Recommended PCB Design & Surface Mount Guidelines for Picor Corporation's LGA Packages

Contents

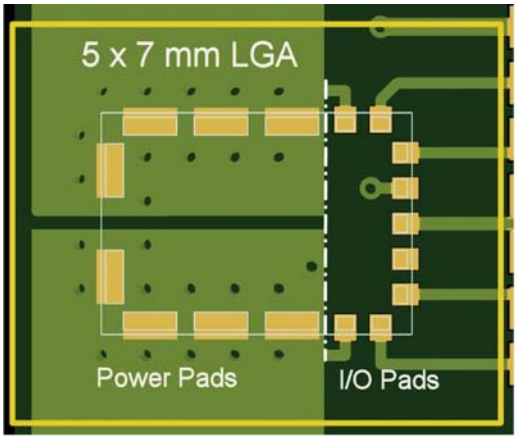


***Figure 1** – Typical 5 x 7 mm LGA receptor PCB layout with a keep-out border for re-working.*

Introduction

Picor's LGA (Land Grid Array) products are high performance, high density solutions suitable for a variety of power applications. The LGA is designed with minimal internal stray parasitic elements, to improve electrical efficiency and dynamic performance and has extremely low thermal resistance to allow for optimal heat transfer from the internal semiconductor junction to the leads of the package. In order to harness the full value of Picor's LGA solutions from a system perspective it is important to follow a few simple guidelines, as explained in this application note, regarding PCB layout and surface mount techniques.

Receiving PCB Footprint

Figure 1 illustrates a typical 5 x 7 mm LGA layout with its external components. The smaller pads along the right half of the LGA are the input and output connections, along with the power and ground connections. These receiving pads are copper defined as shown in Figure 2. The recommended receiving pads are 0.05mm longer on the "sides" and the "heel", and 0.2mm longer in the "toe" than the package pad.[1] This allows for a proper solder fillet during re-flow. The solder-mask is expanded by 0.025mm around the pad edges (copper defined pads). The yellow box around the LGA, as shown in Figure 1, is an optional keep-out border that allows a hot-air re-working nozzle to reach the LGA unobstructed.

The larger pads on the LGA, shown on the left side in Figure 1, are the power connections. The receptor pattern for these pads should be solder-mask defined. The dimensions shown in Figure 3 are the finished solder mask opening dimensions, as they relate to the example in Figure 1. The dimensions shown in Figure 4 are for use when a power pad is not located along the edge of the LGA, but is located completely underneath the body of the LGA. Over-etching of the solder mask could cause a solder mask breakout to occur in areas where the solder mask is the thinnest, which in turn may cause solder to be leeched away from an adjoining pad. Check with the PCB vendor for their recommended minimum solder mask feature.

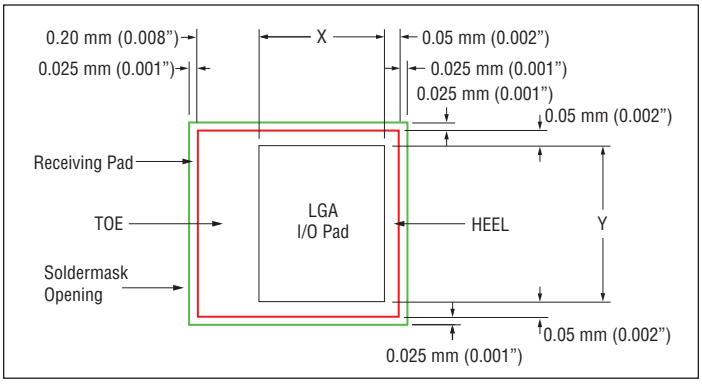


***Figure 2** – Details of the copper defined I/O receiving pads and solder-mask openings.*

[1] The terms heel, toe and side are typically used to describe the 4 sides of a metal lead that mates with a pcb substrate. The heel being the edge that is under the body of the component, the toe the side that extends beyond the body of the component and the sides the edges that connect the heel and toe.

