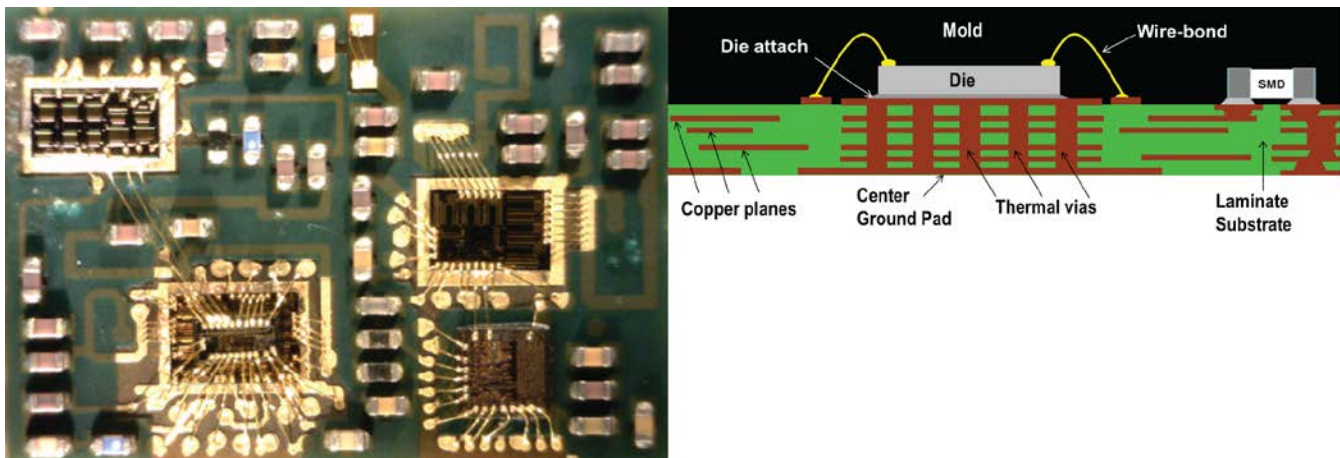


Figure 1. Active and Passive Components in a Skyworks MCM-L RF Module

Power Amplifier MCM-L Packages

A power amplifier MCM-L package features a laminate substrate with Land Grid Array (LGA) type perimeter pads and a large exposed center ground pad. Exhibits in Figure 2 show the top and bottom views of a typical Skyworks 2 x 2.5 mm RF PA module. Aside from a low inductance ground path, the LGA footprint with a center ground pad offers several thermal-mechanical advantages:

- Improved thermal performance with an efficient thermal path to the system PCB
- Enhanced board level interconnect reliability with a large ground pad and comparatively large perimeter pads
- Increased impact and drop reliability with a large area of solder connection and a low standoff.
- With appropriate system PCB designs, MCM-L packages deliver excellent thermal-mechanical reliability as well as electrical performance for the power amplifiers.

Skyworks families of MCM-L RF modules are available in a wide range of body sizes and functionalities.

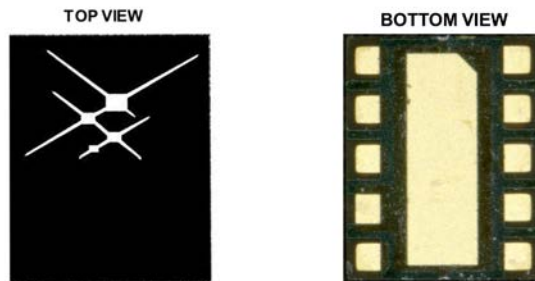


Figure 2. Top and Bottom Views of a Typical Skyworks 2 x 2.5 mm, 10-Pad MCM-L RF Power Amplifier Module

Large Integrated MCM-L Packages

In addition to PA packages, MCM-L packages also comprise integrated modules such as the 8 x 9 mm SkyOne® module (Figure 3). Integrated modules use the same substrate design rules but integrate more semiconductor devices and passive components into the modules for various electrical functions. Though integrated modules usually have larger body sizes, they require less total space on the system board than discrete solutions.

Integrated MCM-L packages feature high counts of devices and SMT components within the modules and larger body sizes. Instead of a single large center ground pad, integrated modules usually divide the ground pad into several sections by the solder masks or use a uniform grid pattern in the solder masks. LGA-type perimeter pads are also used in integrated MCM-L packages. Figure 3 shows integrated modules with the center ground pad divided into grid patterns by the bottom solder mask.

Integrated MCM-L packages possess the same advantages as MCM-L PA modules. With an appropriate system PCB design, an integrated MCM-L package provides excellent thermal-mechanical reliability as well as complex and extensive electrical functions.

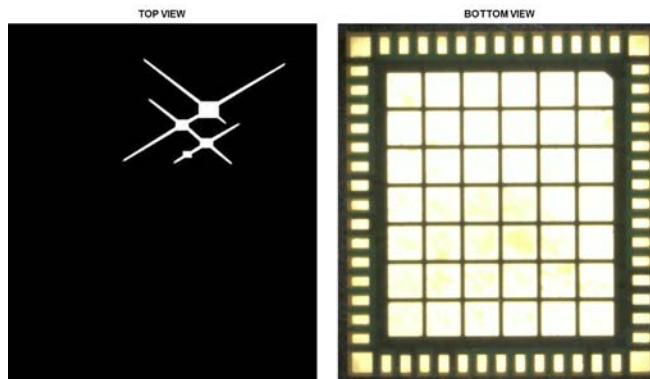


Figure 3. Top and Bottom Views of Skyworks 8 x 9 mm 60-pad MCM-L integrated Module

THERMAL-MECHANICAL CHARACTERISTICS

Thermal Considerations

In an MCM-L package, very little heat is dissipated to the air through the top surface of the mold compound. The major thermal path for heat dissipation from heat sources on the device (see Figure 4) is the path from the die to the package substrate, to the system PCB, through the PCB surfaces and into the air.

