PCB Design Guidelines

Completed by: *Tommy Ewing*	Date: 3/16/2007	Approved By: *Stanley Baker, Tom Post*	Date: 3/16/2007

Our fully automated Fuji surface mount machines require specific standards for PCB panelization. Suppliers should meet the standards below when quoting and must provide mechanical documentation to confirm these requirements upon ARO. For non-standard applications, specific PCB requirements should be determined on an individual basis.

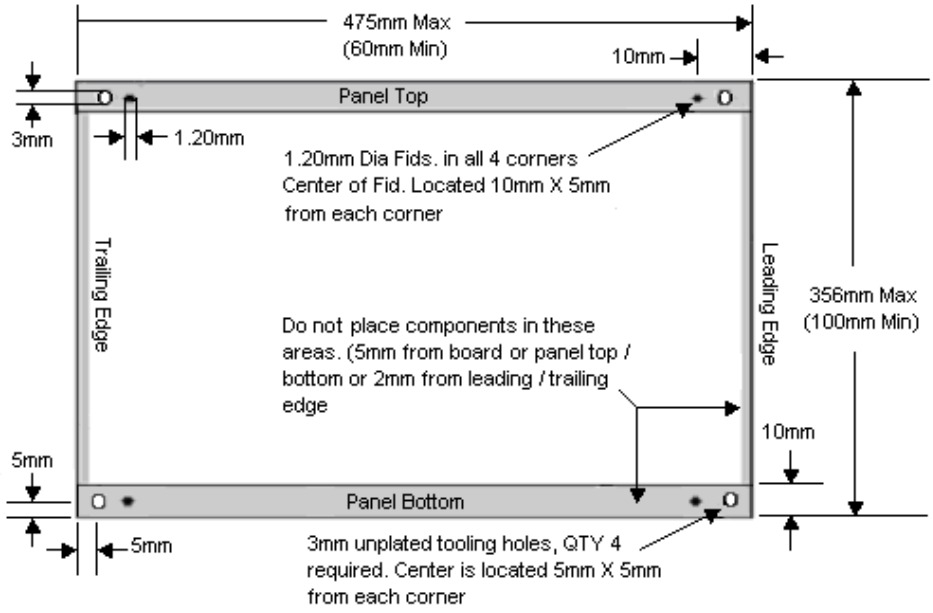
Fuji SMT Line and Wave Solder Requirements

- Maximum size: 356mm X 457mm
- Minimum size: 60mm x 100mm
- Maximum component height: 30mm

For panelization on single-up or multi-up panels:

- **QTY 4 Tooling Pin Holes - 3mm diameter**, non-plated, shall be placed on all 4 corners of rails. The CENTER of the tooling holes shall be located 5mm x 5mm from the corners as show below.
- **QTY 4 Global Fiducials - 1.20mm diameter** shall be placed on all 4 corners of rails. CENTER of Fiducials shall be 10mm X 5mm from the corners as show below.

  NOTE: If SMT parts are located on top and bottom, fiducials need to be placed on both sides accordingly.
- **Panel Top and Bottom Rails - 10mm** when multi-up or as required.



PCB Design Guidelines

Fiducials:
These should be included in 3 main categories:

A. Local Fiducials – For each fine pitch device (25mil lead pitch or less), fiducial marks should be located near the component on all 4 corners of the IC or diagonal at minimum.

B. Global PCB Fiducials – For each single PCB, fiducial marks should be included on all 4 corners or diagonal at minimum and spaced to encompass the entire placement area on the PCB.

C. Global Panel Fiducials – If the PCBs are panelized in an array form, fiducial marks shall be included on the panel rails on all 4 corners (both sides if SMT parts are located on top and bottom) and spaced to encompass the entire placement area on the panel. The fiducial mark should be round, and 1.20mm in diameter, and should have clearance of at least 50mil around from any traces, silk, stamps, mask or anything that can obscure the fiducial image.

Silkscreen Layer:
Silk legends should not cover any portion of the circuit where solder is to be applied.

Solder Paste Stencil:
For panels with surface mount parts, the PCB fabricator must generate the artwork for the solder paste stencil (a.k.a. step and repeat) from the Gerber data. To ensure compatibility with Sibex assembly process, the stencil specifications should be determined on an individual basis after PCB or panel review. If the CAD package generates the artwork, the Solder paste stencil apertures should be made 1:1 with the component pads.

