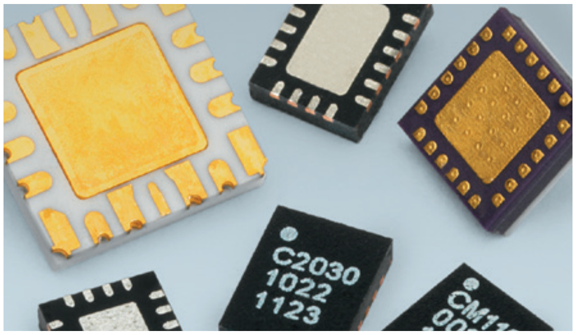
Custom MMIC

Recommended Land Patterns for QFN Devices

Introduction

This Application Note documents the recommended PCB Land Pattern, Solder Mask, and Solder Stencil designs for the QFN SMT devices.



The information provided in this application is for a guideline. Users are advised to optimize their own PCB level design in order to obtain proper solder reflow and device attach.

Custom MMIC recommends using a Non Solder Mask Defined (NSMD) land pattern due to tighter tolerance on copper etching than a SMD land pattern. NSMD also provides a larger copper pad area and allows the solder to adhere to the edges of the pad providing improved solder joint attachment and reliability.

It should be emphasized that these guidelines are general in nature and should be considered a starting point only. The end user must apply their experience and development efforts to optimize their design and manufacturing processes to meet their specific application.

General PCB Land Pattern Guidelines

The exposed PCB metal I/O pads should be designed to be 0.2 mm to 0.5 mm longer than the device I/O pads. Inward corners may be rounded to match the device I/O pad. The I/O land width should match the device I/O pad width, 1:1 ratio.

The PCB thermal land pad should be designed with multiple vias so that when the thermal pad is properly soldered, it will connect the exposed pad of the device to a metal plane in order to improve the thermal dissipation of the device during operation. The thermal land pattern should match a 1:1 ratio with the exposed thermal pad of the device.

Please review the drawings on pages 4 - 6 for recommended PCB land pattern design for various typical packages.

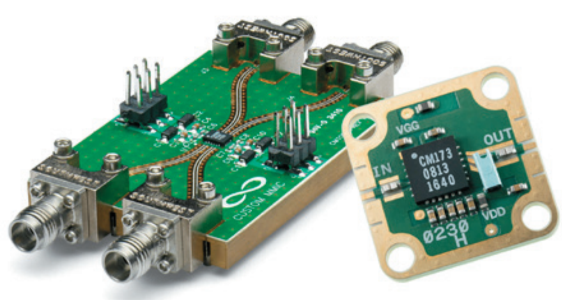
General Solder Mask Design Guidelines

Two types of solder mask land patterns are used for SMT devices; Solder Mask Defined (SMD), where the solder mask openings are smaller than the metal pads, and NSMD, where the openings are larger than the metal pads. As introduced earlier, control of the copper etching process as compared to the solder masking process makes NSMD the preferable design. The SMD pad definition can also introduce stress concentrations near the solder mask overlap region that can result in solder joints cracking under extreme fatigue conditions. NSMD design also improves the reliability of solder joints, as solder is allowed to "wrap around" the sides of the metal pads on the board.

Custom MMIC recommends a non-solder mask defined design where the solder mask openings are slightly larger than the land pattern metal pads. The solder mask opening should be about 120 um to 150 um larger than the metal pad, providing a 60 um to 75 um clearance between the pad and solder mask. Rounded pads should have matching rounded solder mask openings.

For package sizes with the thermal pad size near the maximum size for the package, the gap between the thermal pad and the exposed I/O pads may not be wide enough to minimize solder bridging concerns. In this case, we recommend a SMD design for the thermal pad solder mask. The solder mask should be 100 um smaller than the thermal pad on both sides, which increases the solder mask web between the thermal pad and the exposed I/O pads.

Solder masking is also recommended for the thermal vias in order to prevent solder from wicking away from the thermal pad into the vias and creating potential solder voids under the thermal land pad. The solder mask diameter should be about 80 um to 100 um larger than the via diameter. The vias can be plugged or tented with solder mask either from the top or bottom surface. Tenting from the top surface is preferred since it will result in smaller voids under the die paddle.



General Solder Stencil Design Guidelines

Custom MMIC recommends using a stainless steel stencil with electropolished trapezoidal walls to minimize surface friction in order for a clean paste release (**Figure 1**).

The stencil aperture opening should be a 1:1 ratio to the metal pad. In the illustration above, dimension "B" is equal to the metal pad width, dimension "A" is equal to 90% of dimension "B", and dimension "C" is typically 100 um to 150 um. Actual thickness will depend on other components on the board.