The efficiency of heat dissipation (measured by the device junction temperature) is largely dependent on the thermal resistance of the package and the PCB (including the thermal resistance of the PCB to the ambient air):

$$T_j = \Delta T_{ja} + T_a = P(R_{pkg} + R_{PCB}) + T_a$$

where,

- T_j is the junction temperature of the device
- ΔT_{ja} is the temperature difference between the junction and the ambient
- T_a is the ambient temperature
- P is the total power dissipation from the device
- R_{pkg} is the thermal resistance of the MCM package
- R_{PCB} is the thermal resistance of the PCB and PCB to ambient air

For a given maximum junction temperature, T_j max, the maximum power that can be dissipated through the package and the PCB to the ambient air is determined by:

$$P_{max} = \frac{T_{j,max} - T_a}{R_{pkg} + R_{PCB}}$$

This shows that in order to reduce the junction temperature or to dissipate more power from the device, the thermal resistances of the package, the PCB, and the PCB-to-ambient air must be minimized. Thermal resistance of the package is determined by the package size, materials, and structures. In MCM-L package designs, high thermal conductivity die-attach materials are used. Thermal vias and large metal pads are implemented in the substrate (Figure 4) to minimize the thermal resistance and enhance the efficiency of the heat dissipation from the device to the PCB.

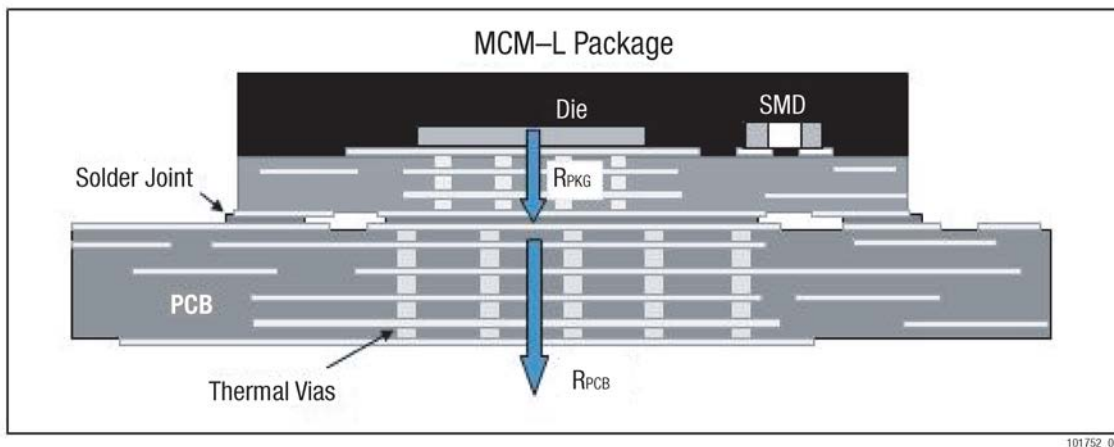


Figure 4. Heat Dissipation of a Mounted MCM-L RF Package through the PCB to the Ambient Air

When assembled onto a system PCB, the package center ground pad is soldered to the PCB ground pad for an effective thermal path. Almost all the heat generated from the package must eventually dissipate through the PCB to the air. Since the PCB-to-air thermal resistance is the major portion of the overall thermal resistance, appropriate design of the system PCB and proper assembly of the package onto the PCB are crucial to overall system thermal performance.

The following guidelines should be considered for system PCB designs and board level assembly:

- Optimize the board level assembly process and minimize the voids in the solder joints
- Maximize the common ground copper planes in the PCB at the top and the bottom surfaces. More copper content in the inner layers of the PCB can also help reduce the thermal resistance of the PCB.
- Ensure sufficient thermal vias connect the top and bottom ground copper planes in the PCB. These are most effective when as many thermal vias as possible are placed under the RF ground pad. Effectiveness of thermal vias diminishes when they are placed farther from the package ground pad.
- Minimize the interaction of the RF package with other heat sources on the PCB. Heat sources near the package can increase the PCB temperature and thus increase the ambient temperature. This is especially critical for the double-sided assembly where placement of heat sources should be avoided in the PCB area opposite the RF site. Conversely, passive components on the PCB can increase the efficiency of heat dissipation from the PCB to the air. Passive components placed near the RF package on either side of the PCB can improve the efficiency of heat dissipation from the RF package.
- Increase the contact areas between the PCB and the case, such as the phone case. Heat transfer is much more efficient via conduction than convection. More contact area increases the heat dissipation to the case and eventually to the air.
- In general, a larger PCB area is better for heat dissipation through the PCB to the ambient air. A large PCB should be used if allowed by the system design.

Mechanical Considerations

MCM-L package footprints are designed to satisfy the requirements for electrical, thermal, and mechanical performance, module and board level assembly, and board-level testing. For mechanical considerations, the footprint was designed to:

- Minimize package stresses during the assembly process and component level reliability testing
- Ensure an easy and robust assembly process for the board level attachment
- Provide sufficiently strong solder joints that will resist the environmental stresses such as impact and thermal cycle stresses in the applications

MCM-L packages with the LGA-type perimeter pads and large exposed ground pads exhibit excellent mechanical integrity and are tested against JEDEC standards for impact and thermal stress reliability. Results indicate that mechanical reliability is not an issue for packages properly attached to the PCB with the proper metal finish and sufficient solder volume.

Thermal Vias

To achieve best thermal and electrical performance of a mounted MCM-L package, an array of thermal vias placed on the area of the ground pad should be connected to the internal and bottom common ground copper planes of the PCB. The number of vias is based on via configuration and the thermal and electrical requirements of the particular module under consideration. In general, there is a direct correlation between the thermal via cross-sectional area and the heat dissipation rate. However, large and excessive thermal vias may introduce more voids in the solder joint and actually reduce overall heat dissipation performance.

Plated Through Hole (PTH) vias that has a drill diameter in the range of 0.25 mm to 0.33 mm with 1 oz. of plated copper in the via holes can be used. The thermal via array can be arranged evenly with a pitch of 0.4 mm to 1.0 mm, depending on the form factor of the package. For the exposed region of the ground pad, if the plating thickness is not sufficient to effectively plug the barrel of the via when plated, solder mask should be used to cap the vias with a minimum dimension equal to the via diameter plus 0.1 mm. This will prevent solder wicking through the thermal via during the soldering process, resulting in voiding.

Another way to plug thermal vias uses solder mask tenting on the bottom of the copper plane. Solder mask tenting must completely cover the vias. An example of this solder mask coverage is shown in Figure 5.

When PTH vias are not possible, array of Microvias can be designed on the PCB under the exposed ground pad. Microvias allow for via-in-pad technology, which reduces or eliminates traces routed between attachment pads. This allows more relaxed solder mask tolerances, reduces possibilities of exposed conductors, and larger assembly pads. However, dimensions and structures of the microvias vary significantly depending on the PCB manufacturing processes. The thermal microvia array must be determined by cross-area analysis and actual experiments performed on the specific microvia PCB.

