



Gull Wing Surface mount Foot Soldering Instructions

The gull wing style, surface mount foot (parts with a 'SF-' prefix and a '-G' suffix) is designed to solder to a quad flat pack (QFP) surface mount land pattern. The emulator foot emulates the physical characteristics of a QFP gull wing package very closely, allowing the foot to be soldered to a target board land pattern using the methods commonly employed in attaching actual QFP packages. The recommended method is explained below with visual aids showing the step-by-step process. This method has produced very good results. Figure 1 shows the surface mount emulator foot and a clean target printed circuit board. The steps involved in the soldering process follow.

CAUTION: During secondary reflow (i.e. mounting the emulator foot to a board), the temperature profile should be tightly controlled. The gull wing leads are attached to

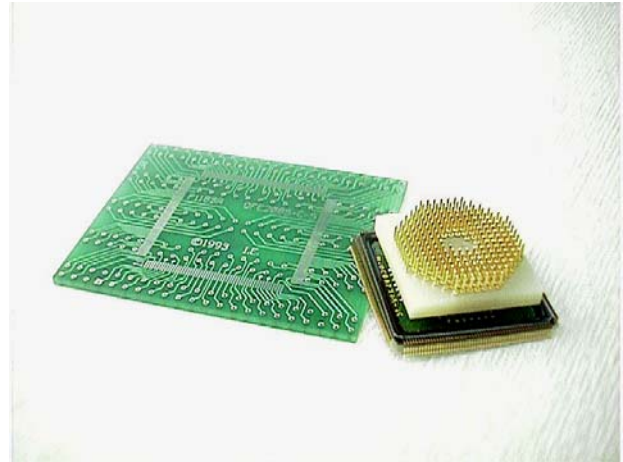


Figure 1: Target PCB/Emulator Foot

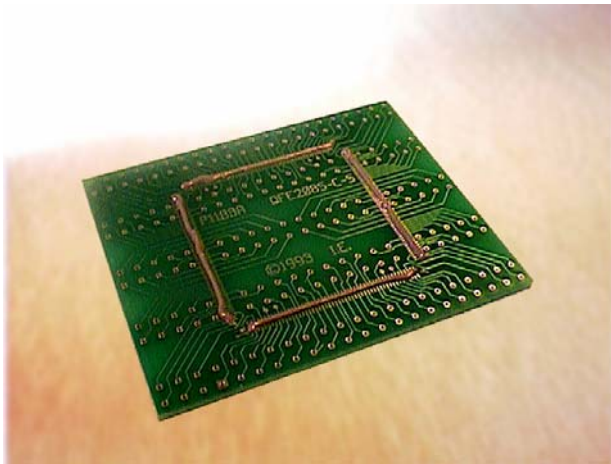


Figure 2: Apply Solder Paste

pad between the center of the pad and the outer edge. Begin with this amount and add additional paste after reflow, if necessary (excessive paste on an initial trial will be difficult to remove).

(c) Note the target PCB QFP land pattern and the emulator foot Pin 1 locations.

(d) Align and place the emulator foot onto the solder paste and land pattern as shown in Figure 3. 'Pick and place' equipment or a vacuum pen, are recommended (if they will accommodate the foot), but, handling the foot by the gold pins and placing on the land pattern by hand will suffice.

(e) Reflow target PCB with emulator foot in reflow oven (convection, IR, etc.). The recommended reflow profile is shown in the Figure 5. Time and temperature settings will be determined by the manufactures of the solder paste and

the emulator foot with 95/5 tin/silver solder (melting point 245°C). If the temperature during reflow exceeds 245°C, it will cause bridging between the leads or create opens at the clip head. The peak temperature during reflow should not reach more than 240°C. Peaks to 260°C are allowed only if the residence time is less than 40 seconds above 240°C. If you are attaching the emulator board to a target board using a hot soldering iron, this can easily happen. We strongly recommend that furnaces are profiled every day.

REFLOW OVEN - Soldering method #1

(a) Determine an appropriate temperature solder paste for your application.

