ANADIGICS

# Application Note

## Soldering Guidelines for Module PCB Mounting

Rev 3

## OBJECTIVE

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

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

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.

## 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.

## 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 liquidous, 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 [3]. 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 [4].

### 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 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). 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

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, 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 [3].

- 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 1 provides a breakdown of the reflow conditions provided by the JEDEC standard J-STD-020B [5] for lead-based solders.

**Table 1: 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 |

*Notes:*
*(1) $T_L$ is the solder Eutectic Temperature*
*(2) $T_P$ is the peak Temperature*

Table 2 provides a breakdown of the reflow conditions provided by the JEDEC standard J-STD-020B [5] for leadfree solders. Note that Anadigics' leadfree products are qualified to meet peak reflow temperatures of 260 –5/+0 C, 10 degrees higher than JEDEC recommendations.

**Table 2: Lead-free MSL Reflow Profile Breakdown**

| | 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 | 10-30 seconds max |
| Peak Temperature (JEDEC) | 250 -5/+0 °C |
| Peak Temperature (ANAD) | 260 -5/+0 °C |
| Average Ramp-down | 6 °C/second max |

*Notes:*
*(1) $T_L$ is the solder Eutectic Temperature*
*(2) $T_P$ is the peak Temperature*

Figure 1 compares the 240 °C and 260 °C profiles.

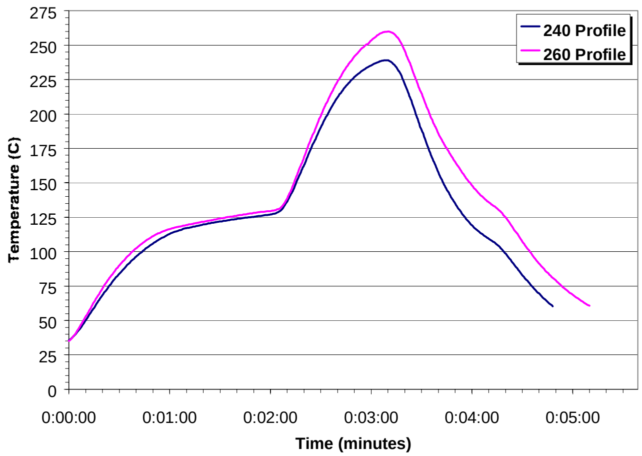



**Figure 1: Comparison of the 240 °C and 260 °C Reflow Profiles**

## REWORK GUIDELINE

The most common method of repairing surface mount devices is by using hot air devices. During this rework process care should be taken to prevent thermal damage to adjacent component or substrates. The following guidelines should be used to prevent thermal damage and to produce an acceptable solder joint after repair/rework [1]:

- Characterize the rework process carefully so as not to overheat and damage the device.
- Keep the number of times a part is removed and replaced to a maximum of two.
- Preheat the substrate for about 30 minutes to about 95 °C.
- Use an appropriate attachment to direct the flow of hot air to the component to be removed or replaced.
- Minimize the heat time to reduce the device exposure to high temperatures.

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

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

[5] JEDEC Standard J-STD-020B. *Moisture/Reflow Sensitivity Classification for non-hermetic Solid State Surface Mount Devices.* July 2002.

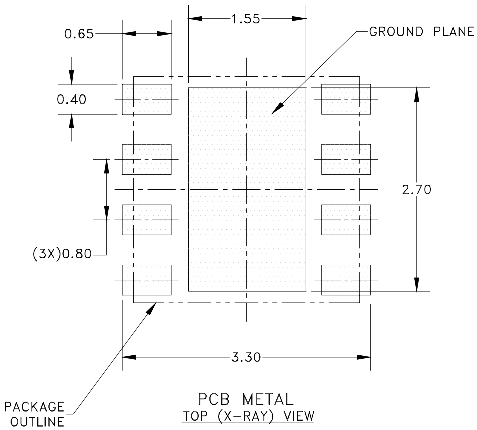
[6] ANADIGICS Application Note: Solder Reflow Report. Revision 1.

