Application Note

Soldering Guidelines for Mounting ANADIGICS QFNs on PCBs

Revision 2

OBJECTIVE

The objective of this application note is to provide ANADIGICS' customers' general guidelines for PCB second level interconnect design when assembling with ANADIGICS' QFN products.

INTRODUCTION

Reflow soldering of surface mount assemblies provides mechanical, thermal and electrical connections between the component leads or terminations, and the customer surface mount land pads. Solder paste can be applied to the surface mount lands by various methods, the most common being screen-printing and stencil printing [1].

ANADIGICS' QFNs are categorized as surface mount components (SMCs). Unlike through-hole components, SMCs rely entirely upon the solder interface for mechanical strength [2]. The solder joint properties, and therefore the solder joint design, are of critical importance to the user of the SMC.

PCB BOARD DESIGN [3]

Most thermal data for QFNs is based on a 4 layer PCB incorporating vias which act as the thermal path to the layers. (Ref: Jedec Specification JESD 51-5). Two layer boards have no vias, thus any heat sinking must be accomplished in the same plane as the metal traces. This will typically require an increase in the pc board area. Also note that 2 layer heat sinking is only practical for packages with terminals on two sides. There is no routing room when using a package configured with leads on all 4 sides.

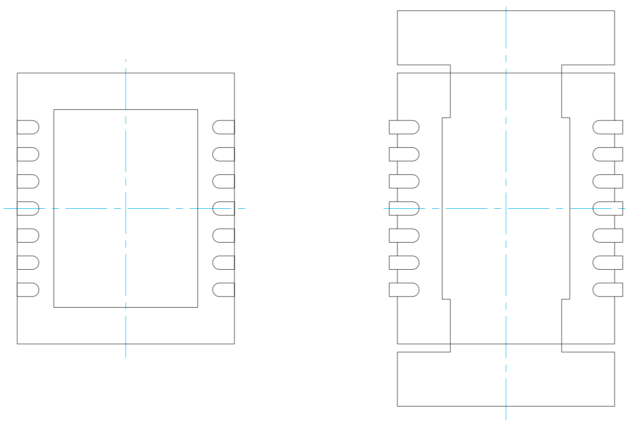


Figure 1: Thermal Routing for Two Layer PCB

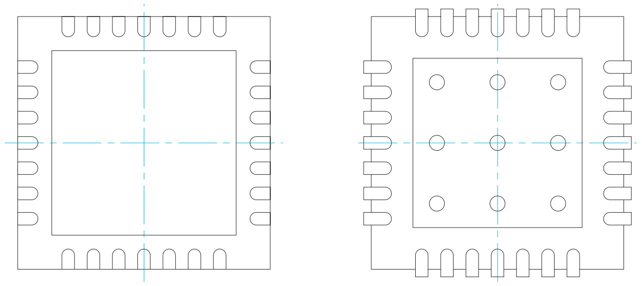


Figure 2: Thermal Routing for Two Layer PCB

Based on thermal performance it is recommended to use 4 layer PCBs with vias to effectively remove heat from the device. Typical thermal vias have the following dimensions: 1.2 mm pitch, 0.3 mm diameter. For 3 mm x 3 mm units, the pitch is reduced to 1 mm. It is important to note that vias should be plugged to prevent voids being formed between the exposed pad and PCB thermal pad due to solder escaping by the capillary effect. **ANADIGICS does not recommend non-capped vias on PCBs for mounting QFNs**.

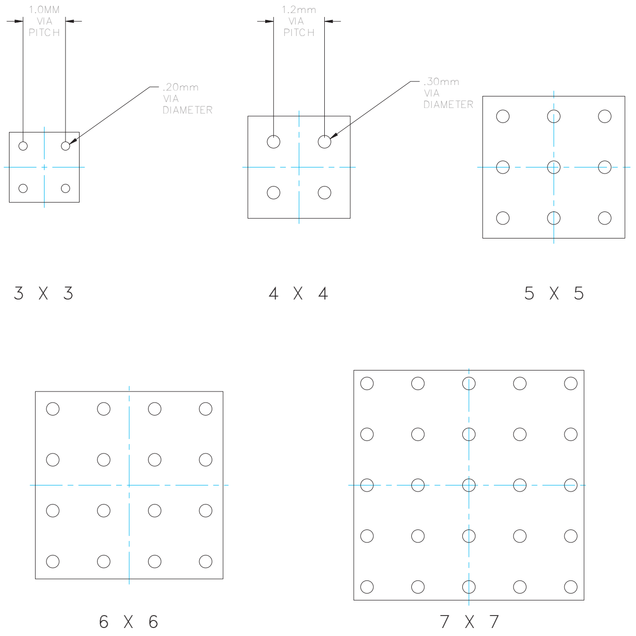


Figure 3: Suggested Thermal via layout for QFNs

Refer to the appendices for the recommended PCB metal design, soldermask design, and stencil print patterns when assembling with Anadigics' QFNs.

It is important to note that the PCB metal design is dependent upon several factors: the electrical and thermal performance requirements of the product and the PCB-to-device interconnect pattern. The PCB metal design recommendations in the appendices primarily deal with the PCB-to-device interconnection. Specific board-level electrical and thermal performance requirements will be dictated by the physical geometry of the specific application and are the responsibility of the end product manufacturer.

SOLDER PASTE APPLICATION METHODS

Solder paste can be applied using screen printing, stencil printing or dispensing, with screen/stencil printing being the most common high volume solder application method. Using these methods, the solder paste is applied on the top surface of the screen or stencil with the print squeegee at one end of the stencil. During the printing process, the squeegee presses down on the stencil to the extent that the bottom of the stencil touches the surface of the board,. The solder paste is then printed on the land through the opening in the stencil when the squeegee traverses the entire length of the image area on the metal mask.