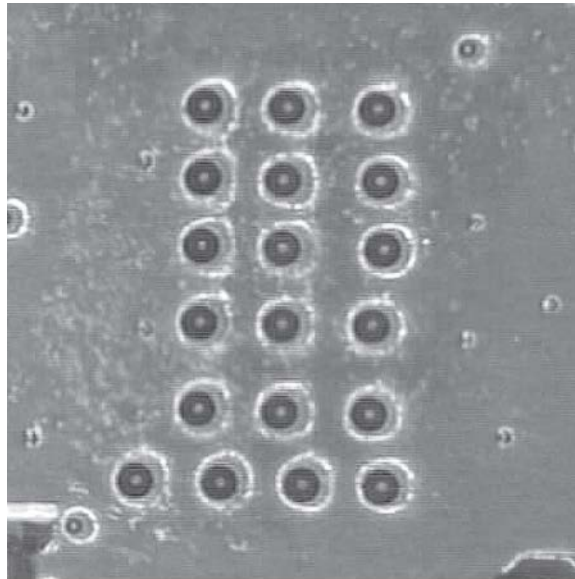


Figure 5. Optimal Bottom Copper Pad with Thermal Via Array Tented by Solder Mask

Inner and Bottom Copper Pad

Complete connection of the inner and bottom copper pads to the large common ground plane is recommended for the PCB heat sink to maximize heat dissipation from MCM-L package. This simple layout design is important and should be used effectively. The bottom copper pad should be free of interaction with other components on the PCB and completely covered by solder mask to tent the thermal via array (Figure 5).

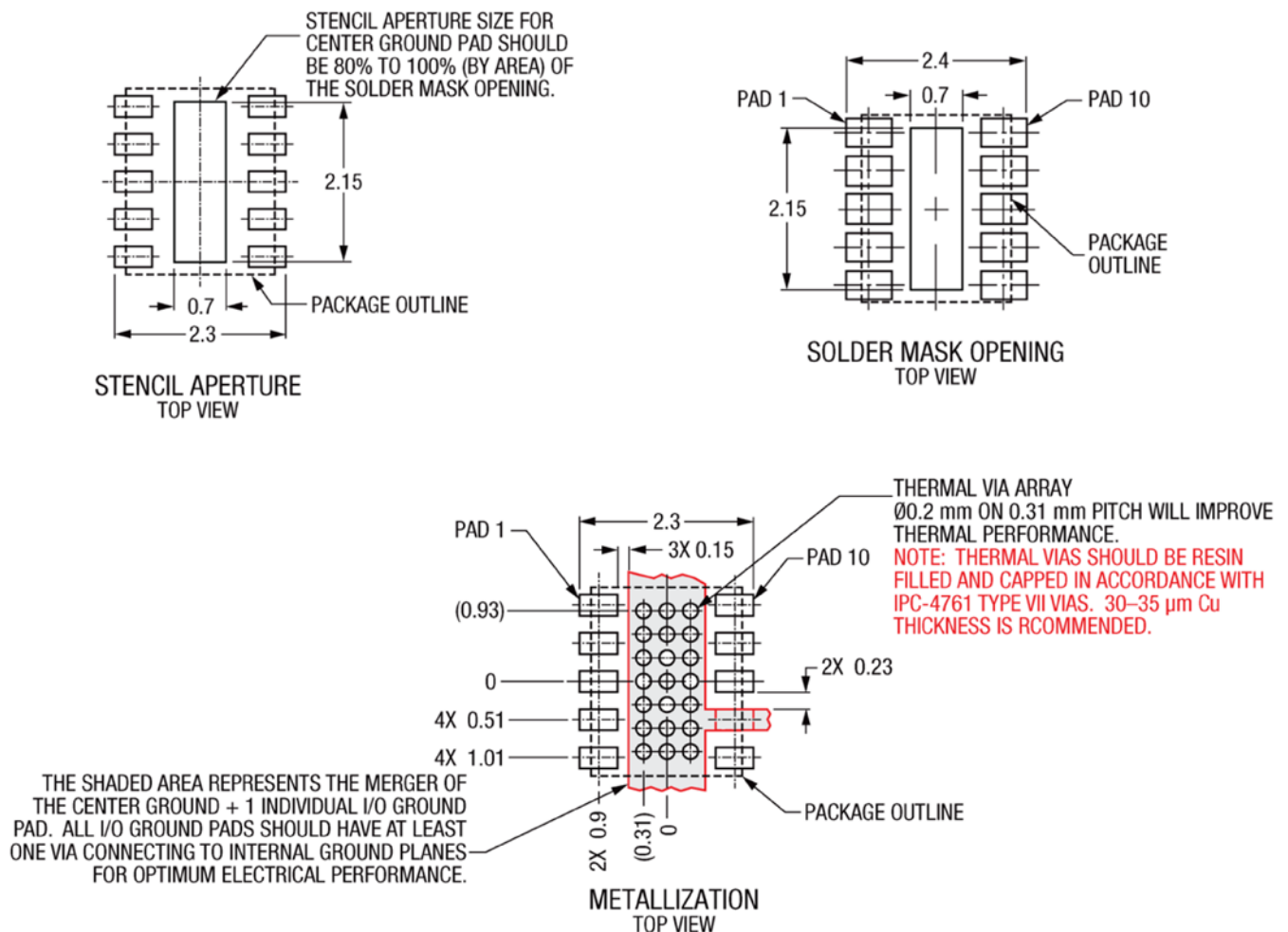
SURFACE MOUNT TECHNOLOGY (SMT) ASSEMBLY GUIDELINES

Control of solder paste volume and the solder reflow process is critical for SMT assembly of MCM-L modules onto the PCB. Stencil thickness and aperture openings should be adjusted according to the optimal solder volume. The following are general recommendations for SMT mounting MCM-L modules onto the PCB.

Stencil Design

MCM-L RF packages can be SMT reflowed onto PCB boards using a range of stencil thicknesses from 0.10 mm to 0.15 mm. For packages with terminal pad pitch equal to 0.5 mm or less, a 0.10 mm stencil is recommended. However, if a 0.15 mm stencil must be used due to the solder paste volume required by other PCB components, it is recommended to reduce the apertures (use the low end of the aperture opening sizes listed in Figures 6 and 7) to avoid excessive paste.

NOTE: An excessive reduction in aperture size will increase the area ratio, thus reducing the paste-transfer efficiency. (The area ratio of an aperture is defined as the ratio of the area of the aperture opening to the area of the aperture walls.)