[7] ANADIGICS Application Note: High Temperature Report. Rev. 1

SPECIFICATION: P8002131 2nd LEVEL INTERCONNECT 3mm X 3mm MODULE.
SUBJECT: PCB DESIGN

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| – | INITIAL RELEASE | M.A.W. M.J.A. W.E.A. | 10-06-03 | A | METAL & SOLDER MASK GND.PLANE WIDTH DIMENSIONS CORRECTED. | M.A.W. W.E.A. | 11-12-03 |
| | | | | | | | |






PCB METAL
TOP (X-RAY) VIEW

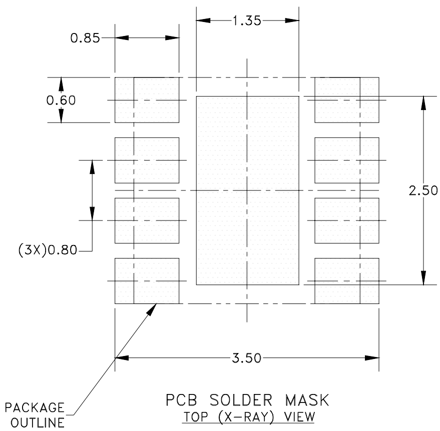
ONLY PACKAGE I/O's AND GROUND REQUIREMENTS SHOWN.

NOTES:

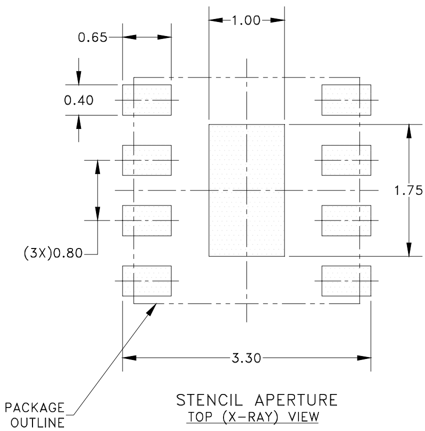
(1) OUTLINE DRAWING REFERENCE:
 * P8002087
 * P8002088

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

(3) DIMENSIONS IN MILLIMETERS.




PCB SOLDER MASK
TOP (X-RAY) VIEW



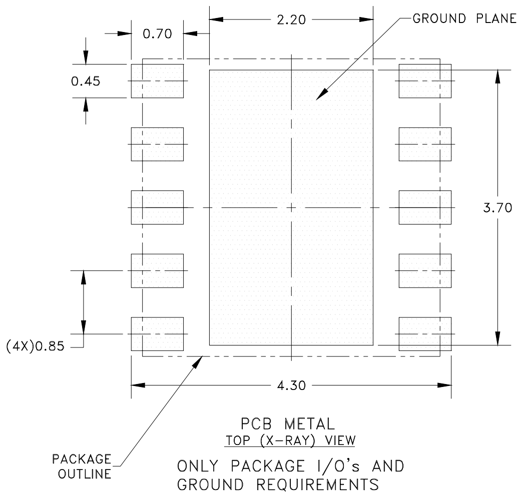

STENCIL APERTURE
TOP (X-RAY) VIEW

## APPENDIX A: 3 x 3 mm Module Package Outlines

SPECIFICATION: P8002132    2nd LEVEL INTERCONNECT
SUBJECT: PCB DESIGN    4mm X 4mm MODULE.

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| – | INITIAL RELEASE | M.A.W. M.J.A. W.E.A. | 10-06-03 | A | CORRECTED PCB SOLDER MASK I/O DIMENSIONS. | M.A.W. W.E.A. | 11-12-03 |
| | | | | | | | |






PCB METAL
TOP (X-RAY) VIEW

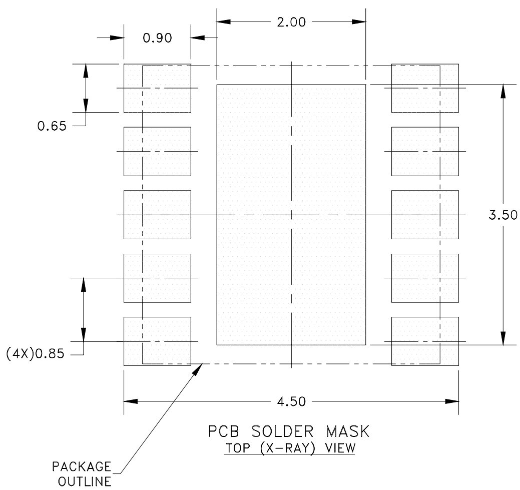
ONLY PACKAGE I/O's AND GROUND REQUIREMENTS SHOWN.