Although the print processes for stencil and screen-printing are similar, the differences lie in the construction of the mask. A screen utilizes an emulsion laminated over a wire mesh. The wire mesh provides mechanical support while the emulsion defines the solder print pattern. Therefore an opening in a screen will contain wire mesh around which the solder paste must flow to reach the PCB surface. A stencil is fabricated from a thin sheet of metal in which the solder print patterns are defined by etched or lasered openings in the metal sheet. Therefore an opening in the stencil will provide an unobstructed path for the solder paste flow. Stencils however have the disadvantage of being dependent on over-etching, which can occur during its fabrication. The choice of stencil or screen depends on the application.

Solder paste can also be dispensed by pressure/time systems, auger valves or positive displacement valves. Using these methods solder paste is dispensed in a serial manner, thus the process rate is much slower than that of stencil or screen-printing. Dispensing is often used in small volume engineering, high product mix or rework applications due to its process flexibility. Additionally, dispensing may have special applications for products that require different print thicknesses on a single board.

SOLDER PASTE DISCUSSION

Solder paste, by one definition, is a homogeneous and kinetically stable which is capable of forming metallurgical bonds at a set of soldering conditions and can be readily adapted to automated production in making reliable consistent solder joints.

In terms of functionality, a solder paste can be considered as being composed of three major components. These are solder alloy powder, vehicle system, and flux system. The vehicle system primarily functions as a carrier for the alloy powder, a compatible matrix for the flux system, and a basis for a desirable rheology. The flux cleans the alloy powder and the substrates to be joined so that high-reliability metallic continuity results and good wetting can be formed. The cleaning process is called fluxing. Several methods are available to achieve fluxing, though the most common is to incorporate the flux into the solder paste as opposed to applying it externally.

The flux is classified based on its activity and chemical nature, namely rosin-based such as RMA, water soluble, and no-clean. Water-soluble flux is designed so that its residue after soldering can be removed by using either pure water or a water medium with an addition of a saponifier or an additive. No-clean solder paste, as its name states, is designed to not require cleaning after it is reflowed. The amount of residue left behind is often designed to not interfere with bed-of-nails/pin (short/open electrical) tests and is nontacky.

The choice of flux depends primarily on the process and several factors including, performance, process, reliability, and cost must be considered when choosing a flux.

The acceptance criterion for use of a particular paste should be as follows:

- The printed solder paste weight should not vary more than 10% among the average measurements taken on one substrate
- The printed paste height should not vary more than ±1 mil among the average measurements taken on one substrate.
- The solder paste pattern would have uniform coverage, without stringing and without separation of flux and solder (bleedout), and it should print without forming a peak.

SOLDER PASTE PROPERTIES [1]

Metal Content

The metal content in solder paste determines the solder fillet size. This is due to the fact that flux activators and other additives, which take up some volume during print, are washed or evaporated away during the reflow and subsequent cleaning processes. Fillet size increases with an increase in the percentage of metal, but the tendency for solder bridging also increases with increase in metal content at a given viscosity. A higher metal content will result in higher thickness of the reflowed solder.

Particle Size and Shape

Powder particle shape determines the oxide content of the powder and as well as the paste's printibility. Larger particles have more surface area thus greater oxide content which needs to be cleaned by the flux in order for the solder to function as desired. Failure to remove the oxide results in formation of solder balls, which are moved aside by surrounding oxide free molten solder. Solder balls, after reflow, are a hazard since they can potentially short metallic conductors.

Solder pastes containing powders of irregular shapes are prone to clog screens and stencils. A commonly used powder size is –200/+352 mesh i.e., at least 99% by weight of the powder particles will pass thought the 200 (holes/square inch) and less that 20% of the powder particles by weight will pass through a 325 mesh.

Flux activators and wetting action

The flux is one of the main constituents of the solder paste vehicle and is activated during the soak zone of the solder reflow profile. The flux activators promote wetting of the molten solder to the surface mount lands and component termination or leads by removing oxide and other surface contaminants. The type of flux has a direct impact on the cleanliness of the assembly. The wetting action of the paste is determined by the activity of its flux.

Solvent and Void formation

The solvent dissolves the flux and imparts the pasty characteristics to the metal powder in the solder paste. It controls the tackiness of the paste by its evaporation under ambient conditions. The solvent should not be hygroscopic. It should have a high flash point and should be compatible with the activator.

The two most important factors that control the formation of voids in fillets are the solvent in the solder paste and the reflow profile. Incomplete outgassing (gasses trapped in the solder joint) is the main cause of voiding and these voids, formed during reflow, lower the strength of the fillet. It is important to have sufficient dwell time in the molten state (above the melting point) to ensure that the gasses have enough time to separate and escape from the molten solder.

Rheological properties

Rheological properties of solder paste such as viscosity, slump, tackiness, and working life are controlled by the addition of thickening agents or secondary solvents.

Additional factors affecting solder performance include equipment and set-up parameters, fabrication methodology, component lead density, operator skills, component and board solderability, as well as ambient temperature and humidity.

PRINT THICKNESS

Both print thickness and print pattern will determine the volume of solder in the resulting joints. Once the proper print area is defined, the thickness can be varied to obtain the proper volume. Solder paste that is too thick will result in excessive solder in the joints. This may cause solder balls and/or solder bridging between components or pads. Solder paste that is too thin may result in insufficient solder fillets and/or voids in the solder. This may degrade the mechanical, thermal and electrical properties of the solder. To the first order, the thickness of the paste print is determined by the thickness of the metal mask of the stencil (or the emulsion thickness and mesh diameter of a screen). Varying the print process parameters and the percent metal content of the solder paste can modify this baseline thickness.

PROBE PLACEMENT

Thermocouple probes are the most common way to determining the temperature being experienced by a SMC/substrate, and in turn, the solder. This information is critical when determining proper reflow profiles, the detail of which have been discussed in the next section.