**Figure 6. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design
2 x 2.5 mm, 10-Pad MCM-L Integrated Module**

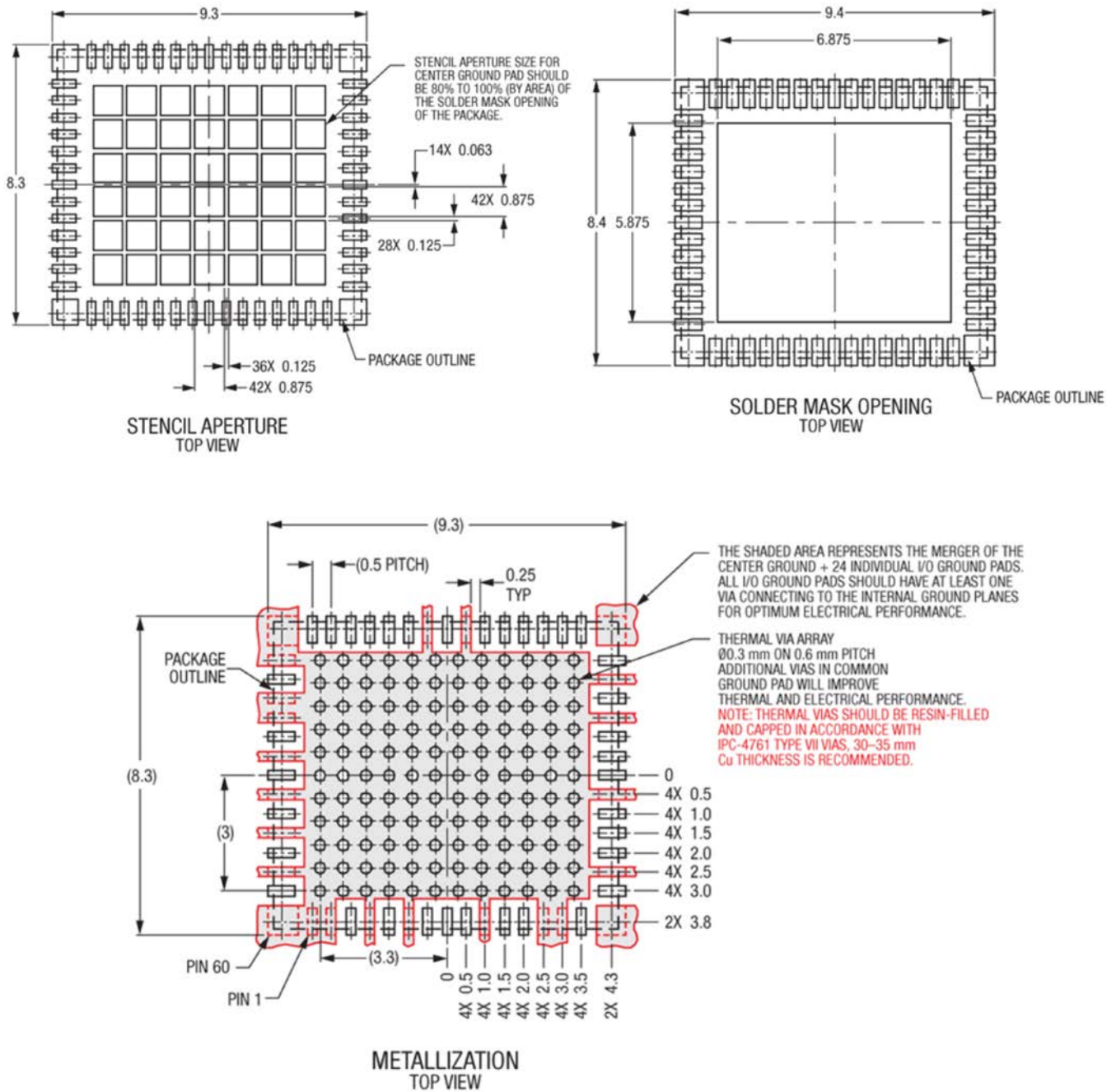


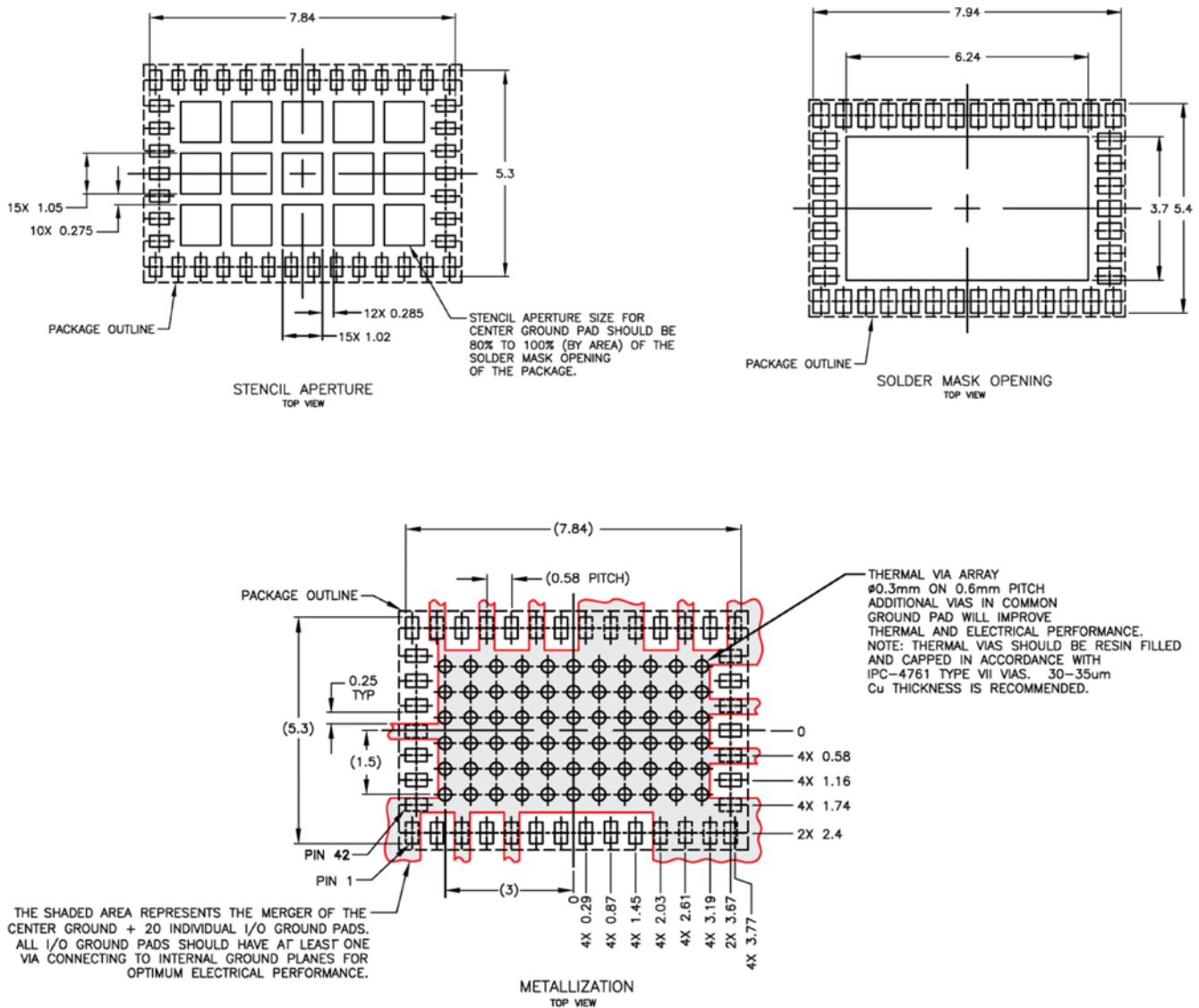
Figure 7. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design 8 x 9 mm, 60-Pad MCM-L Integrated Module