NOTES:

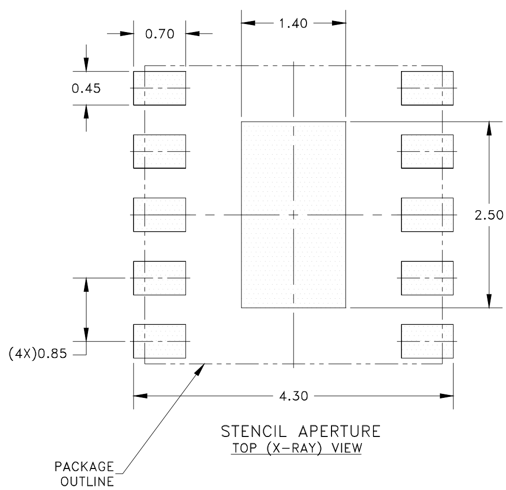
(1) OUTLINE DRAWING REFERENCE:
- * 98000-042
- * 98000-045
- * P8002079

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

(3) DIMENSIONS IN MILLIMETERS.




PCB SOLDER MASK
TOP (X-RAY) VIEW



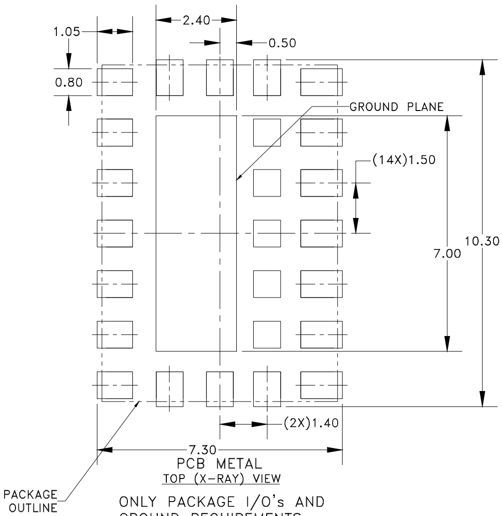

STENCIL APERTURE
TOP (X-RAY) VIEW

## APPENDIX B: 4 x 4 mm Module Package Outlines

SPECIFICATION: P8002138 2nd LEVEL INTERCONNECT
SUBJECT: PCB DESIGN 7mm X 10mm MODULE.

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| A | Resized Stencil Openings | M.A.W. | 10/22/03 | – | – | – | – |
| | | | | | | | |



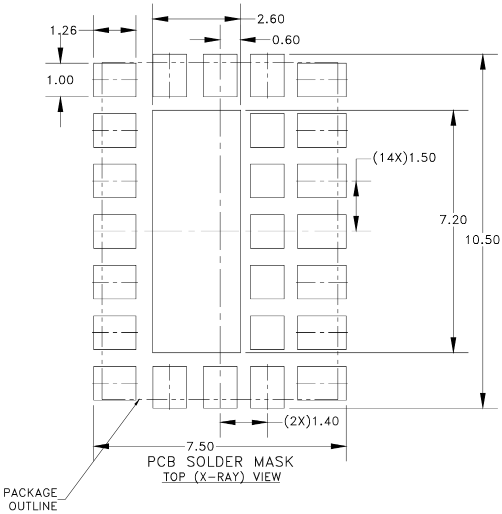


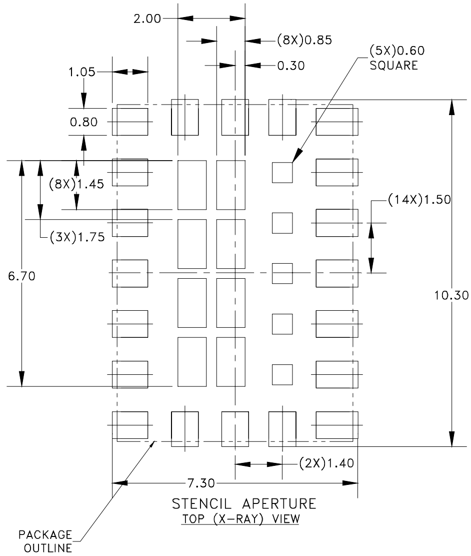

NOTES:
(1) OUTLINE DRAWING REFERENCE: 98000–044

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

(3) DIMENSIONS IN MILLIMETERS.