The Probe location is based on the premise of obtaining the most accurate presentation of the temperature experienced by the SMC. As such, the probes should be placed as close to the actual device as possible. It is also advisable that the probe is resting on the same base as the die. If the die is attached to a panel, then the probe should be placed such that contact between the probe and the panel is not broken during the reflow process.

LEADFREE SOLDERS

The worldwide leadfree initiative for microelectronics has caused the industry to revamp the design, material selection, and manufacturing processes of its products. Japan and Europe have launched industry wide programs banning lead from electronics products in 2005 and 2006, respectively. These programs are backed by OEMs in Japan and by the Parliament in Europe. Although the United States does not have any legislation for the removal of lead from electronic components, in order to remain competitive in this global market, the United States must follow suit or face exclusion from markets requiring the elimination of lead.

Lead based solders are now being replaced by leadfree solders for 2nd level interconnects. The most common replacements for 63Sn37Pb (eutectic point 187 °C) are the 96.5Sn3Ag0.5Cu (SAC305) and 95.5Sn4Ag0.5Cu (SAC405) solders, which have a eutectic point of 217 °C. Due to the non-uniformity of heat distribution on boards during 2nd level interconnection reflow, the SAC solders reflow profiles contain a peak temperature of 250 °C. This peak temperature could potentially be as high at 260 °C depending on the assembler's equipment and process capabilities. Subjecting components to such high temperatures can cause a multitude of problems such as delamination at any of the various interfaces, solder extrusion, external cracking, and cracks on any of the internal features and I/Os.

In order to overcome these problems, ANADIGICS uses innovative designs, processes and materials in the manufacture of its products. The following table provides a few leadfree solders being used in industry. Although no standard replacement has been identified by the microelectronics industry, research organizations specializing in lead-free solders have recommended several replacements for the standard Sn37Pb solder.

Table 1: Common Leadfree solders [4,5,6]

ALLOY	MELTING POINT (°C)	INDUSTRY
Sn3.5Ag	221 - 226	Automotive
Sn3.5Ag3Bi	208 - 217	Military/Aerospace
Sn3.5Ag1Bi4In	208 - 213	Consumer
Sn3.0Ag0.5Cu	217	Automotive Telecom.
Sn57Bi	138	Consumer
Sn0.7Cu	227	Consumer Telecom.
Sn9Zn	199	Consumer

PROPERTIES OF LEADFREE SOLDERS [1]

Some of the properties common to leadfree solders are given below:

- Typically have much higher melting points than traditional Sn63Pb37.
- Are stronger and less ductile than lead or indium-bearing alloys.
- Typically exhibit poor wetting when compared to Sn37Pb.

SAC305 and SAC405 provide certain advantages over other leadfree compounds due to the presence of copper.

- Copper in solder retards the dissolution rate of copper from boards.
- Copper can improve the wettability of the solder during reflow.
- Copper can improve the thermo-mechanical properties of solder joints.
- Copper reduces the melting point slightly. 96.5Sn3.5Ag has a melting point of 221 °C.

Table 2: SAC305/SAC403 data [5,7]

CATEGORY	PROPERTY
Melting Point	217 °C
Tack Force	2.63 g/mm^2
Density	7.44
Elec. Resistivity	13 $\mu\Omega$-cm
CTE	~14
Ultimate Tensile Strength	42 MPa
Yield Strength	45 MPa
Shear Strength	27 MPa
Creep	27 MPa (100 hrs to fail)
Wetting Time	~0.27 sec

Reflow Specifications

The reflow profile is a critical part of the PCB assembly process. A proper reflow profile must provide adequate time for flux volatilization, proper peak temperature, time above liquidus, ramp up and cool down rates. The profile used has a direct bearing on manufacturing yield, solder joint integrity, and the reliability of the assembly [8]. A typical reflow profile is made up of four distinct zones: the preheat zone, the soak zone/flux activation zone, the reflow zone, and the cooling zone [9].

Preheat Zone

Typically the heating rate in the preheat zone should be 2 °C to 4 °C/second and the peak temperature in this zone should be 100-125 °C. If the temperature ramp is too fast, the solder paste may splatter and cause formation of solder balls. Also, to avoid thermal shock to sensitive components such as ceramic chip resistors, the maximum heating rate should be controlled.

Soak Zone

The soak zone is intended to allow the board and components to reach a uniform temperature, minimizing thermal gradients. The soak zone also acts to activate the flux within the solder paste. The ramp rate in this zone is very low and the temperature is raised near the melting point of solder (183 °C for standard 63Sn27Pb solder and 217 for SAC305/SAC405 solder). The consequences of being at too high a temperature in the soak zone are solder balls due to insufficient fluxing (when the ramp rate is to fast) and solder splatter due to excessive oxidation of paste (when the ramp rate is too slow). Typical soak times are usually around the range of 130 –170 °C for 60 to 90 seconds.

Reflow Zone

In this zone the temperature is kept above the melting point of the solder for 30 to 60 seconds. The peak temperature in this zone should be high enough for adequate flux action and to obtain good wetting. For standard 63Sn37Pb solders, a peak temperature range of 215 – 220 °C is generally considered acceptable. For leadfree solders this range goes up to 250-260 °C.

The temperature, however, should not be so high as to cause component damage, board damage, discoloration or charring of the board. Extended duration above the solder melting point will damage temperature sensitive components and potentially create excessive intermetallic growth between the solder and the I/O pad metallization which makes the solder joint brittle and reduces solder joint fatigue resistance. Additionally high temperatures can promote oxide growth, depending upon the furnace atmosphere, which can degrade solder wetting.

Cooling Zone

The cooling rate of the solder joint after reflow is also important. For a given solder system, the cooling rate is directly associated with the resulting microstructure which in turn, affects the mechanical behavior of solder joints. The faster the cooling rate, the smaller the grain size of the solder will be, and hence the higher the fatigue resistance of the solder joint. Conversely rapid cooling will result in residual stresses between TCE mismatched components. Therefore the cooling rate needs to be optimized.