Panelization:
Panelization is required for all SMT boards and smaller boards (whether through-hole or SMT) to increase processing efficiencies. There are some considerations when deciding to panelize or not. From an assembler's perspective, panels are desirable and sometimes necessary to provide the tooling holes and PCB edge clearance requirements. The PCB manufacturer's material utilization and cost of an electrical test fixture should also be considered, which is often the main priority.

It is recommended that panelization of PCB's should be determined by joint review between the PCB vendor and Sibex. This will ensure maximum material usage and process compatibility. The following are some considerations for circuits intended for panelization.

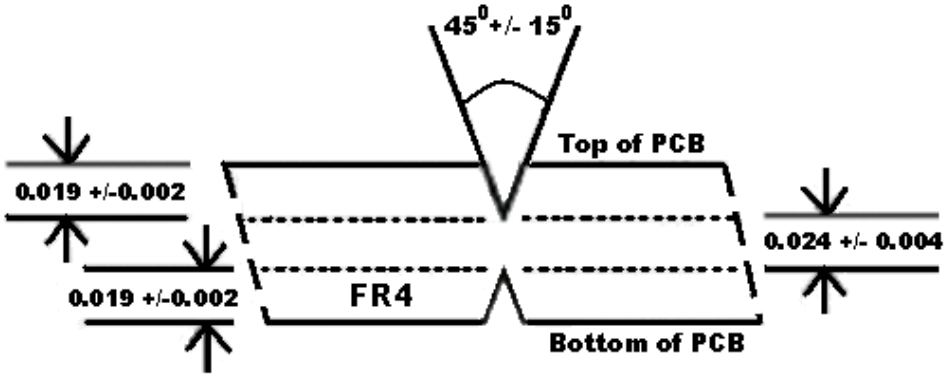
- If components overhang the edge of the PCB, adequate space between circuits to allow all components to be installed on the panelized PCBs is necessary. The layout may allow for component clearance on the adjacent PCB so that material usage is not sacrificed.

- Panels can be either scored or routed or a combination of each depending on the application.
  For routed PCB's: The breakaway tabs should be drilled for inside breakaway. Copper Trace or plane clearance at the location of the breakaway tabs should be at least 0.10" from the PCB edge.

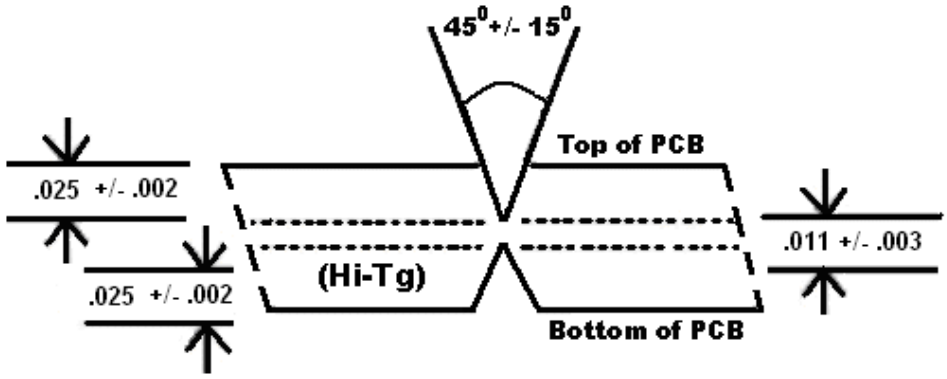
PCB Design Guidelines

For V-Scored PCB's: Unless otherwise reviewed and approved between Vendor and Sibex, V-scoring shall adhere to detail "A' for .062" FR-4 PCB material and detail "B" for .062 Hi Tg PCB material.

Detail "A": Typical V-Groove for .062" FR-4 PCB Material



Detail "B": Typical V-Groove for .062" Hi Tg PCB Material



- Component weight and panel size should be considered to reduce the possibility of warpage and/or breakage. Panels that are wave soldered (i.e. Through-Hole Components) will tend to sag in the center when subjected to preheat and reflow temperatures.

Sibex Engineering

727-726-4343