

## BURN-IN BOARD DESIGN GUIDELINES

### INTRODUCTION

Burn-in boards should be designed to provide many years of reliable performance. To achieve long-term, trouble-free performance, boards must be built to proper design requirements. Conductor pads must be aligned with the socket contacts and be of proper size and plated with 40-50  $\mu$ nches of gold to have adequate protection against the formation of nonconductive oxidation. Reliable burn-in performance is achieved only when sockets and boards work well together and are properly maintained. By adhering to guidelines for fabrication and maintenance, burn-in boards can achieve a good ROI. These guidelines are consistent with industry-accepted standards for routing, dimensioning and plating.

### ROUTING DESIGN RULES

Routing design rules are found in *Loranger Assembly Notes for BGA and LGA Sockets, December 2002*. It provides dimensional requirements for pads, vias, and traces. Of particular importance are the pads and traces on the top layer of the board, which interfaces with the socket and its contacts.

Conductor pads on the socket side of the board should be square and within dimensions given on the socket drawing. These conditions are necessary to achieve reliable connections between the contact and board (see *Figure 1*).

The square shape provides maximum surface area for contacts to make a reliable electrical connection. Circular pads with a diameter equal to the square edge ( $P_{top}$ ) will provide 21% less area than the square, making it more difficult for the contact in the socket to fall properly on the conductor pad.

Via diameter must not exceed the guideline values to ensure the contact does not drop into the via hole. Maximum via diameters range from 0.006in

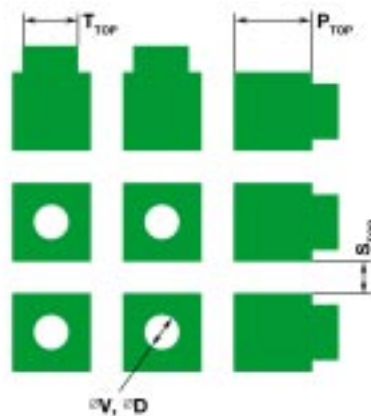


Fig 1 – Typical conductor pads on top layer of burn-in board. Note square shape. The labels are for dimensions provided in the referenced documentation.

for conductor pads on a 0.5-0.65mm pitch to 0.012in for conductor pads on a 1-1.27mm pitch.

Spacing between conductor pads ( $S_{top}$ ) must be large enough to prevent shorts between pads but small enough to maintain nominal pad dimensions. Spacing varies between 0.0057in for 0.5-0.65mm pitch to 0.0095in for 1-1.27mm pitch ranges.

Note that “tear-dropping” between traces and drilled pads on inner and bottom layers is called out in the referenced *Assembly Notes, March 2003*. Tear-dropping assures a connection between trace and pad if the drilled hole is off-center in the direction of the trace (see *Figure 2*).



Fig 2 – Examples of breakout and tangency in pads. These are round pads on the bottom side.

### FABRICATION OF BOARD

Guidelines for manufacturing burn-in boards are also provided in the *Assembly Notes*, which include tolerances,

plating and drilling recommendations for socket mounting pad features, solder mask clearances and guide pin hole locations. One set of critical features are the 4 drilled mounting holes to assemble the socket to the board. These non-plated through-holes are responsible for aligning the socket contacts over the conductor pads. The holes must be drilled 0.001in over the maximum thread size of the screw (specified on the socket drawing) and held to  $\pm 0.001$  in with a true position of 0.004in.

Solder mask free zone in an annulus around the plating surrounding the mounting hole is always specified on the drawing. This requirement assures socket standoffs will maintain a level gap, generally 0.015in between socket and board. In the referenced guidelines solder mask free polygon(s) are recommended over the entire footprint area on top and bottom sides of board. This recommendation allows conductor pads to remain free of nonconducting solder mask.

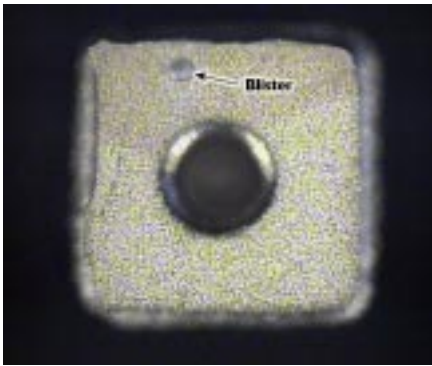
Many sockets have 2 plastic molded guide pins protruding from the bottom of the body which are captured by holes at outer corners of the footprint on the board. These holes must be precisely drilled into the board to within  $\pm 0.005$ in from the center of mounting hole to function correctly – helping align the socket over the conductor pads.

### CLAD PLATING

For reliable electrical connections between contacts and conductor pads, it is necessary to electroplate conductor pads with 40-50 $\mu$ n gold over 50 $\mu$ n nickel. The nickel prevents the oxidation of the base copper while the gold adds an excellent electrical connection and prevents nonconductive nickel oxide from forming. The material and coating thickness guidelines are provided in *Specification For Electrodeposited Gold Coating On Burn-In Board Conductor Pads And Contacts, September, 2000*.

The purity and hardness of gold plating specified is ASTM Type 2 with Knoop Hardness range 130-200 HK25. The 40-50µin gold has been found to be durable over the working life of the board. Thin gold coatings have more open porosity and present less of a barrier to oxide growth. *Figure 3* shows a blister on a conductor pad with 5µin gold over nickel after 1,000 hours at 125°C.

Burn-in boards will provide many years of service if sockets are assembled correctly and maintained properly. In *Printed Circuit Board Maintenance for Burn-In*, proper storage of boards, socket loading instructions and procedures for unloading and handling of burn-in boards are discussed.