The profile of choice can affect any of the following areas, to a different degree, by one of more of the profile zones [8].

- Temperature distribution across the assembly
- Plastic IC package cracking
- Solder balling
- Solder beading
- Wetting ability
- Residue cleanability
- Residue appearance and characteristics
- Solder joint voids
- Metallurgical reactions between solder and substrate surface
- Board flatness
- Microstructure of solder joints
- Residual stress level of the assembly

REFLOW PROFILES

Table 3 provides a breakdown of the reflow conditions provided by the JEDEC standard J-STD-020C [10] for lead-based solders.

Table 3: Standard Reflow Profile Breakdown

	JEDEC SPECIFICATIONS
Avg. Ramp-up (T_L to T_P)	3 °C/second max
Dwell Time (125 ± 25 °C)	60-120 seconds
Time above T_L	60-150 seconds
Time within 5 °C of peak	10-30 seconds max
Peak Temperature (JEDEC)	240 -5/+0 °C
Average Ramp-down	6 °C/second max

*T_L is the solder Eutectic temperature
*T_P is the peak temperature

Table 4 provides a breakdown of the reflow conditions provided by the JEDEC standard J-STD-020C [10] for leadfree solders. While, this standard specifies a peak reflow temperature of 260 °C, the actual peak temperature subjected to the parts during qualification will be dependent on the particular products' high temperature tolerating capabilities.

Table 4: Leadfree MSL Reflow Profile Breakdown [10]

	JEDEC SPECIFICATIONS
Avg. Ramp-up (T_L to T_P)	3 °C/second max
Dwell Time (175 ± 25 °C)	60-180 seconds
Ramp-up 200 °C to 217 °C	3 °C/second max
Time above 217 °C	60-150 seconds
Time within 5 °C of peak	20-40 seconds max
Peak Temperature*	260 -5/+0 °C
Average Ramp-down	6 °C/second max

*T_L is the solder Eutectic temperature
*T_P is the peak temperature
*Actual peak temperature will be product dependent

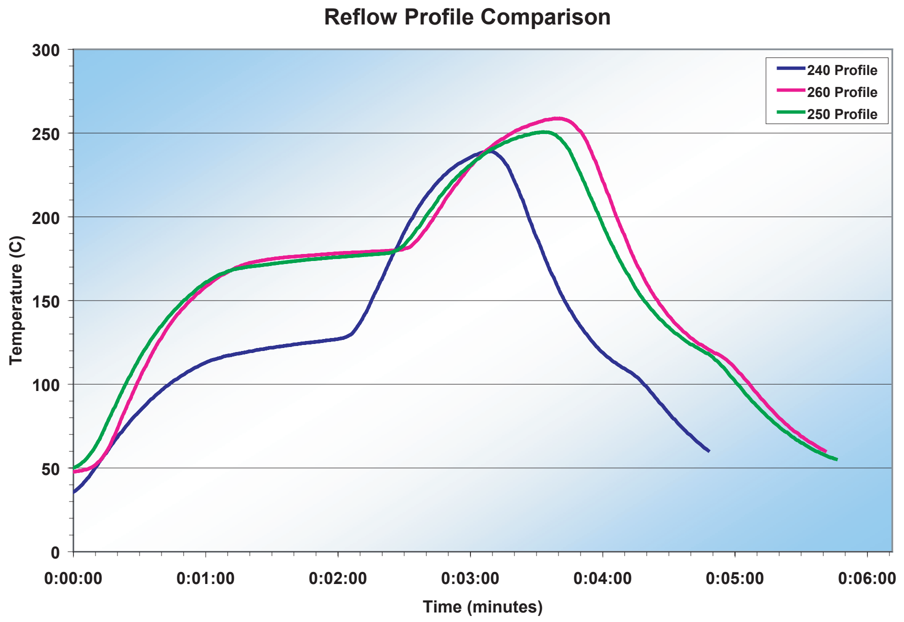


Figure 4. Comparison of product reflow temperature profiles

REWORK [3]

MLP rework procedures are an adaptation (and in some cases a simplification) of Ball Grid Array Package rework procedures. The basic elements of this procedure are as follows:

- PC board preheat
- Reflow of component solder
- Vacuum removal of component
- Cleaning and prep of pcb lands
- Screening of solder paste
- Placement and reflow of new component
- Inspection of solder joints

PC Board Preheat

It is recommended to bake the PCB for approx 4 hours at 125 °C prior to rework in order to drive off residual moisture that could cause other component failures during the rework reflow process. Once the PCB has finished the baking process, the PCB under rework is then placed in the rework holder and heated again to a temperature of 125 °C. Localized heating of the area under rework is recommended.

Reflow / Removal of the Component from the PCB

Specialized vacuum collets come in contact with the rework component. These collets incorporate a hot gas shroud that heats up the part to a temperature required for reflowing the solder interconnects. Once the solder reflows, the vacuum collet lifts the unit from the PCB. The collet size and hot gas flow should be optimized to keep the heat flow localized.

Cleaning and Prep of the PCB Land

The PCB can be cleaned and prepared using conventional tools and processes currently used for gullwing packages. Removal of excess solder using a hot iron, a small scraping tool and solder wick is typical. Place the solder wick under the scraping mechanism to remove the solder from the land area.

Screen Printing of Solder Paste

Based on some of the tight geometries used on today's PCB's, it is difficult to screen print a PCB that is nearly 100% populated with components. Hence the approach of screen printing the solder paste directly onto the new component has been adopted. It is recommended to use a type 3 or 4 Printing no-clean solder paste.

Placement and Reflow of Component

The placement of the new QFN component should be done with a split field vision system. The image of the screen printed component and the PCB land pattern are superimposed during the placement operation, thus making the placement easier to align with the terminal footprint. The reflow of the new component should be done with a localized gas shroud similar to that used during the component removal operation. The profile used for the reflow should have ramp rates and peak temperatures that follow the guidelines specified in JEDEC-STD-20C.