(b) Apply a continuous bead of paste to the target PCB pads as shown in Figure 2. Cover approximately 1/3 of the

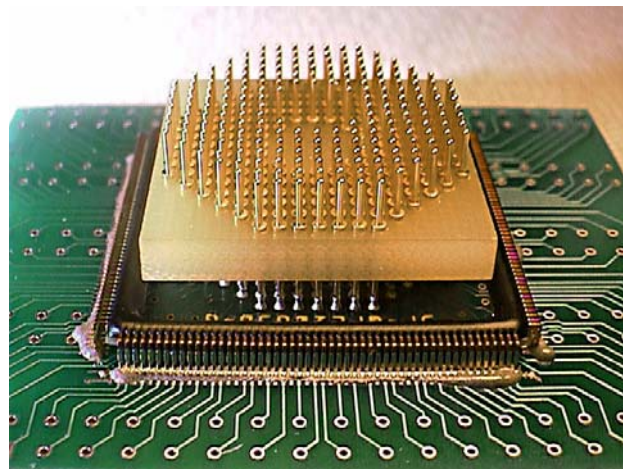
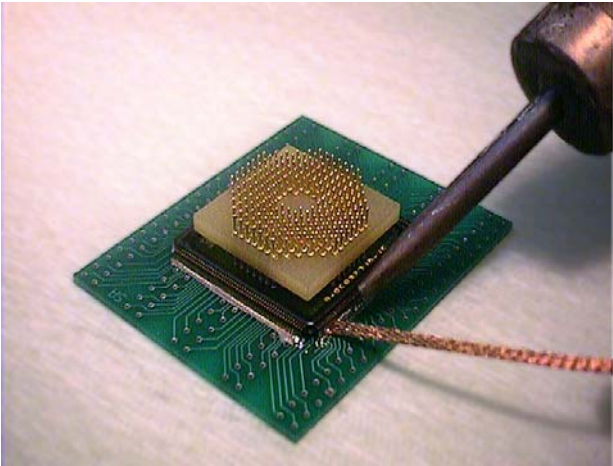


Figure 3: Align foot on the lands



Gull wing Surface mount Foot Soldering Instructions (continued)

reflow oven. The Ironwood gull wing foot has a larger thermal mass than an actual QFP package, and therefore,



may require longer reflow time and/or higher temperature settings.

(f) Inspect solder fillets. Add additional solder paste to solder deficient areas as needed or remove excess with small tip solder iron and copper desoldering braid (Figure 4). If the solder has not completely reflowed, add solder flux and repeat step (e).

Figure 4: Removing excess solder

Because of the construction of the emulator foot, a **Low Temperature** solder paste must be used. Set reflow equipment to the lowest setting that will reflow the solder paste. Reflow the emulator foot with target assembly in a temperature range of **185 - 210 degree C**. This can be varied depending upon the profile of the oven and the customer assemblies. The above temperature range is safe for soldering the emulator foot without thermal damage to

the foot. It is recommended that reflow oven be set for a minimum time initially. If reflow was not complete, reflow and reflow for a longer duration.

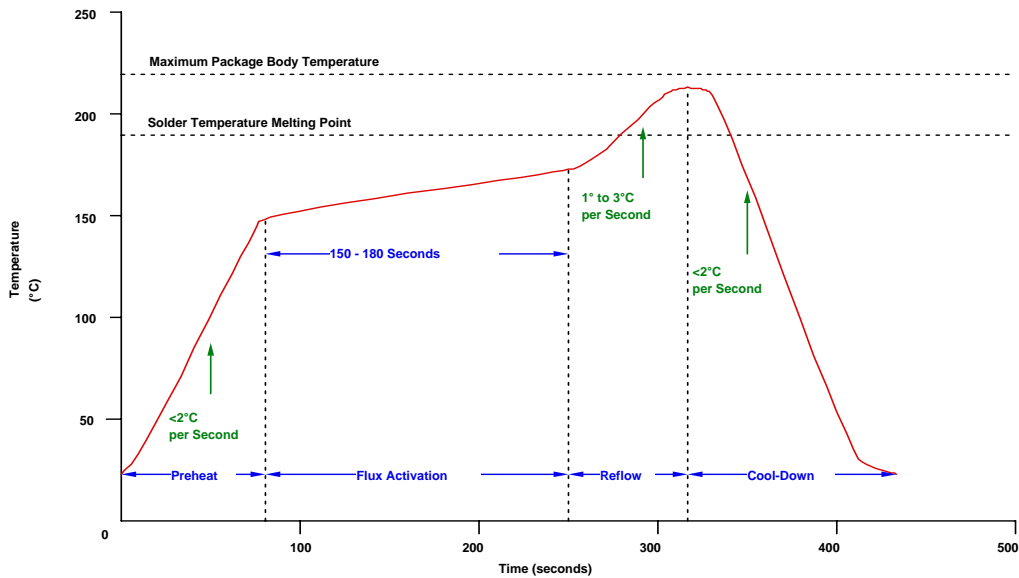


Figure 5: Recommended Convection Oven Reflow Profile

HOT AIR TOOL - Soldering method #2

Repeat steps (a) through (d) in method #1. The surface tension present between the solder and the emulator foot in method #1 will not be present in method #2, due to the fact that only a small portion of the solder in this method will be liquid at one time. It is necessary therefore, to align the foot over the land pattern with greater accuracy.