*Fig 3 – Conductor pad with 5µin gold plating heated for 1,000 hrs at 125°C. Note blister on top left which is from Ni oxide pushing up against the gold coating.*

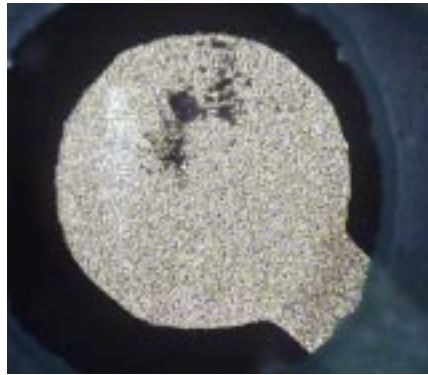
### BOARD MAINTENANCE

Loading of sockets onto boards requires mounting screws be torqued to correct values provided. Torque for the mounting screws ranges from 20-40 oz-in with the lesser for 0.5mm pitch and larger for 1.27mm pitch. Exceeding torque guidelines will cause the screws to rotate, loosening the nuts in the socket, which will lead to faulty operation of the socket. Too little torque will result in an uneven gap between socket and burn-in board. This discrepancy may result in contact forces dropping below the minimum 10 grams necessary for maintaining a reliable electrical connection.

Keeping boards clean is critical. Dirty conductor pads will reduce the area available for good electrical contact, decreasing contact reliability. Cleaning the board every 240 hours for burn-in times of 1 to 2 hours, and every 6 months for burn-in times longer than 2 hours is recommended. Cleaning boards in an ultrasonic bath is discussed in the referenced document above. See *Figure 4* for an

example of a dirty conductor pad.

A board out of use for more than 1 month should be cleaned.



*Fig 4 – Dirty conductor pad. Note pad is circular which is contrary to routing guidelines.*

### SUMMARY

A summary of considerations for the design of burn-in boards (and illustrated in *Figure 5*) are:

(a) Conductor pads must be square and within the dimensional tolerances called out on socket drawings.

(b) Conductor pads must have between 40-50µin electroplated gold over nickel to assure adequate protection from formation of nonconductive oxides.

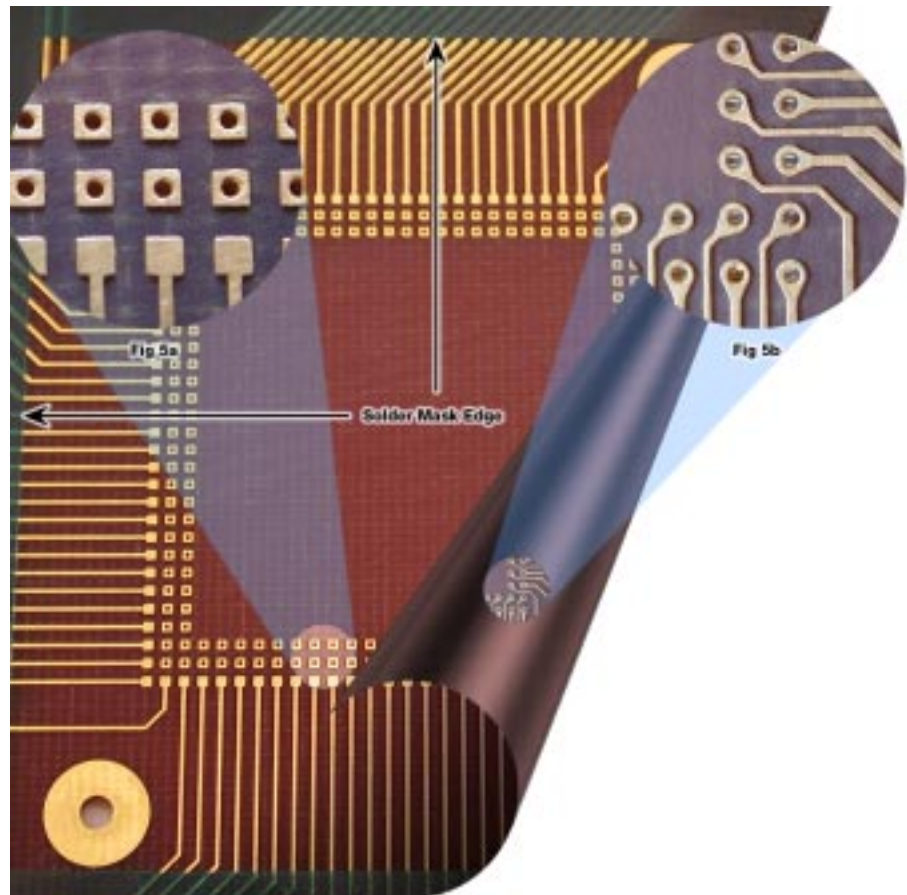
(c) Via diameters in conductor pads must be no larger than recommended in guidelines to assure optimum contact.

(d) Non-plated mounting holes for sockets must be drilled with location and diameter tolerances recommended on drawing, have level pads and solder-free zones in an annulus around the pad. If these conditions do not exist, misalignment between socket contacts and conductor pads will occur and contribute to less-than-optimum performance.

(e) Solder-mask-free zones must exist in an annulus around mounting hole pads, over footprint on top and bottom sides of board.

(f) Cleanliness of boards is critical to maintaining reliable performance of sockets.

(g) Mounting nuts should be torqued within the range of values specified on the drawing.



*Fig 5 – Typical footprint of correctly routed board. Fig 5a shows closeup of top side of board, while Fig 5b shows closeup of bottom side of board.*