Generally, printing performance is affected when the area ratio is greater than 0.66, though good paste-transfer uniformity can be successfully obtained with more accurate calculations and careful stencil design.

For MCM-L modules larger than 4 x 4 mm and with terminal pad pitch greater than 0.5mm, a 0.15 mm thick stencil is recommended to maintain sufficient standoff, though a 0.125 mm and 0.10 mm thick stencil can be used. For packages with small pitch, such as the 0.5 mm pitch MCM-L module shown in Figure 7, thinner stencil (0.10 mm) should be used to reduce the solder paste volume on the terminal pads and prevent solder shorting.

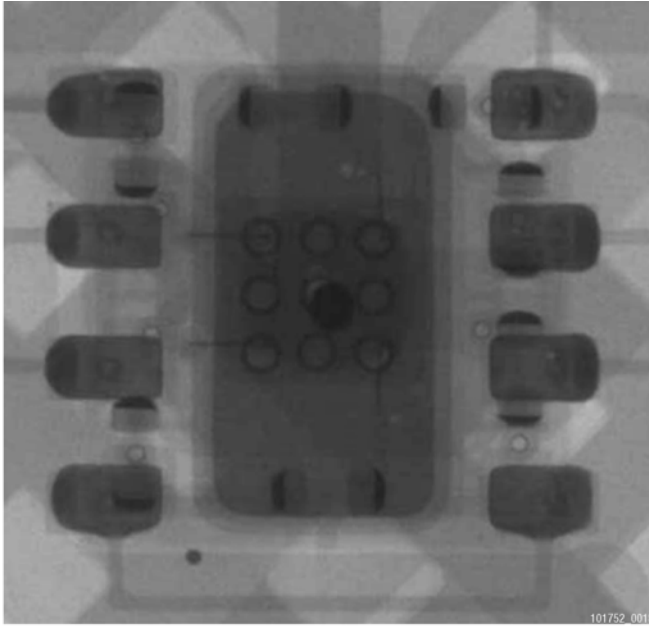
In all cases, the stencil aperture openings for the terminal pads should be equal to or smaller than the corresponding pad metallization on the PCB. The aperture opening reductions should be evaluated and determined according to the thickness of the stencil used in the assembly process.

In the case of conformal shielded modules, the PCB pad should not extend beyond the module edge for the risk of shorting the pins to the ground shield on the package side. It is important to follow the recommendation on the datasheet in such cases. Figure 8 shows the typical PCB design for mounting MCM-L modules with conformal shield. Since there is no extension of the terminal pads on the PCB for visual inspection of solder joint, care should be taken to follow the PCB recommendations provided in the datasheet.

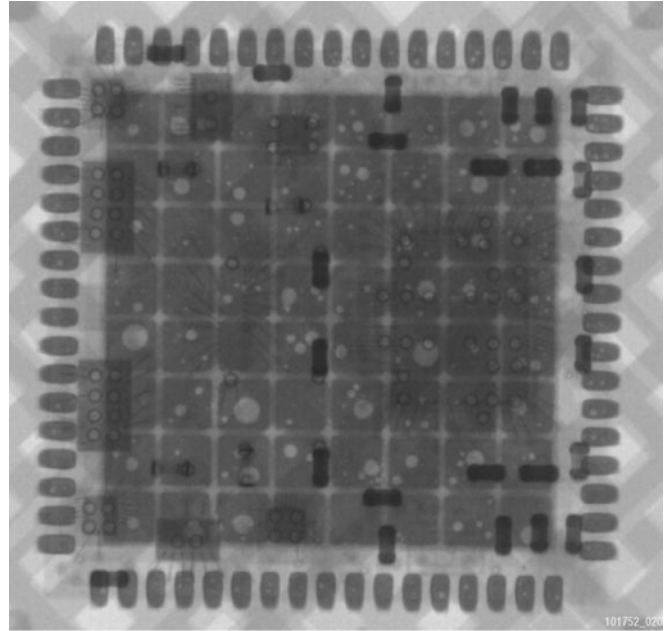


**Figure 8. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design
8.15 x 5.60 mm, 42-Pad MCM-L Integrated Module with Conformal Shielding**

In the case of packages without conformal shield, extending the peripheral pads beyond the edges of the package produces solder joint fillets that can be inspected visually as shown in Figures 9 and 10. Because the stencil apertures for the peripheral pads are extended accordingly, this PCB pad design further improves the solder paste release. To ensure efficient solder transfer, stencil apertures should be designed with tapered walls. Paste transfer efficiency as high as 80% can be achieved using 5 degree-tapered walls with the larger opening sides toward the PCB. A stainless steel, laser cut, and polished stencil is suggested for all assemblies. Circular features on the stencil apertures for the land pads should be avoided as it greatly reduces the paste transfer efficiency.



