

## Leadless Surface Mount Foot Soldering Instructions

The Ironwood Electronics, leadless style, emulator foot (parts with a '*SF-*' prefix and a '*-L*' suffix) is designed to solder to the standard gull-wing type quad flat pack (QFP) surface mount land pattern. Because the Ironwood emulator foot does not emulate the physical characteristics of a QFP gull-wing package, the methods used to solder it to a target PCB are different. Several methods are possible, 3 are listed here in order of preference:

### **1. Solder Paste and Reflow Oven**

- (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 Fig. 1. Cover approximately 1/3 of the 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 Fig. 2. '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.). Time and temperature settings will be determined by the manufactures of the solder paste and reflow oven. The Ironwood leadless foot has a larger thermal mass than a gull-wing QFP package, and therefore, may require longer reflow times or higher temperature settings. Surface tension between the liquid reflowing solder and the emulator foot will center the emulator foot on the land pattern.
- (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. If the solder has not completely reflowed, add solder flux and repeat step (e). It may then, be necessary to increase the reflow temperature and/or time.

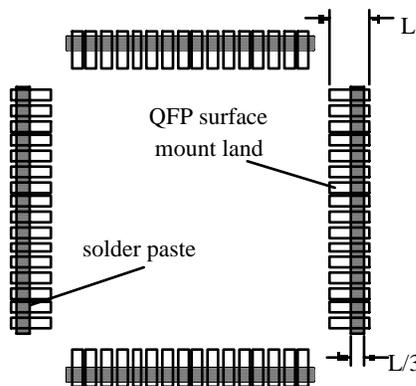


Fig. 1 Top view of land pattern with paste

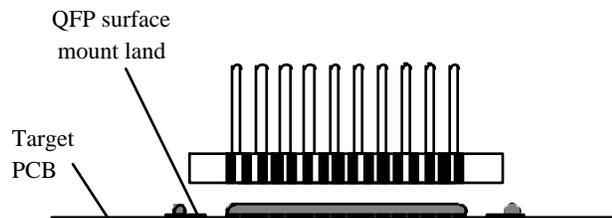


Fig. 2 Side view of emulator foot oriented over land pattern

## Leadless Surface Mount Foot Soldering Instructions (continued)

### **2. Solder Paste and Hot Air Wand**

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.

(e) Reflow, with a hot air wand/gun, the solder over a few of the pads in opposite corners (diagonally) of the land pattern.

(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.).

### **3. Soldering Iron**

•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.

(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 (do not worry about shorts between adjacent pads).

(e) Apply a generous amount of flux along the side of the foot.

(f) Start at one corner and pull the tip of the iron along the side of the foot to remove excess solder deposits. Clean the tip of the iron often. Repeat this step several times starting at a point on the foot ahead of the excess solder. Continue down the side of the foot until shorts are removed and a fillet is present between all foot pads and target PCB pads.

(g) Repeat steps (e) and (f) for the remaining three sides.

### **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 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)



Rev. C, 6/1/0, PJ, No Rev.

1. Changed document title from Leadless QFP Emulator Foot Soldering Instructions.
2. filename: LSI.doc, Rev C" with footer autotext filename, Rev. C.