



Tips on Improving Manufacturability and Minimizing Your Costs

October, 2015

Page 1

INTRODUCTION

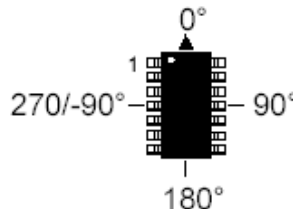
This Application Note will help you make your design more manufacturable. It will help you define the elements of your project, as well as provide Wavelength Electronics with all of the necessary information to give you a cost-effective alternative to doing your own manufacturing.

PLEASE PROVIDE THE FOLLOWING DOCUMENTATION:

1. Name of your project
2. Pick and Place file from PCB software that includes
 - X & Y centroid data: [0,0 should be near the board, not several inches away]

	A	B	C	D	E	F
1	X Center	Y Center	Rotation	Part Number	Reference Designator	Layer
2	526	319	90	70-85-6	R3	T

- Rotation of the part relative to 0 of our machine



- Your part number
 - Reference designator
 - Layer (if top & bottom are in one file)
3. Bill of Materials of parts to be loaded that includes:
 - Part Number
 - Number of parts used on assembly
 - Associated Reference Designators
 - Manufacturer part number
 - Part descriptions
 - A list of reference designators of no-load parts (this is particularly helpful)
 4. Component locator with polarity & reference designator markings. A “polarity key” to identify non-industry standard markings is helpful from first-time customers.
 5. Gerber file for production of stencil screens and boards.
 6. Specify leaded or unleaded processing.
 7. Boards will be washed. Please specify any parts that cannot be submersed in water.
 8. Our inspection standard is IPC Class 2. If you would like us to work to IPC Class 3, please let us know.

IF YOU PROVIDE PARTS, PLEASE CONSIDER THE FOLLOWING:

- During machine loading, a few parts will be lost. Please provide an excess of SMT parts (about 2" of tape or 5%) to ensure a complete build.
- For plate-through parts, please provide an extra or two.
- Seal the bag of raw circuit boards so contamination is minimized (fingerprints on pads, etc.).
- Mark the part bag with your part number. We easily accommodate:
 - » Reels (7"-preferred or 13")
 - » Cut tape (ideally with an extra 20 parts or so)
 - » Tubes
- A spreadsheet of our standard parts is available. Note our part number on your Bill of Materials if you want us to provide the part.

DESIGN FOR SURFACE MOUNT (SMT) MANUFACTURABILITY:

- To take advantage of the automated equipment, small boards should be panelized. The smallest single board we can reliably process is about 1.5" square.
- The smallest pasted dimension of the board or panel should not exceed 15". The maximum board or panel dimensions can be 17" x 21."
- For fine pitch ICs, to avoid tombstoning, misalignment, etc., align ICs such that one side of legs are heated at the same time.
- For small parts (0603 and smaller) parallel the pads so both pads are heated at the same time.
- Use reels of 8, 12, or 16 mm width to match standard tooling.
- Add three fiducials to each circuit board in the panel. Industry standard 40mil circle is preferred. Do not include the fiducials in the stencil file as solder paste will reduce contrast and make it hard to find the fiducial.
- Ideally, we prefer a 0.5" border around all 4 sides of the panel with four 1/8" diameter guide holes, 1/4" in from each corner, along with 3 fiducials on the border.
- For BGAs or other fine pitch components, please put two or three local fiducials around the part.
- If your board contains bottom termination components (such as BGAs, LGAs, QFNs) it must be set up to run through the automated process.
- Ideally, parts should be 0402 or larger or have minimum 0.4mm pitch (space between SMT pads). We can place smaller, but it adds cost.

DESIGN FOR PLATE-THROUGH MANUFACTURABILITY (AUTOMATE WITH SELECTIVE SOLDER MACHINE):

- Keep SMT parts away from plate-through holes on the solder side. Clearance from outside edge of plate-through hole pad to the outside edge of the surface mount pad should be a minimum of 100 mils (0.100").
- When panelizing, leave space between boards if the connectors or other components overhang the edge. If the parts from one board overlap parts on the next board on the panel, the overlapping part may need to be hand soldered, increasing costs.

DESIGN FOR OPTIMAL YIELD:

- If we do not conduct electrical testing on these assemblies, please consider that visual inspection cannot catch all potential electronic defects. Bad parts, cold solder joints, and other issues can only be found during electrical test. If you would like us to include electrical testing, so all boards can be guaranteed free of error, please let us know.
- If you test, give us feedback. Report any problems ASAP so repairs and root cause analysis can be completed promptly. We expect high yield to mean that 95% of the boards we build pass without any rework and the other 5% pass with minimal rework.
- We want to be considered a valuable part of your team and a competitive advantage to your business. Communicate your goals, timelines, and expectations to us so we can meet them.

CONTACT INFORMATION:

For a quotation or other information, such as any processes not mentioned above, please contact our EMS Department.

Wavelength Electronics
P O Box 865
Bozeman, MT 59771

Phone: 406-587-4910
Fax: 406-587-4911

<http://www.teamwavelength.com/services/index.php>

KEYWORDS

Electronic Manufacturing Services, EMS, custom design, design for manufacturability, DFM, IPC Class 2, BGA, SMT