**Figure 9. Solder Coverage at Center Ground Pad –
Mounted 3 x 3 MM, 8-Pad MCM-L Power Amplifier Module**



**Figure 10. Solder Coverage at Center Ground Pad –
Mounted 10 x 10 mm, 68-Pad MCM-L Integrated Module**

Generally, as stencil thickness increases, aperture openings for the large center ground pads should be reduced. This is accomplished by optimizing the aperture opening such that the ground pad makes contact with the ground pad on the board without affecting peripheral I/Os. The first method to reduce the solder volume uses a single stencil aperture with a reduced aperture opening. Depending on the stencil thickness (and the thermal via configurations), the stencil opening can be reduced by 20% to 50% of the total area of the exposed PCB ground pad. (After reduction, the stencil opening is 50% to 80% of the exposed area of the PCB ground pad.) Figure 11 shows the calculated aperture dimensions versus the dimensions of the exposed area of the center ground pad for the 4 x 4 mm package when a thick stencil is used. This method works well with smaller aperture sizes but may produce insufficient solder volume on large openings.

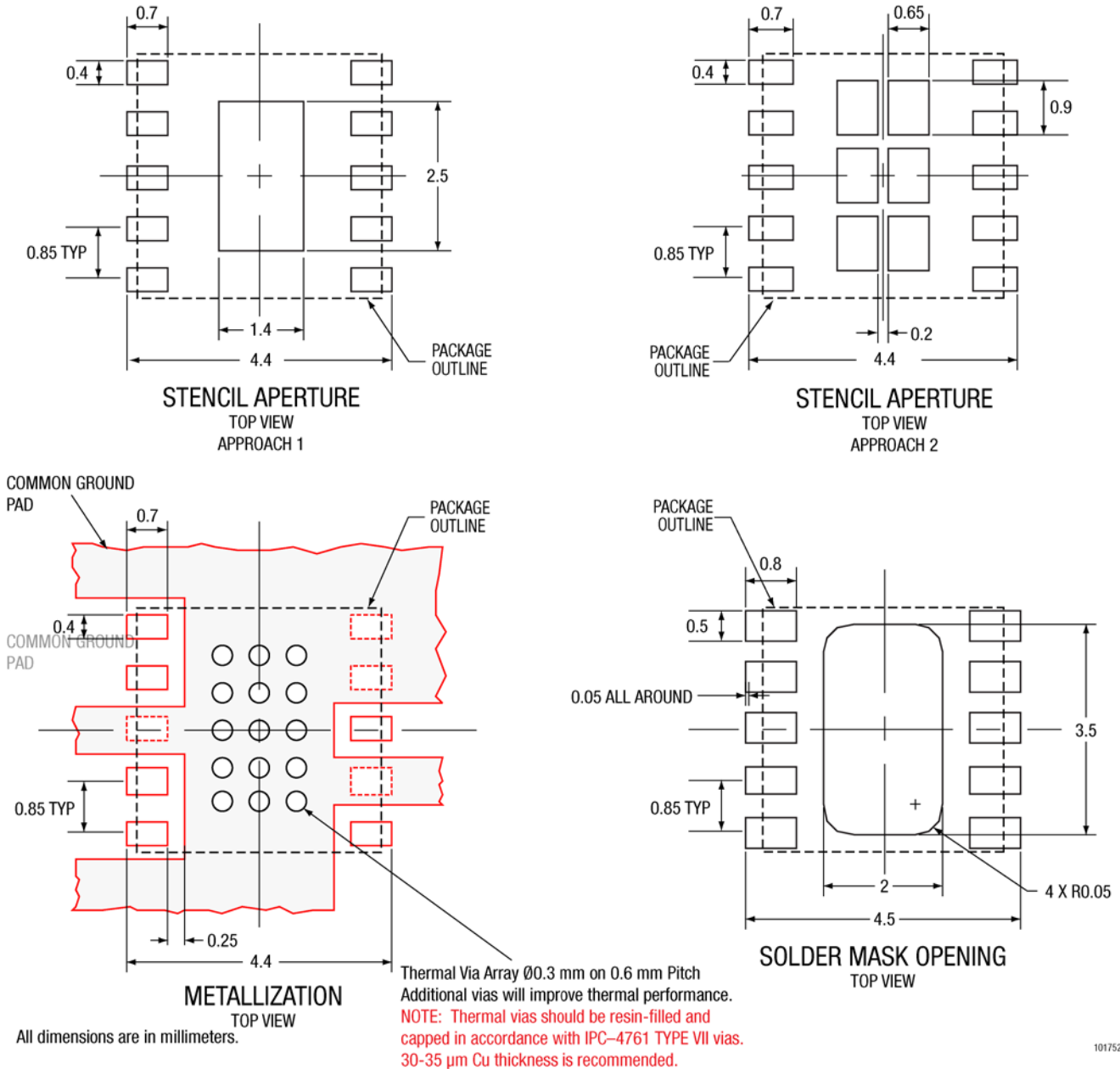


Figure 11. Typical PCB Metallization, Solder Mask Opening, Suggested Paste Stencil Design – 4 x 4 mm 10-Pad MCM-L Module

An alternative method uses webs in the stencil aperture to split a large aperture into smaller windows. The total aperture area (the sum of all the window areas) should equal the area required for the total aperture, which is 50% to 80% of the exposed metal area of the PCB ground pad. This method is shown in Figure 7 as the second approach of the stencil design.

For MCM-L modules that have a webbed solder mask pattern for the center ground pad, the stencil aperture should be designed individually for each exposed area. The size of each aperture should equal 70% to 100% of the corresponding exposed metal area on the module depending on the stencil thickness, using the low end of the aperture opening sizes when a thicker stencil is used.

There are cases of the solder mask for the MCM-L module center ground pad is designed with a uniform grid pattern. The stencil design should replicate the grid pattern with a stencil aperture size of 70% to 100% of the corresponding window size in the grid in the module solder mask. It should be pointed out that when the solder mask for the module center ground pad is designed with a grid pattern, the solder mask for the PCB center ground pad can use a single opening without the grid pattern. However, for the stencil, the grid pattern should be used whenever the module solder mask has a grid pattern.

The recommendations of PCB metallization patterns, solder mask-opening designs, and stencil aperture geometries and dimensions are all provided in MCM-L modules data sheets. In the PCB and stencil design process, the information in the data sheets should be reviewed carefully and applied properly.

Solder Paste

The most critical issue related to assembly materials and process parameters is the proper fluxing of the attachment surfaces and the creation of proper metallurgical bonds during the SMT reflow process.

Solder paste characteristics that are important from a process perspective include the solder powder particle size, metallurgy, slump, temperature and humidity sensitivity, solder content, type of flux residue, viscosity, and the propensity for solder balling. No-clean eutectic solder pastes with high metal content are used widely in mobile terminal PCB assembly. The choice of the particle size used in the solder powder for area array is dependent on the device pitch. As the device pitch decreases, the aperture size will shrink and a smaller particle size solder paste may be required. In most MCM-L modules, Type III solder paste will result in proper paste printing and solder joints. For fine pitch modules, such as the 0.5mm pitch MCM-L modules, Type IV paste should be considered.