If the thermal pad of the device is larger than 2 mm X 2 mm, the stencil aperture for the thermal pad area may need to be designed as an array of openings instead of one large opening. In this case, the total open area of the stencil should equal 50% to 60% of the total area of the exposed metal thermal pad of the device. This will allow proper solder flow with minimal voiding, and eliminate the potential for the device to "float" on excessive solder. Custom MMIC recommends a "window pane" design be utilized for any device with a device thermal pad larger than 2 mm X 2 mm.

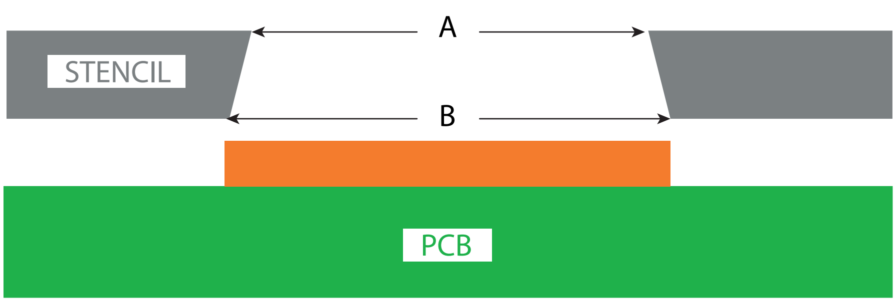
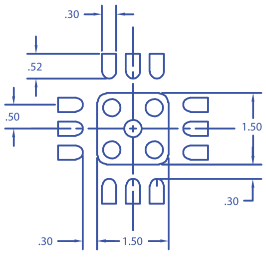
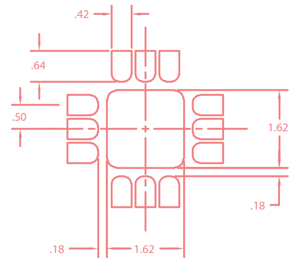