The values for X and Y, shown in Figures 2, 3 and 4, are the LGA pad dimensions and can be found at the end of this guide or in the corresponding Picor LGA product data sheet.

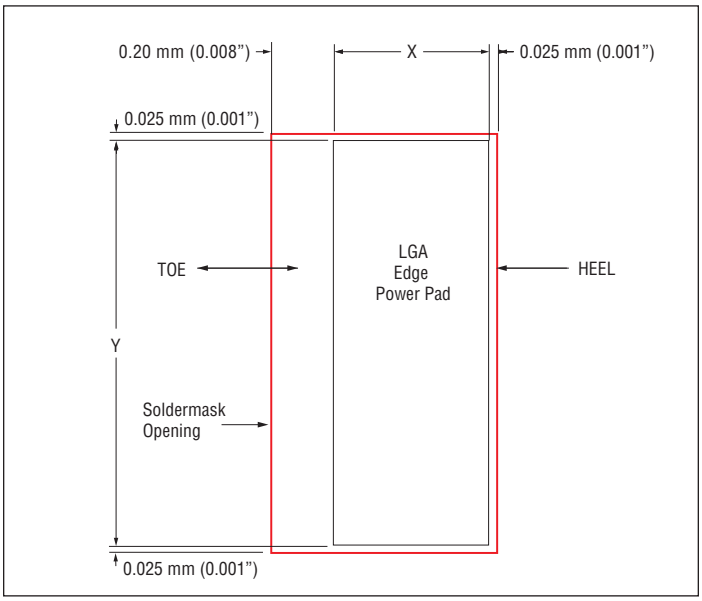


***Figure 3** – Solder-mask defined LGA edge power pads.*

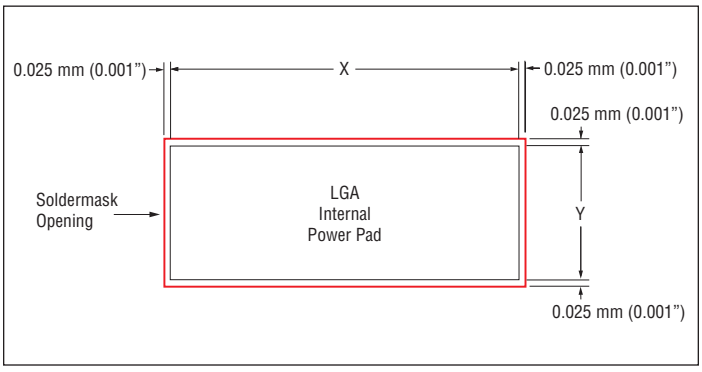


***Figure 4** – Solder-mask defined LGA internal power pads.*

Stencil Design

Picor recommends a 6mil thick stencil, with stencil openings of 80% of the receiving pad area. For solder mask defined openings, the receiving area is the same area as the solder mask opening.

Copper Pattern

When designing a board using Picor's LGA package, it is very important to make allowances for copper filled board area around the power pads. The power pads of the LGA should be connected directly to the polygon with solid connections on all sides, not with thermal relief pads. The greater the area of copper connected to the power pads of the LGA, the better the thermal performance of the LGA package. If there are internal layers available, then it is recommended that at a minimum, the copper area of the top layer be mirrored on inner layers with as many vias, on either side of the receiving pads, as can be reliably manufactured. The vias in Figure 5 are 10mil holes with a 5mil annular ring and should be tented top and bottom with solder mask.