REFERENCES

[1] Ray P. Prasad; *Surface Mount Technology - Principles and Practice*; Van Nostrand Reinhold – New York; 1989; Pages 311-328.

[2] http://www.tutorialsweb.com/smt/smt.htm

[3] Application note. *MLP – Micro Leadframe Package, A comprehensive User's Guide.* CARSEM. April 2002.

[4] http://www.kester.com/leadfree_alloys.htm

[5] John H. Lau, *Electronic Manufacturing with leadfree, halogen-free & conductive-Adhesive Materials.* McGraw Hill Publishing. 2003.

[6] Jennie Hwang, *Environment-Friendly Electronics: Lead-free Technology.* Electrochemical Publications Ltd. Isle of Man, 2001.

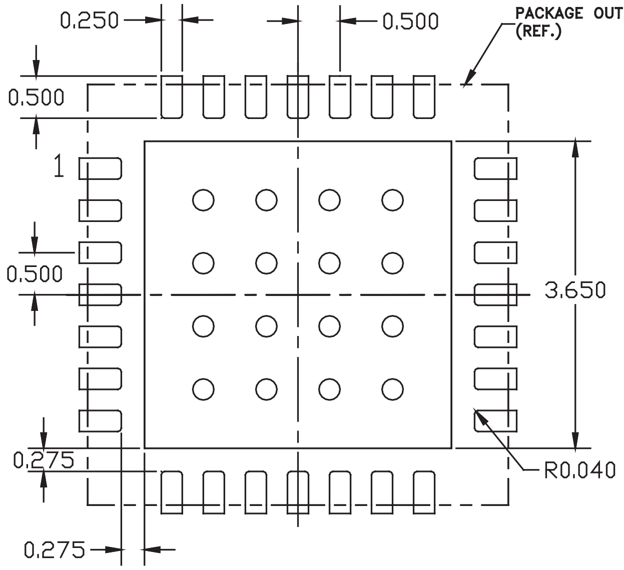
[7] http://www.omnixsolderpaste.com/pdfs/OM_310.PD

[8] Charles Harper; *Electronic Packaging and Interconnect Handbook;* "Solder Technologies for Electronic Packaging Assembly"; McGraw-Hill 2000; Pages 6.1-6.83.

[9] http://www.ecd.com/emfg/instruments/tech1.asp

[10] JEDEC Standard J-STD-020C. *Moisture/Reflow Sensitivity Classification for non-hermetic Solid State Surface Mount Devices.* July 2004.

SPECIFICATION: AN–P8002427 RevB
SUBJECT: PCB METAL & SOLDERMASK
QFN 5mm x 5mm
28 LEAD



PCB METAL
TOP (X-RAY) VIEW

NOTES:

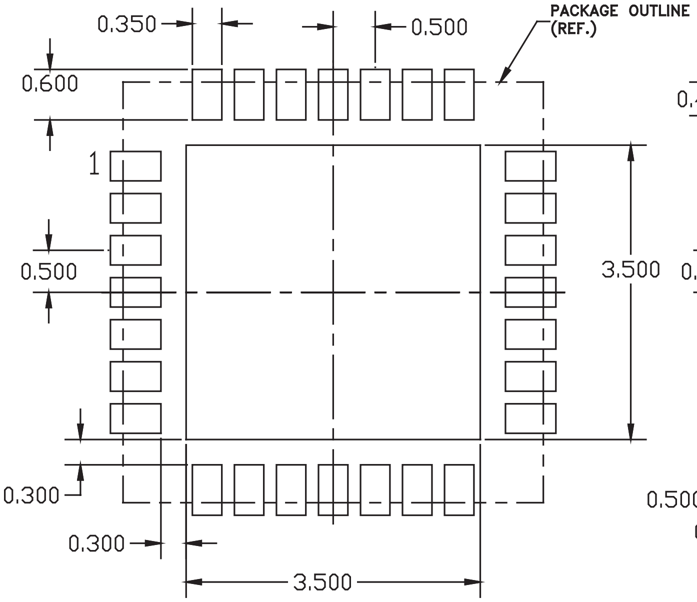
(1) OUTLINE DRAWING REFERENCE:
98001–054 & 98001–054A

(2) UNLESS SPECIFIED DIMENSIONS ARE SYMMETRICAL ABOUT CENTER LINES SHOWN.

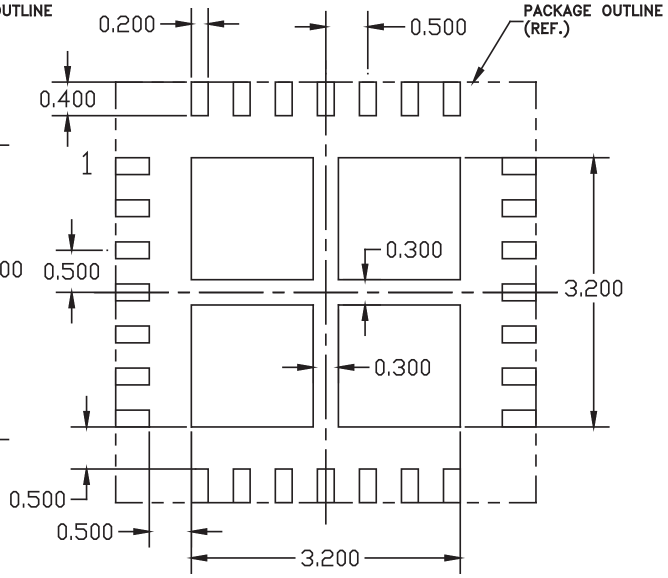
(3) DIMENSIONS IN MILLIMETERS.

(4) VIAS SHOWN IN PCB METAL VIEW ARE FOR REFERENCE ONLY.
NUMBER & SIZE OF THERMAL VIAS REQUIRED DEPENDENT ON HEAT DISSIPATION REQUIREMENT AND THE PCB PROCESS CAPABILITY.