FIGURE 1.
Example of solder stencil opening dimensions

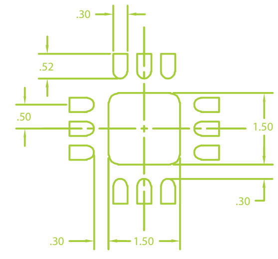
Ceramic QFN 3X3 mm x 12 leads



METALIZATION

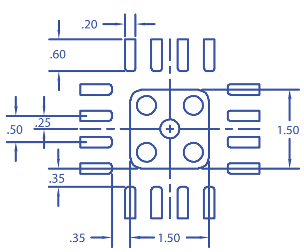


SOLDER MASK

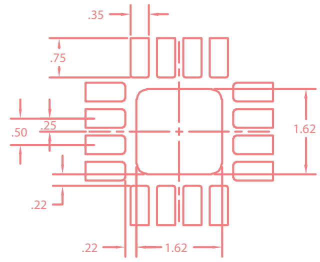


SOLDER STENCIL

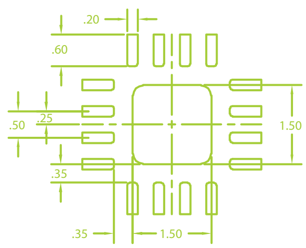
Plastic QFN 3x 3 mm x 16 leads



METALIZATION

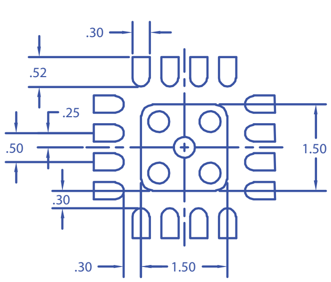


SOLDER MASK

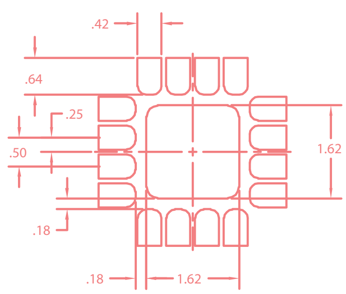


SOLDER STENCIL

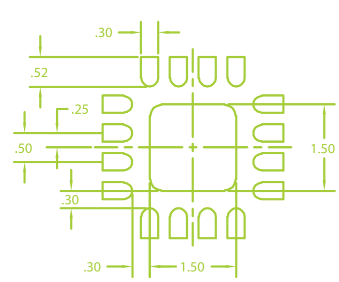
Ceramic QFN 3 x 3 mm x 16 leads



METALIZATION

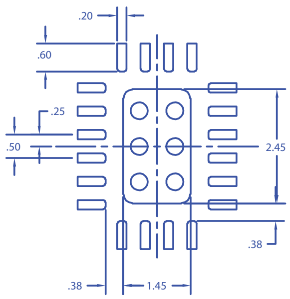


SOLDER MASK

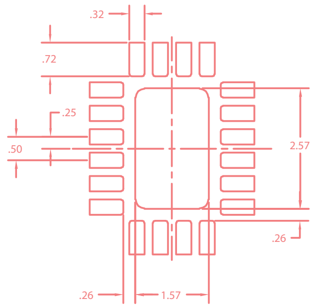


SOLDER STENCIL

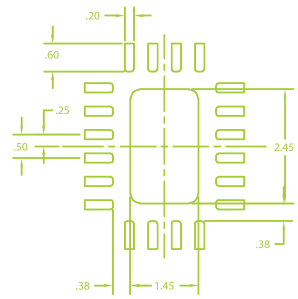
Plastic QFN 3 x 4 mm x 20 leads



METALIZATION

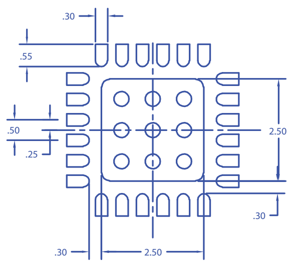


SOLDER MASK

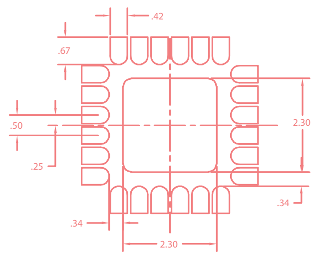


SOLDER STENCIL

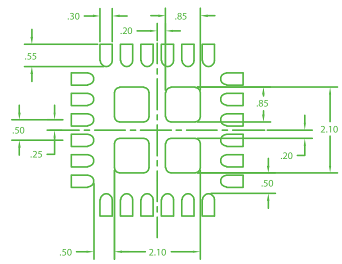
Ceramic QFN 4 x 4 mm x 24 leads



METALIZATION

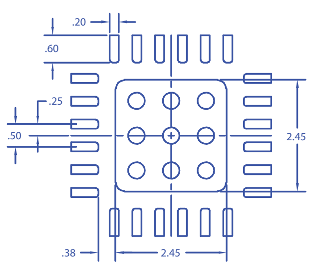


SOLDER MASK

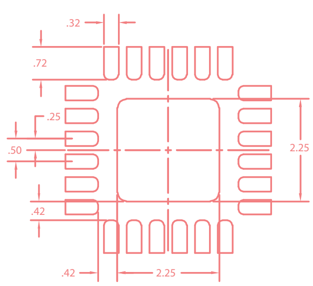


SOLDER STENCIL

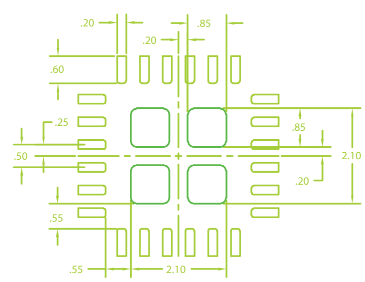
Plastic QFN 4 x 4 mm x 24 leads



METALIZATION

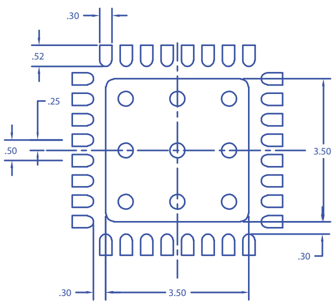


SOLDER MASK

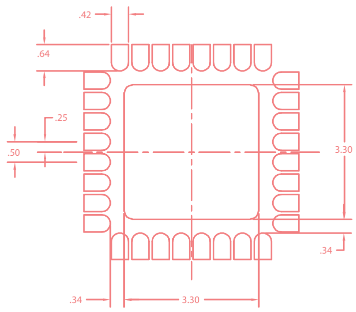


SOLDER STENCIL

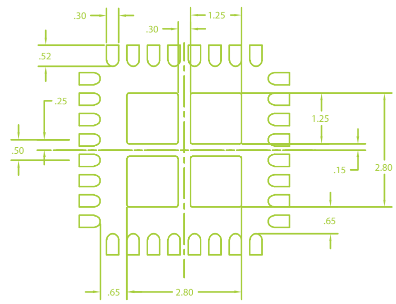
Ceramic QFN 5 x 5 mm x 32 leads



METALIZATION



SOLDER MASK



SOLDER STENCIL

Consider downloading these additional Application Notes:

Application Note 101 - Carrier Tape and Reel Dimensions and Devices Orientation

Application Note 102 - Solder Reflow Profiles for Standard and Lead-Free SMT Packages

Application Note 103 - Biasing Scheme for Dual Passive Supply Amplifier MMICs

For application support and additional technical resources, visit our Support Page.