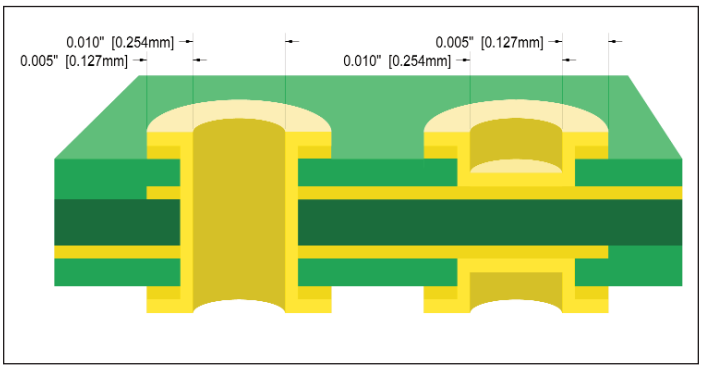


***Figure 5** – Recommended blind and through-hole Via Construction (dimensions shown are minimum values).*

The downside of large areas of copper is the amount of heat required to reflow the power pads of the LGA. If the reflow zone's temperature has to be greatly increased to reflow the LGA, then components mounted on smaller areas of copper could be thermally damaged. We recommend that routine reflow temperature profiling should be completed before assembling with a new PCB design to determine peak device temperatures during the SMT process.

IPC/JEDEC Reflow Guidelines

The 5 x 7 mm LGA package is designed to be re-flowed using lead-free solder, with a maximum reflow temperature of 260°C for 20 to 40 seconds. The package can be maintained at a temperature above 217°C for 60 to 150 seconds. The reflow profile should adhere to the JEDEC standard, taken from IPC/JEDEC J-STD-020C July, 2004, shown in Figures 6 and 7. Figure 7 is a lead-free reflow profile, but the 5 x 7 mm LGA package is compatible with leaded solders as well.