Moisture Sensitivity

Skyworks' manufacturing procedures for MCM-L packages minimize the effects of moisture absorption and associated problems. Packages that are sensitive to moisture absorption are baked and vacuum packed for shipping with a desiccant and a humidity indicator card. It is highly recommended that procedures for handling moisture sensitive devices shall follow the Joint Industry Standard Institute for Printed Circuits IPC/JEDEC J-STD-020. Parts maintained within the specified humidity and temperature limits must be used in reflow manufacture prior to the MSL set time limit marked on the dry bag. Parts that have been exposed to the open air longer than the allowed time as indicated by the MSL on the dry bag, must as a general rule, be removed from the reel and baked for 8 hours at 125 °C before reflow.

Another important aspect is the heating profile used for the reflow soldering process. The majority of Skyworks products are reliable up to a maximum of 260°C reflow temperature. The MSL and peak temperature rating for which the product is qualified is printed on the moisture barrier bag. Always observe the moisture caution label on the dry pack prior to setting up storage conditions and developing solder reflow profiles.

SMT Reflow Profile

Due to the nature of the solder pad configuration, a wave solder process shall not be used during soldering assembly of Skyworks MCM-L packages. Common IR or convection reflow SMT processes shall be used for the assembly. Standard SMT reflow profile such as one shown in Figure 12, can be used to surface mount the MCM-L packages onto the PCB. A range of recommended parameters for the SMT reflow profiles is listed in Table 1. Typically, the paste manufacturer's recommendations should also be considered to determine the proper reflow profile.

NOTE: Maximum peak temperature for each part can be found in the device technical specification data sheet.

In all cases, a temperature gradient of 3 °C/sec, or less, should be maintained to prevent warpage of the package and to ensure that all joints reflow properly. Additional soak time and slower preheating time may be required to improve the out-gassing of solder paste. In addition, the reflow profile depends on the PCB density and the type of solder paste used. The user should make final adjustments of the reflow profile according to their applications conditions. Standard no-clean solder paste is generally recommended. If another type of flux is used, removal of flux residual may be necessary. If solder balling occurs during reflow, the paste aperture may be decreased to reduce

the amount of paste deposited to the PCB. In addition, nitrogen may be used to help counteract the tendency of the paste to form solder balls.

The peak reflow temperature should not exceed the MSL rated temperature on the data sheet for the specific part.

During the assembly process, the PCB must be supported properly to ensure board flatness. Typically, supports under the board are provided at each workstation but are not always available in the conveyor system. When thin or large boards are populated with components, the weight of the board itself plus the weight of the components may deform the PCB in reflow and cause board sagging. This effect may shift the component from its designated location on the board after placement. Consequently, there is a large variation in solder joint height, which increases the potential for solder joint defects such as bridging and opens. In general, carriers need to be designed and used for large or thin boards in order to ensure proper board flatness.

Although a certain amount of voids are acceptable in the solder joint, solder coverage on the center ground pad should be greater than 80% to ensure electrical, thermal and mechanical performance. In general, 20% voiding distributed across the entire center pad as small voids should NOT be an issue. However, a single (or multiple) large void(s) underneath the thermal vias can be detrimental to the performance of the module. Acceptable solder coverage can be seen in the x-ray images in Figure 9 and Figure 10.

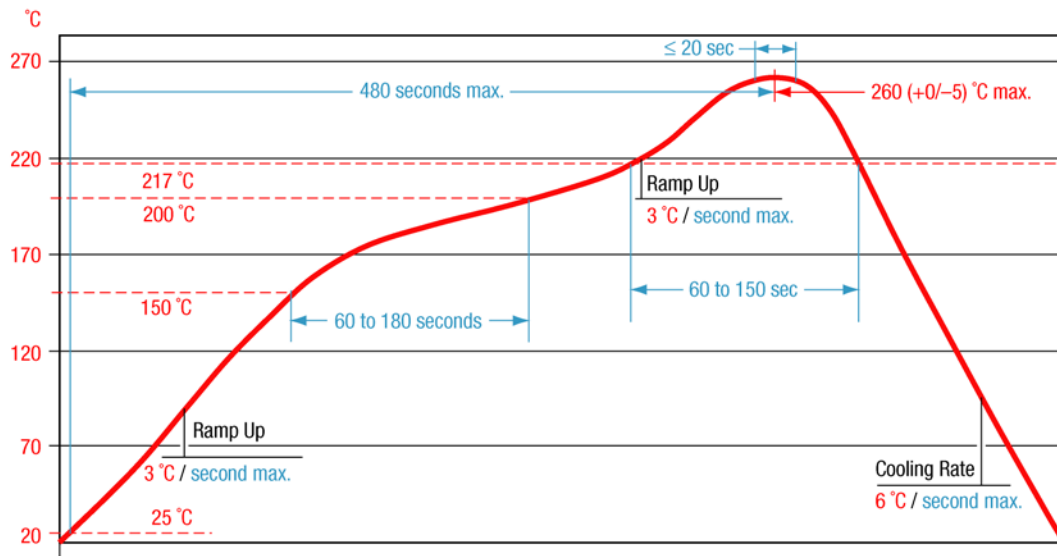


Figure 12. Typical SMT Reflow Profile for Maximum Temperature = 260 °C

Table 1. Typical SMT Reflow Profile for Maximum Temperature = 260 °C

Parameter	Convection or IR/Convection
Average ramp-up rate	3 °C/second max.
Preheat Temperature 175 (±25) °C	60 to 180 seconds.
Temperature maintained above 217 °C	60 to 150 seconds
Time between 255 °C and 260 °C (peak temperature)	10 to 20 seconds
Peak temperature range	260 (+0/-5) °C or 255 (+5/-0) °C
Ramp-down rate	6 °C/second max.
Time from 25 °C to peak temperature	480 seconds max.

REWORK GUIDELINES

Due to the nature of MCM-L land pads, retouch can be difficult. By extending the pads 0.1 mm to 0.3 mm beyond the package edge, some localized retouching can be performed but it is generally recommended to remove the package from the PCB after any opens or shorts are determined. Furthermore, rework of MCM-L modules can be a challenge due to the small package size. In most applications, MCM-L modules will be mounted on smaller, thinner, and denser PCBs that introduce further challenges due to handling and heating issues. Since reflow of adjacent parts is not desirable during rework, the proximity of other components may further complicate this process. A checklist and general procedures are provided to achieve an optimal rework/removal and replacement.