(5) RECOMMENDED STENCIL THICKNESS:
APPROX. 0.125mm (5 Mils)



PCB SOLDER MASK
TOP (X-RAY) VIEW



STENCIL APERTURE
TOP (X-RAY) VIEW

APPENDIX A: 5 mm x 5 mm (28 Lead) QFN Package Outline

SPECIFICATION: AN_P8002418
SUBJECT: PCB METAL & SOLDERMASK
QFN 7mm x 7mm
48 LEAD

NOTES:

(1) OUTLINE DRAWING REFERENCE: P8002418

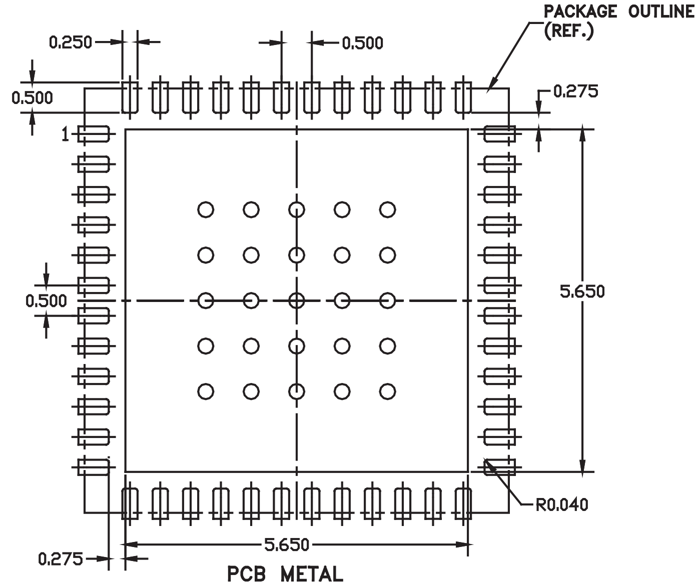
(2) UNLESS SPECIFIED DIMENSIONS ARE SYMMETRICAL ABOUT CENTER LINES SHOWN.

(3) DIMENSIONS IN MILLIMETERS.

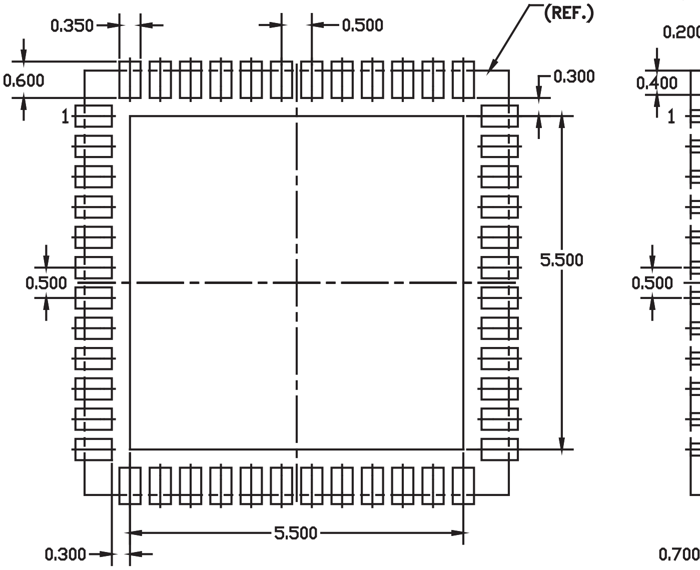
(4) VIAS SHOWN IN PCB METAL VIEW ARE

FOR REFERENCE ONLY.
NUMBER & SIZE OF THERMAL VIAS REQUIRED DEPENDENT ON HEAT DISSIPATION REQUIREMENT AND THE PCB PROCESS CAPABILITY.

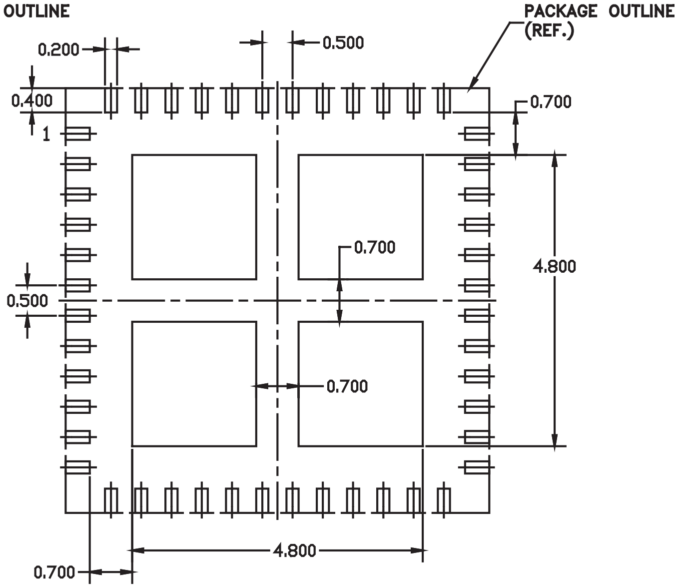
(5) RECOMMENDED STENCIL THICKNESS: APPROX. 0.125mm (5 Mils)



PCB METAL
TOP (X-RAY) VIEW



PCB SOLDER MASK
TOP (X-RAY) VIEW



STENCIL APERTURE
TOP (X-RAY) VIEW

APPENDIX B: 7 mm x 7 mm (48 Lead) QFN Package Outline

SPECIFICATION: AN–P8001_043 RevA
SUBJECT: PCB METAL & SOLDERMASK
QFN 3mm x 3mm
12 LEAD

NOTES:

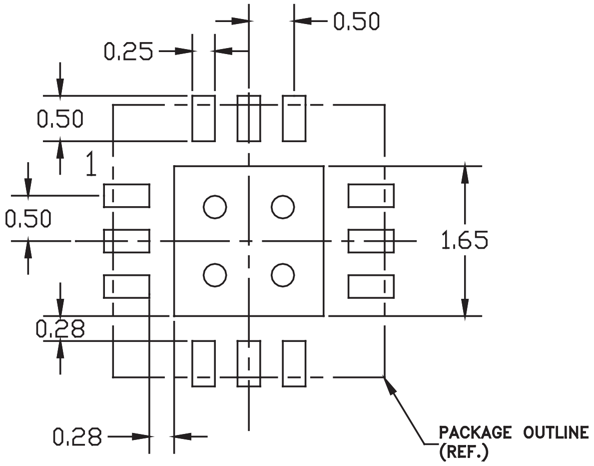
(1) OUTLINE DRAWING REFERENCE:
98001–043

(2) UNLESS SPECIFIED DIMENSIONS ARE SYMMETRICAL ABOUT CENTER LINES SHOWN.