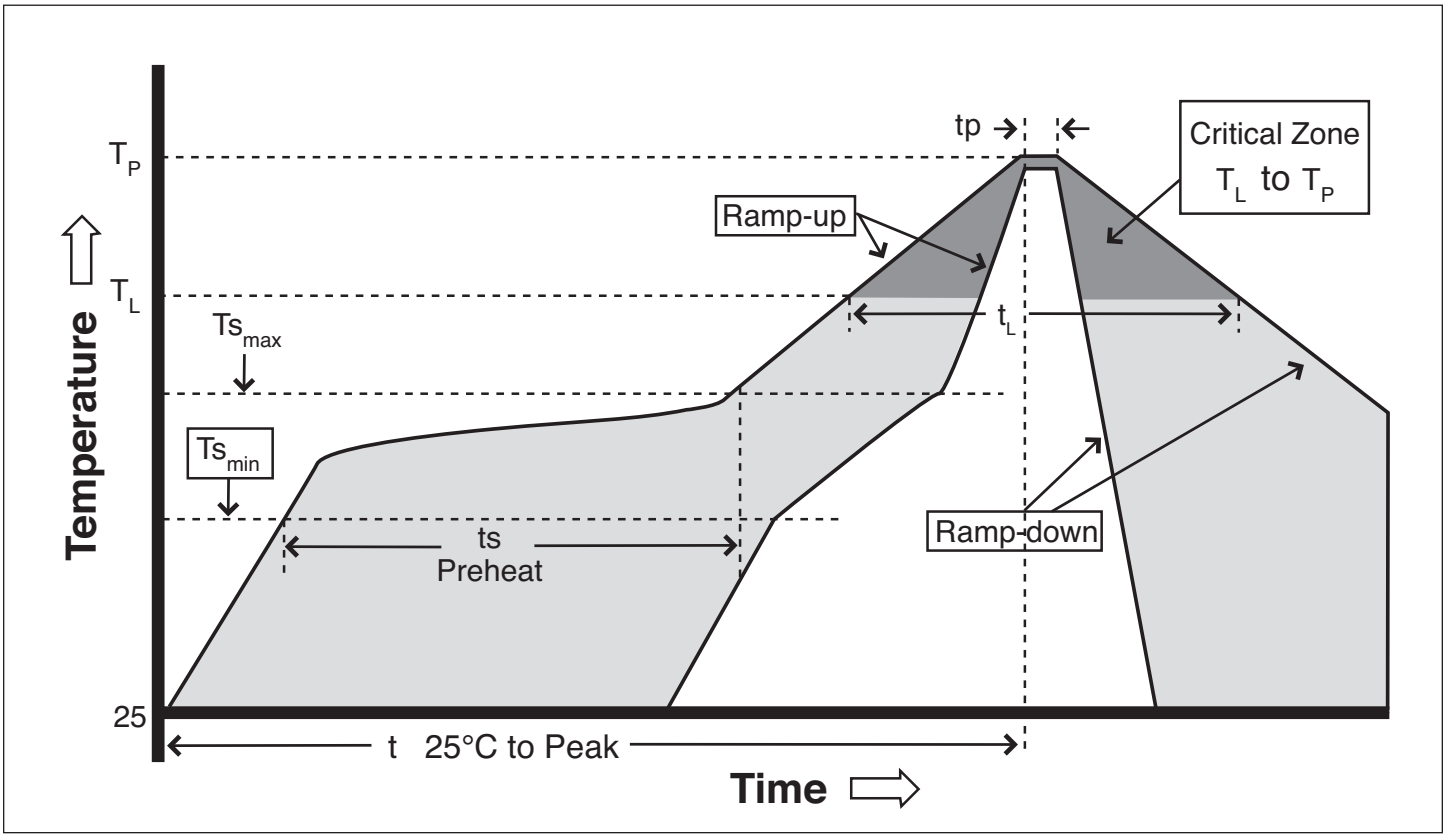


***Figure 6** — IPC/JEDEC Classification Reflow Profile.*

Profile Feature	Pb-Free Assembly
Average Ramp-Up Rate (TS_{MAX} to T_P)	3 °C / second max.
Preheat – Temperature Min (Ts_{MIN}) – Temperature Max (Ts_{MAX}) – Time (ts_{MIN} to ts_{MAX})	 150 °C 200 °C 60 – 180 seconds
Time maintained above – Temperature (T_L) – Time (t_L)	 217 °C 60 – 150 seconds