ONLY PACKAGE I/O's AND GROUND REQUIREMENTS SHOWN.



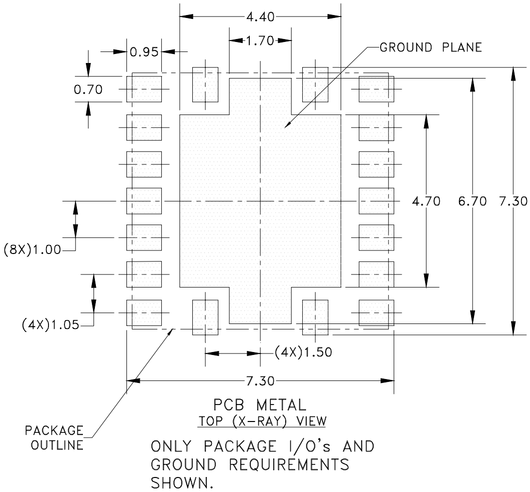




## APPENDIX C: 7 x 10 mm Module Package Outlines

SPECIFICATION: P8002144 2nd LEVEL INTERCONNECT
SUBJECT: PCB DESIGN 7mm X 7mm MODULE.

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| – | INITIAL RELEASE | M.A.W. W.E.A. | 11-12-03 | – | – | – | – |
| | | | | | | | |

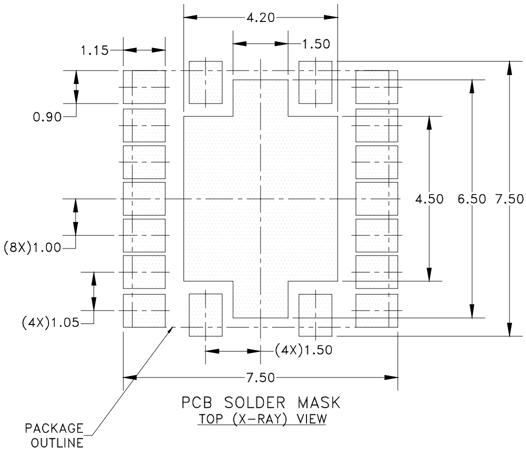


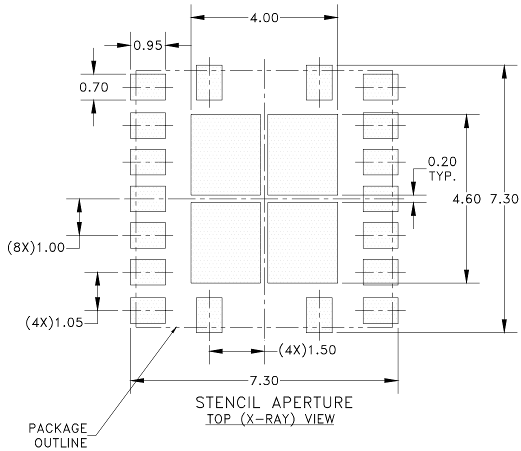



NOTES:
(1) OUTLINE DRAWING REFERENCE: P8002109

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

(3) DIMENSIONS IN MILLIMETERS.




## APPENDIX D: 7 x 7 mm Module Package Outlines

SPECIFICATION: P8002147 (sheet 1 of 3) 2nd LEVEL INTERCONNECT
SUBJECT: PCB DESIGN 11mm X 10.5mm MODULE.