1. Prior to rework, X-ray, serialize and data log the failed part. This helps to confirm the part status prior to rework and for part failure analysis (FA) after removal.
2. Minimize moisture content by baking the system board/PCB assemblies at 125 °C for a minimum 8 hours before starting the removal or replacement process.
3. All Skyworks parts adhere to IPC/JEDECJ-STD-020 and IPC/JEDECJ-STD-033. MSL guidelines should be strictly followed to achieve optimum results. The MSL rating of the part can be obtained from the caution label on the dry pack.
4. Peak temperature of a part should not exceed its recommended maximum temperature. Temperature information is stated on the caution label on the dry pack. (Temperature is measured on the top of the package using a thermocouple and temperature recorder).
5. Rework tools, heating profiles, and operating parameters are customer or application specific, which should be characterized for optimum results.
6. Control the heating rate within the range of 3 °C to 5 °C/sec; the heating rate should not exceed 5 °C/sec for best rework results. Total heating time should be a minimum of 120 seconds. This can be achieved by using a rework station and monitoring the profile using thermocouples.

NOTE: When reworking adjacent packages or other parts or components in close proximity to the MCM-L package (on the same side or the opposite side of the system board), care must be taken to avoid overheating the MCM packages from excessive heat applied during a relatively less controllable manual rework process. This becomes particularly critical if a small MCM-L package, such as a 3 mm x 3 mm or a 4 mm x 4 mm, is very close to a bigger device undergoing rework. In such cases, appropriate shielding should be used to prevent any damage to the package or reflow of its solder joints.

MCM-L Module Removal

To reflow the solder joints that attach the package to the PCB, ideally, the reflow profile for removing the package should be the same as that used for part attachment (Figure 12). However, the time above liquidus can be reduced as long as the reflow is complete.

In the removal process, a convective heater is recommended to heat the bottom side of the board and hot gas, or air, to heat the top side of the component. Special nozzles should be used to direct the heat in the component area to minimize heating of adjacent components. Excessive airflow should also be avoided, as this may cause the MCM-L modules to skew or overheat. When the solder joint temperature exceeds the melting point the component can be readily removed from the PCB assembly.

When the joints have reflow, the vacuum liftoff should automatically engage to pickup the part, or the part should be picked up with tweezers. Do not twist or tilt the component during the removal process. If resistance is encountered during pickup, be careful not to force the package off the board, which may damage the package as well as the system board. In such instances, the heating profile setup needs to be verified.

NOTE: If heating conditions are not properly controlled during manual hot removal from the PCB assembly, the integrity of the MCM-L package can be damaged from overheating. MCM-L modules removed in this manner should not be reused and may not be suited for further functional examination (FA). If failure analysis of the mounted device is required, it is strongly recommended that the low heat method, stated above, be employed. Alternatively, for optimal preservation of the failure mode, mechanical methods such as cutting or grinding can be used to remove the MCM-L module. This will preserve component package integrity and help identify actual failure mechanisms.

PCB Pad Redress

After the component has been removed, the site and solder pads need to be properly cleaned. It is best to use the combination of a blade-style conductive tool and a de-soldering braid. The blade width should be matched to the maximum width of the footprint, and the blade temperature should be low enough to not cause damage to the system board. Once the residual solder has been removed, the land pads

should be cleaned with a solvent. The solvent is usually specific for the type of paste used in the original assembly and the paste manufacturer's recommendations should be followed.

Solder Application

In general, the rework process should follow the same SMT guidelines as those recommended in the previous sections. However, in situations where the stencil printing is not practical, rework of MCM-L modules may require extra steps.

1. PCB Pre-coating

A uniform and precise deposit can be achieved only if a miniature stencil specific to the component is used. When a stencil is used, the stencil aperture should be aligned with the pads under a microscope with 50x to 100x magnification.

The stencil should then be lowered onto the PCB and the paste deposited with a small metal squeegee blade. Blade width should be the same as the package width to ensure a single pass paste deposit to avoid any overprinting. The stencil thickness and aperture size and shape should be the same as that used for the original assembly. Due to the small standoff of MCM-L PA modules, no-clean flux should be used.

Due to the densities the components on the PCB and the keep-out areas required for the stencil, use of the stencil may not always be possible and solder wire may be used. A sharp-tipped soldering iron should be used with a solder wire to replenish approximately equal amounts of solder on each of the relatively small, finely-spaced MCM peripheral pads. Use of a microscope may be necessary. Only one pad at a time should be replenished with solder. Be careful to avoid bridging of adjacent pads. The PCB should NOT be exposed to prolonged heating.

2. New Package Pre-coating

When stencil printing is not possible and solder wire is used, solder coating on the packages should also be performed. All package pads should be solder-coated using a soldering iron and small diameter solder wire. Place just enough solder so that the solder finish is raised above the solder mask next to the pad. Make sure that the solder volume is consistent across all pads.

When a stencil printing is used to deposit the solder paste on the PCB, solder coating on the package is not required.

NOTE: Excessive solder at the center pad(s) will cause the part to float up during replacement. Carefully add an appropriate amount of solder at the center pad. Operator experience and skill is critical.

New Package Placement

If stencil-printing paste is not used, a thin coating of liquid flux should be applied over the pre-coated pads on the PCB before placing the package.

A split-beam optical system should be used to align the package to the PCB. This method will form an image of the land pad overlaid on the mating footprint and aid in proper alignment. The placement machine should have the capability of allowing fine adjustments in the X, Y, and rotational axes.

When a split-beam optical system is not available, carefully place the package by hand. Alignment, in this case, should be done under a microscope of 50x to 100x magnification.

Post-placement Reflow

The reflow profile developed during original attachment or removal should be used to attach the new component. Since all reflow profile parameters have already been optimized, using the same profile will eliminate the need for thermocouple feedback and reduce operator dependencies. When the original reflow profile is not available, follow these steps for the attachment process.

1. Preheat the PCB so that the top of the board measures 100 °C to 125 °C
2. Using a top side hot-air nozzle, run a reflow profile with the similar heating rate as that shown in Figure 12 until the solder joints reach visible melting
3. Allow the solder to remain melted for 30-60 seconds
4. Turn off the heat and allow the package to cool to room temperature.

General Information:
Skyworks Solutions, Inc.
20 Sylvan Road
Woburn, MA 01801