Peak/Classification Temperature (T_P)	260 °C +0/-5 °C
Time within 5 °C of actual Peak Temperature (t_P)	20 – 40 seconds
Ramp-Down Rate	6 °C / second max.
Time 25 °C to Peak Temperature	8 minutes max.

NOTE: All temperatures refer to topside of the package measured in the package body surface"

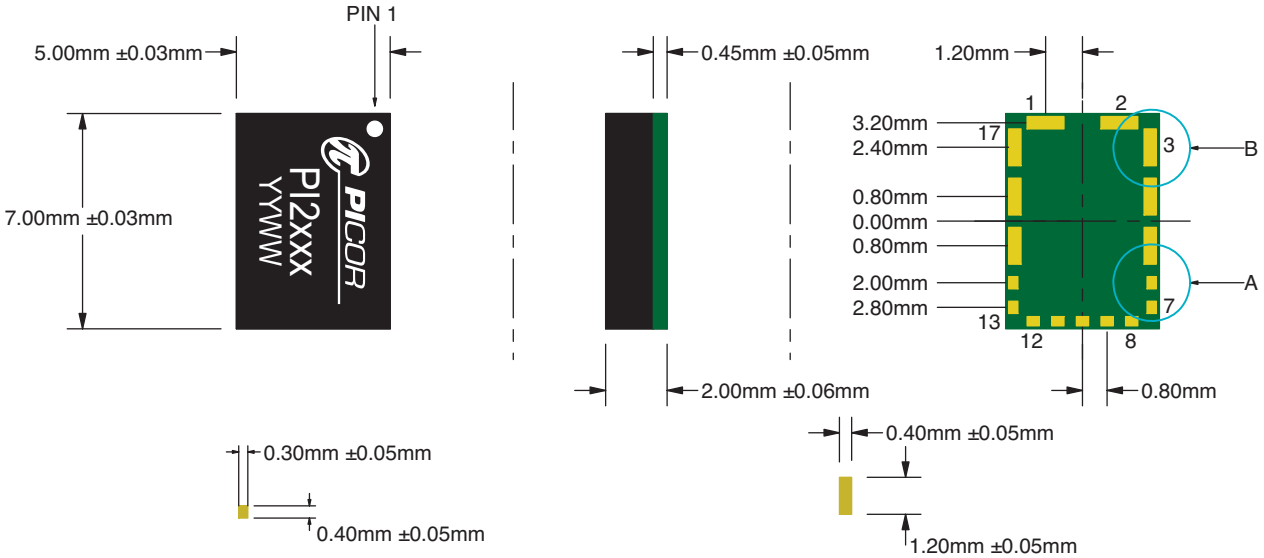
***Figure 7** — IPC/JEDEC Reflow Profiles table.*