Gull wing Surface mount Foot Soldering Instructions (continued)

- (e) Reflow, with a hot air wand/gun, the solder over a few of the pads in opposite corners (diagonally) of the land pattern (Figure 6).
- (f) Check the foot alignment.
- (g) Continue by reflowing the remaining solder paste. Add or remove solder as needed (see step (f) in method 1.).

SOLDERING IRON - Soldering method #3

This method has produced very good results but may be more time consuming than the other two methods. Caution must be used when touching the soldering iron tip to the emulator foot. Excessive heat or pressure may damage the pads on the side of the foot.

- (a) Using a small diameter solder wire (approx. 0.015" or smaller) and a very fine tipped soldering iron, add enough solder to two opposite corner (diagonal) pads to cover them.
- (b) Align and place the emulator foot over the QFP land pattern (see steps (c) and (d) in method 1).
- (c) Holding the foot in place, by pressing down gently on the gold terminal pins, place the iron tip on the two pads to reflow the solder. This will tack and keep the foot in alignment.

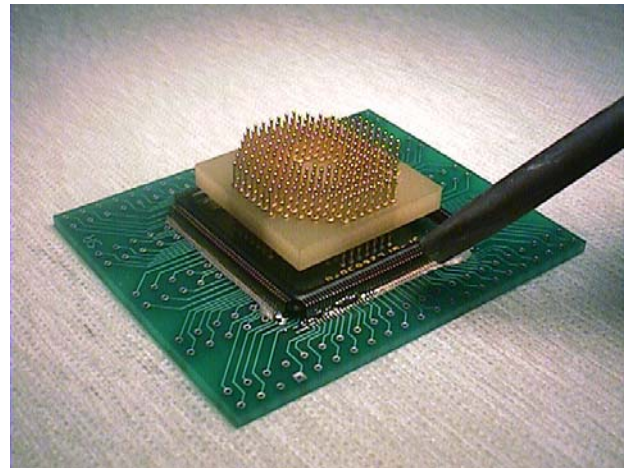


Figure 6: Reflow using hot air tool

- (d) Under a microscope or magnifying lens, if available, solder the remaining edge pads of the foot to the target PCB land pattern using a liberal amount of solder (shorts between adjacent pads can be removed later).

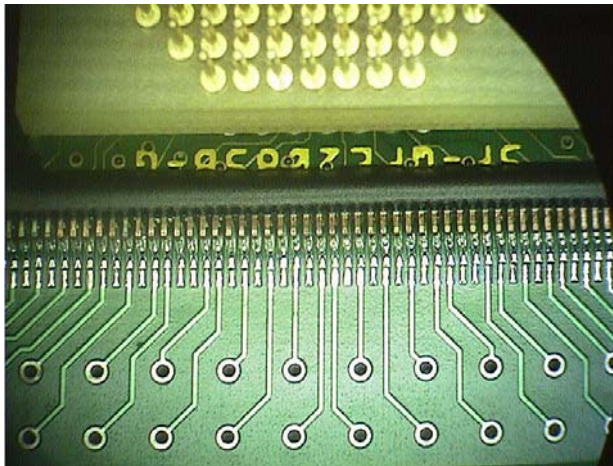


Figure 7: Finished Solder Fillets

Removing or Desoldering

Conventional methods can be used to remove a surface mount foot from your target board, however we recommend the use of PRB Line® D'SOLDER™.; This SMT device removal product avoids the use of excessive heat that can compromise the integrity of our product and your target board. The specially





Gull wing Surface mount Foot Soldering Instructions (continued)

formulated alloy and flux make desoldering quick and easy. This solution can be a time and money saver for many applications. (P/N TL-DS123)

3. filename: GSI.doc, Rev C” with footer autotext filename, Rev. C.Replaced pcb with PCB.