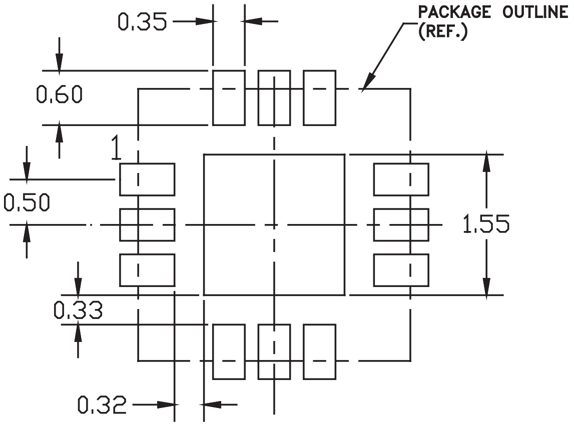
(3) DIMENSIONS IN MILLIMETERS.

(4) VIAS SHOWN IN PCB METAL VIEW ARE FOR REFERENCE ONLY.
NUMBER & SIZE OF THERMAL VIAS REQUIRED DEPENDENT ON HEAT DISSIPATION REQUIREMENT AND THE PCB PROCESS CAPABILITY.

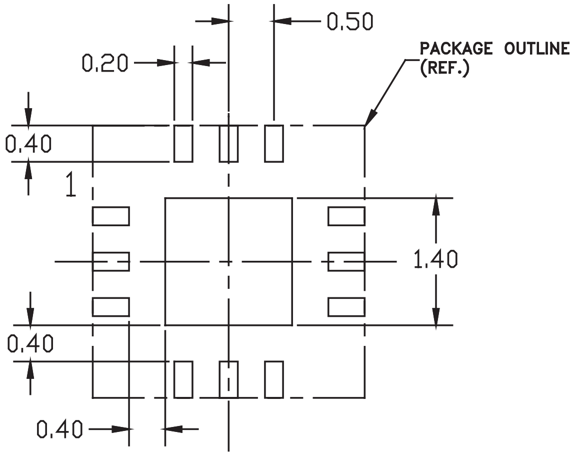
(5) RECOMMENDED STENCIL THICKNESS:
APPROX. 0.125mm (5 Mils)



PCB METAL
TOP (X–RAY) VIEW



PCB SOLDER MASK
TOP (X–RAY) VIEW



STENCIL APERTURE
TOP (X–RAY) VIEW

APPENDIX C: 3 mm x 3 mm (12 Lead) QFN Package Outline

SPECIFICATION: AN–P8001_047RevA
SUBJECT: PCB METAL & SOLDERMASK
QFN 3mm x 3mm
16 LEAD

NOTES:

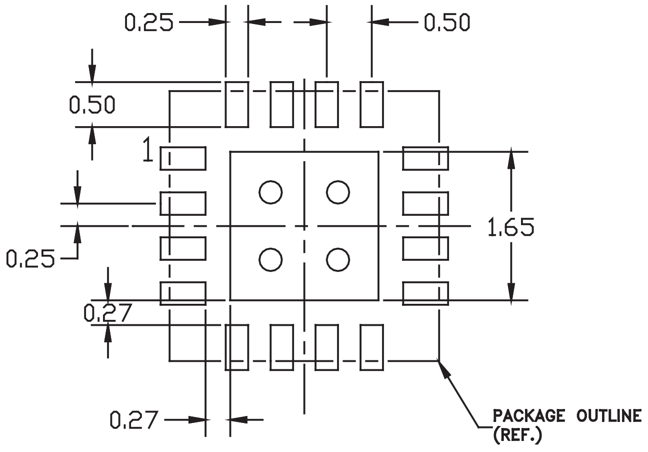
(1) OUTLINE DRAWING REFERENCE:
98001–047 & 98001–069

(2) UNLESS SPECIFIED DIMENSIONS ARE SYMMETRICAL ABOUT CENTER LINES SHOWN.

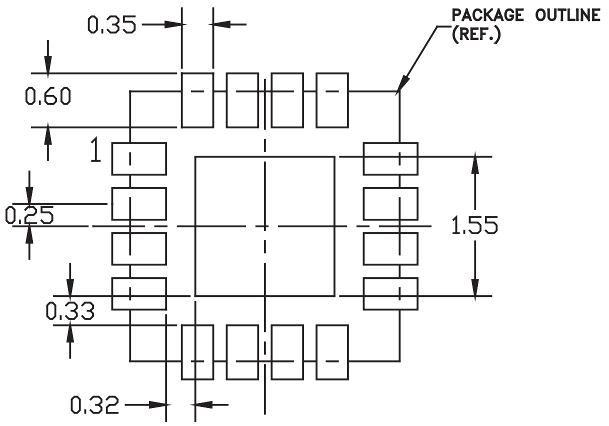
(3) DIMENSIONS IN MILLIMETERS.

(4) VIAS SHOWN IN PCB METAL VIEW ARE FOR REFERENCE ONLY.
NUMBER & SIZE OF THERMAL VIAS REQUIRED DEPENDENT ON HEAT DISSIPATION REQUIREMENT AND THE PCB PROCESS CAPABILITY.