LGA Rework Guidelines

The 5 x 7 mm LGA package is capable of being re-worked in a non-destructive manner using standard hot-air reworking stations. Care must be taken that the temperature the package is exposed to does not exceed the maximum reflow temperature of 260°C. Pre-heating of the board can be done to minimize the time the peak re-work temperature is applied. Also, solder flux should be applied to the area to insure proper solder reflow.

Picor 5 x 7 mm LGA Packages:

Picor's 5 x 7 mm LGA package is based on an industry standard 0.8mm pitch MLCC package.



A: I/O Pads (Pins 6 - 14)
Copper Defined

B: Power Pads (Pins 1 - 5, 15 - 17)
Soldermask Defined

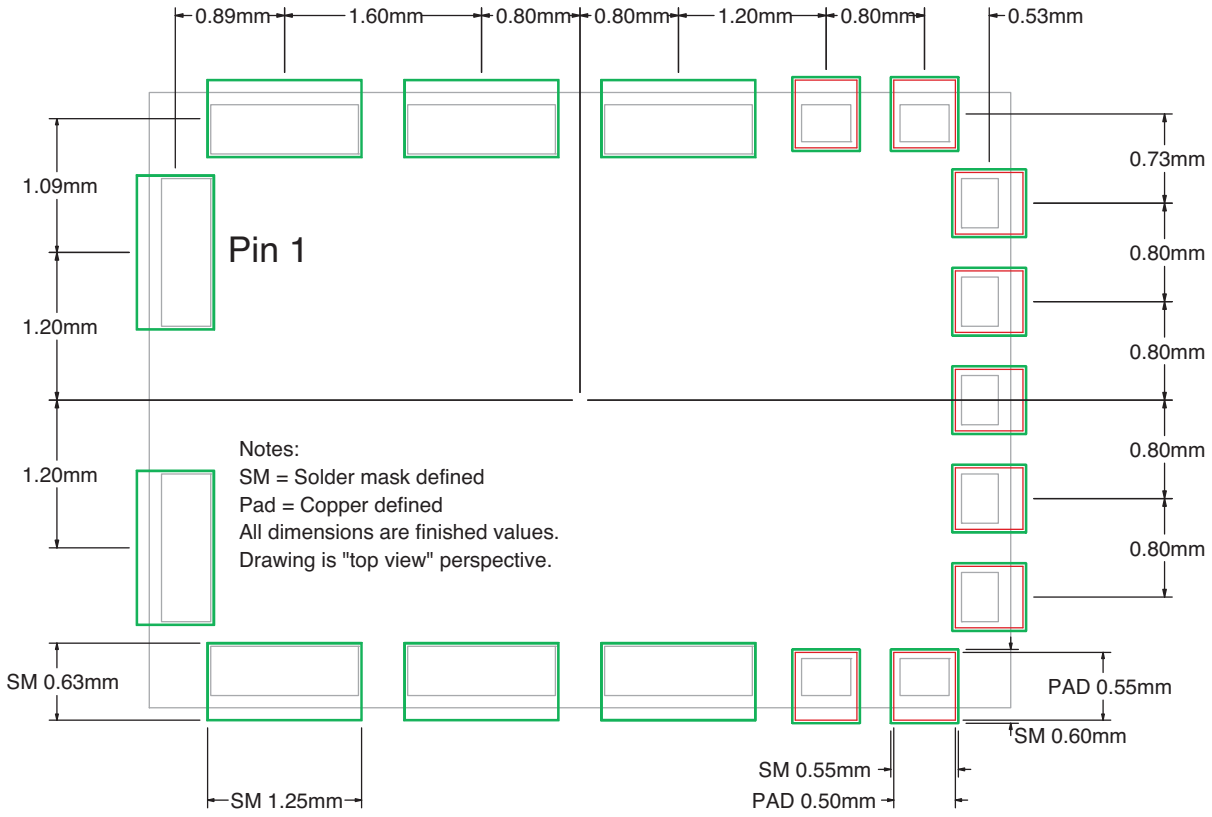
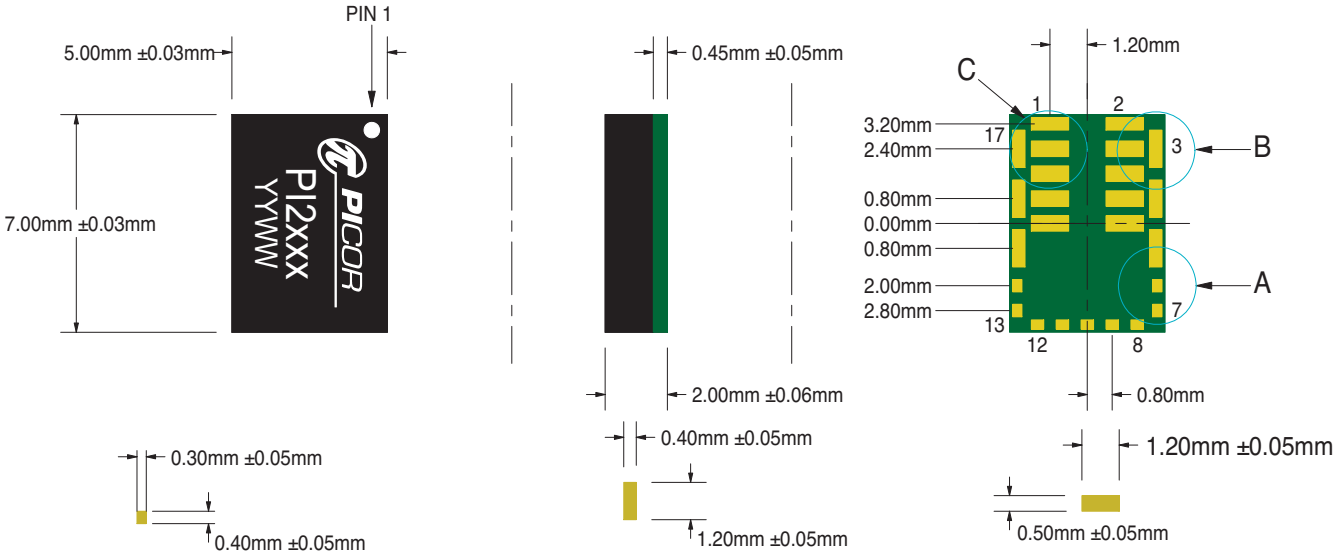


Figure 8 *— 5 x 7 mm, 17 Lead Package. (PI2121, PI2123, PI2125)*



A: I/O Pads (Pins 6 - 14)
Copper Defined

B: Power Pads (Pins 1 - 5, 15 - 17)
Soldermask Defined

C: Internal Power Pads
Soldermask Defined (8 Places)