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| – | INITIAL RELEASE | M.A.W. W.E.A. | 12-09-03 | – | – | – | – |
| | | | | | | | |



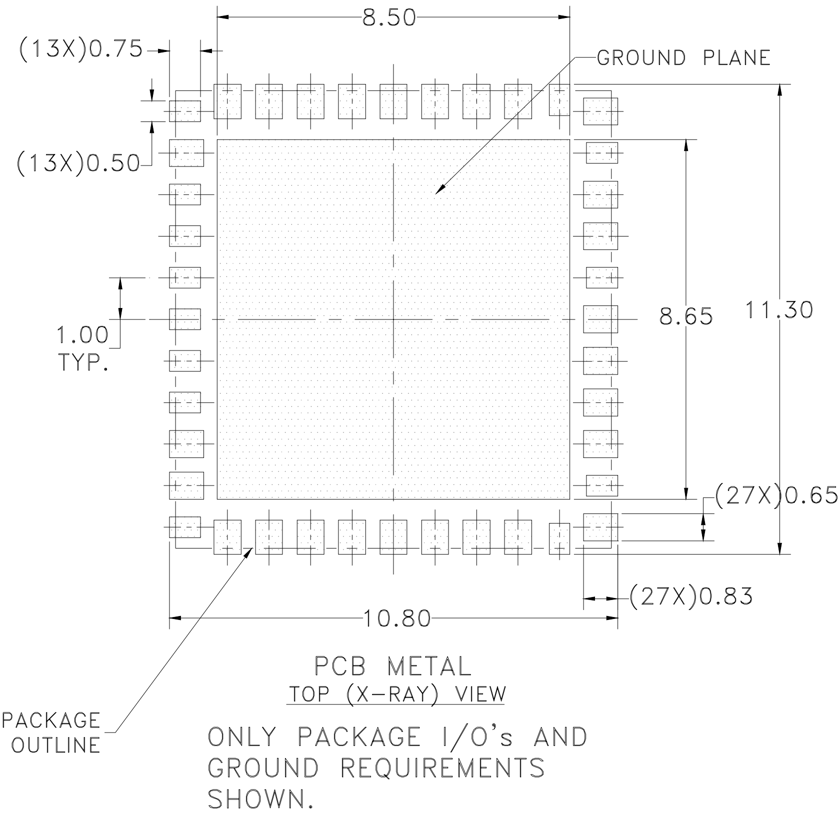
NOTES:

(1) OUTLINE DRAWING REFERENCE: P8002082

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

(3) DIMENSIONS IN MILLIMETERS.

(4) PCB ACAD AND GERBER FILES AVAILABLE FOR CLARIFICATION.




## APPENDIX E: 11 x 10.5 mm Module PCB Metal Package Outlines

SPECIFICATION: P8002147 (sheet 2 of 3) 2nd LEVEL INTERCONNECT
SUBJECT: PCB DESIGN 11mm X 10.5mm MODULE.

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| – | INITIAL RELEASE | M.A.W.<br>W.E.A. | 12-09-03 | – | – | – | – |
| | | | | | | | |



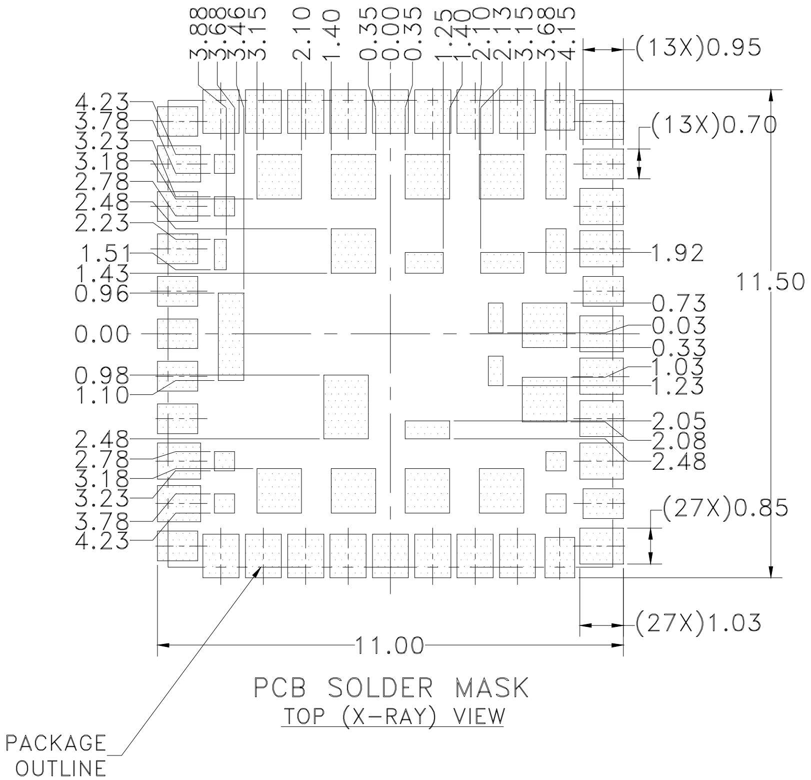
NOTES:

(1) OUTLINE DRAWING REFERENCE: P8002082

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

(3) DIMENSIONS IN MILLIMETERS.

(4) PCB ACAD AND GERBER FILES AVAILABLE FOR CLARIFICATION.




**APPENDIX F: 11 x 10.5 mm Module PCB Solder Mask Package Outlines**

SPECIFICATION: P8002147 (sheet 3 of 3) 2nd LEVEL INTERCONNECT
SUBJECT: PCB DESIGN 11mm X 10.5mm MODULE.

| REV. | DESCRIPTION | CHCKD./APPVD. | DATE | REV. | DESCRIPTION | CHCKD./APPVD. | DATE |
|---|---|---|---|---|---|---|---|
| – | INITIAL RELEASE | M.A.W. W.E.A. | 12-09-03 | – | – | – | – |
| | | | | | | | |



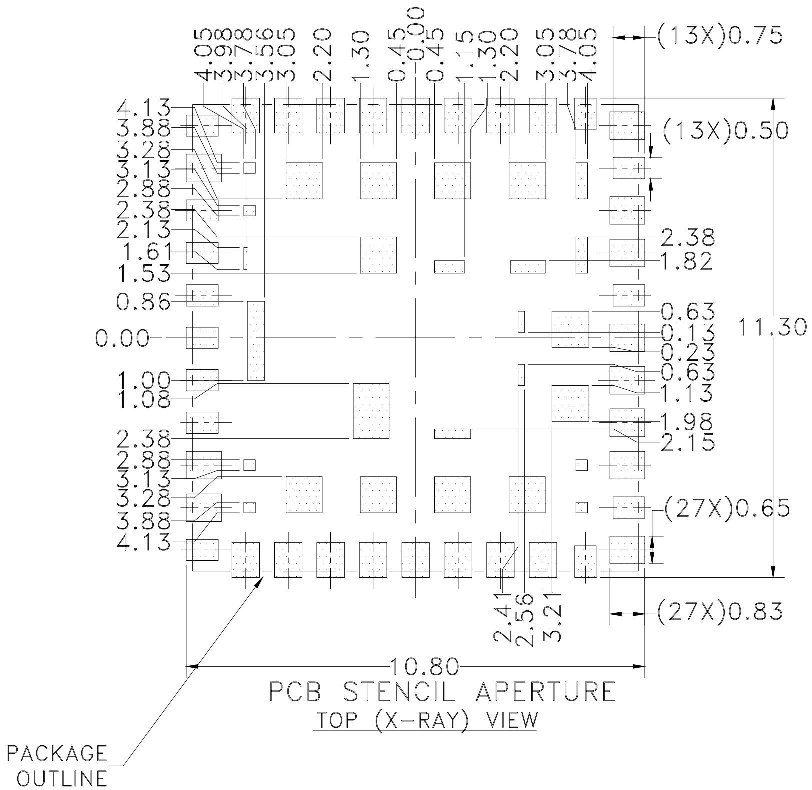
NOTES:

(1) OUTLINE DRAWING REFERENCE: P8002082

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

(3) DIMENSIONS IN MILLIMETERS.

(4) PCB ACAD AND GERBER FILES AVAILABLE FOR CLARIFICATION.




## APPENDIX G: 11 x 10.5 mm Module PCB Stencil Aperture Package Outlines

**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