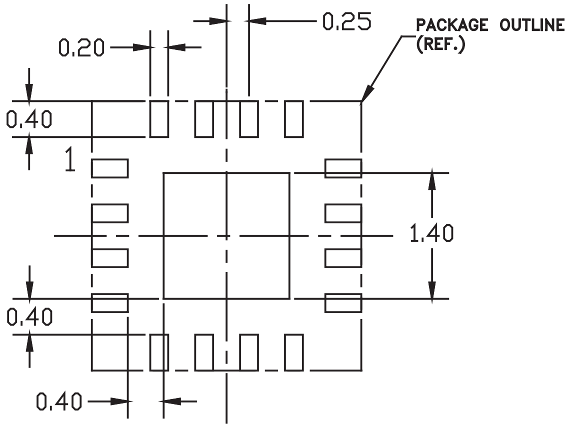
(5) RECOMMENDED STENCIL THICKNESS:
APPROX. 0.125mm (5 Mils)



PCB METAL
TOP (X–RAY) VIEW



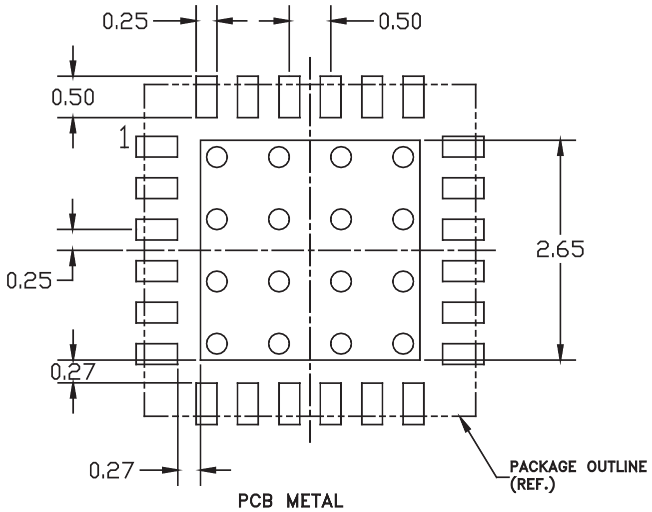
PCB SOLDER MASK
TOP (X–RAY) VIEW



STENCIL APERTURE
TOP (X–RAY) VIEW

APPENDIX D: 3 mm x 3 mm (16 Lead) QFN Package Outline

SPECIFICATION: AN–P8001_067RevA
SUBJECT: PCB METAL & SOLDERMASK
QFN 4mm x 4mm
24 LEAD



PCB METAL
TOP (X–RAY) VIEW

NOTES:

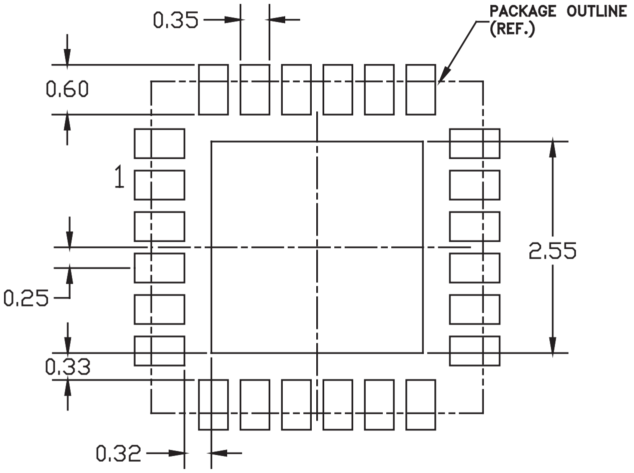
(1) OUTLINE DRAWING REFERENCE:
98001–067 & 98001–070

(2) UNLESS SPECIFIED DIMENSIONS ARE SYMMETRICAL ABOUT CENTER LINES SHOWN.

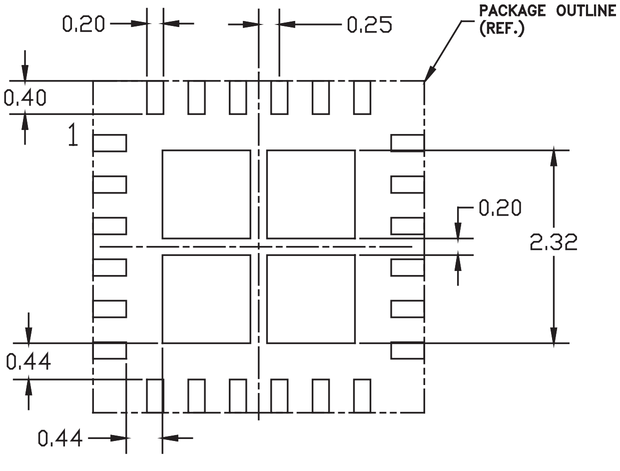
(3) DIMENSIONS IN MILLIMETERS.

(4) VIAS SHOWN IN PCB METAL VIEW ARE FOR REFERENCE ONLY.
NUMBER & SIZE OF THERMAL VIAS REQUIRED DEPENDENT ON HEAT DISSIPATION REQUIREMENT AND THE PCB PROCESS CAPABILITY.

(5) RECOMMENDED STENCIL THICKNESS:
APPROX. 0.125mm (5 Mils)



PCB SOLDER MASK
TOP (X–RAY) VIEW



STENCIL APERTURE
TOP (X–RAY) VIEW

APPENDIX E: 4 mm x 4 mm (24 Lead) QFN Package Outline

NOTES

ANADIGICS, Inc.
141 Mount Bethel Road
Warren, New Jersey 07059, U.S.A.
Tel: +1 (908) 668-5000
Fax: +1 (908) 668-5132

URL: http://www.anadigics.com
E-mail: Mktg@anadigics.com