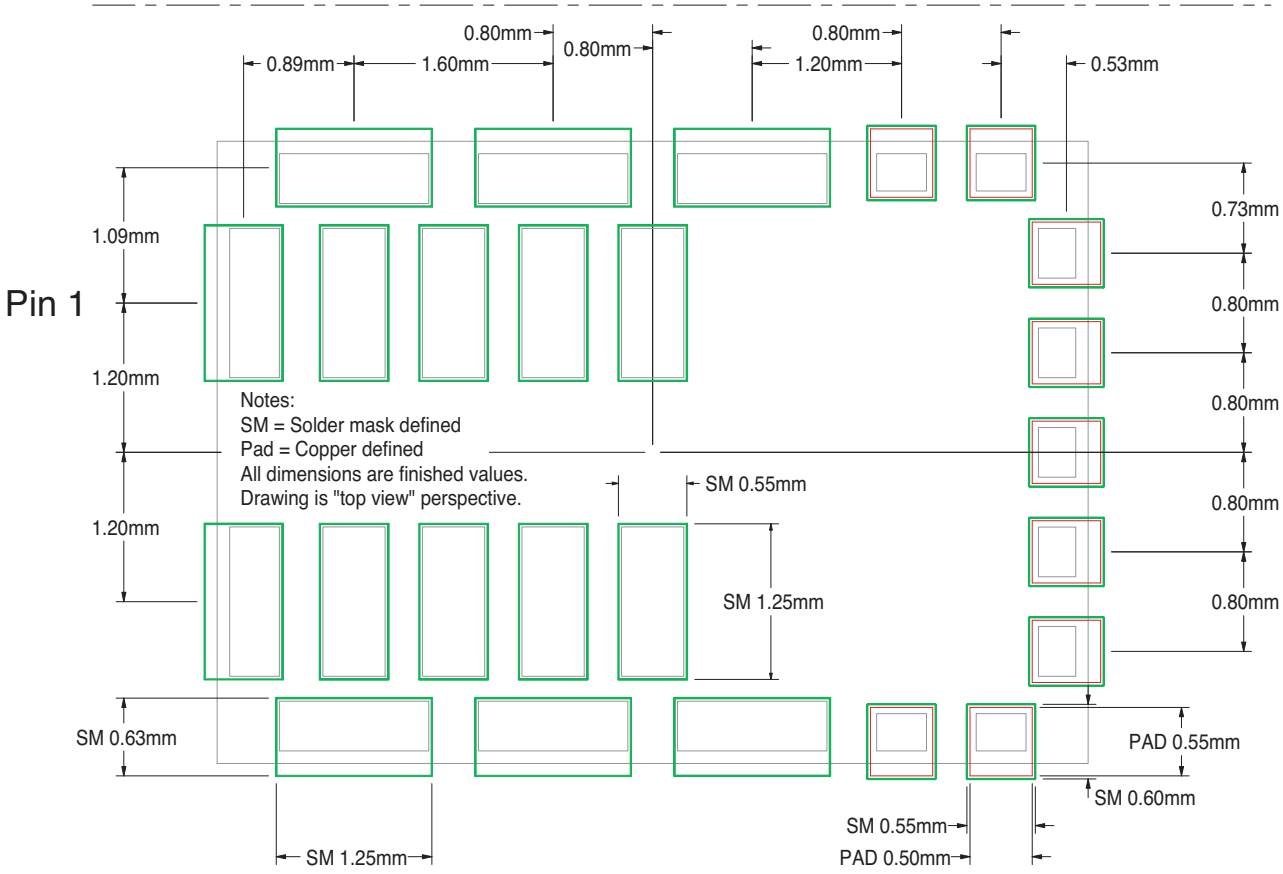


Figure 9 *— 5 x 7 mm, 25 Lead Package. (PI2122)*

Vicor's comprehensive line of power solutions includes high-density AC-DC & DC-DC modules and accessory components, fully configurable AC-DC & DC-DC power supplies, and complete custom power systems.

Specifications are subject to change without notice.