DFM GUIDELINES - WORK INSTRUCTIONS

1. General

This document outlines information and design parameters required by MSI to ensure a smooth transition of new products to the assembly process. Following the DFM (Design for Manufacturability) guidelines will ensure that the product can be manufactured with the highest level of quality and lowest assembly cost possible. While it is requested that the customers' designs conform to the guidelines set out below, MSI can in many cases accommodate certain deviations. These should be reviewed with MSI's Design and Production staff on an individual basis.

2. References

- IPC-7351 – Generic Requirements for Surface Mount Land Pattern and Design Standard
- IPC-A-610 – Acceptability of Electronic Assemblies

3. Customer-Supplied Information

3.1 Process Materials;

The customer should specify the type of materials used to manufacture their assemblies:

3.1.1 Solder -

Leaded (solder used is combination of tin and lead)

OR

Lead-Free (solder does not contain lead and is RoHS compliant)

3.1.2 Flux -

No-Clean (flux used in the soldering process is neutral {ph balance} and the assemblies are not washed)

OR

Water Soluble (flux used is slightly acidic and the boards are processed through a DI water wash system after assembly)

3.1.3 Chemicals -

The customer should specify any adhesives, conformal coatings or other chemicals required for the assembly of their products. MSI can assist in the selection of suitable materials for the application. Please consult with MSI Production Supervisor and Design staff.

3.2 Bill of Material;

A Bill of Material (BOM) is required in soft copy Excel or text format. The BOM should contain the following minimum information:

Part #	Manufacturer	Manufacturer P/N	Description	Location
123-456-789	ABC Inc.	LO987654321	Res 0805 1/10w 5% 100k	R1

3.3 Gerber Data;

"Gerber Data" refers to the photoplot files created from CAD PCB layout packages. One file is created for each layer of the PCB, plus a file each for silkscreen, soldermask, drill files and apertures. Gerber files are primarily used for the following purposes:

- Manufacturing the bare PCB's
- Creating stencils, wave solder pallets and other tooling
- Providing visual views of each layer for test and inspection purposes

Gerber data should be provided as soft copy files.

3.4 Placement Data;

"Placement Data" is a loose term that refers to PCB data that specifies part shape, location, rotation, reference and other data. This data is used to program the machines and create visual aids. There are three formats this data can be supplied in (listed in order of preference):

1) CAD Data - This data is a direct output from a board layout software package, such as PADS or PROTEL. It contains information such as part location, part rotation, reference location, part type, and part pin outs (VCC, GND, etc.)
2) CENTROID Data - This data is the "x, y, theta" output. It contains part x and y locations, as well as part rotation, and reference location.
3) GERBER Data – This is the same photoplot data as discussed above. It can also be used for machine programming, but is not recommended. The data provided within the file is minimal and is very prone to human error and miscalculation during programming.

All forms of data should be supplied in soft copy. They should be in ASCII format (text file format), not binary format (if you can open and read the file in a text editor such as Notepad, it is in ASCII format. Binary will appear as non-readable characters in Notepad).

3.5 Documentation;

In addition to the essential documentation listed in sections 3.2 to 3.4, the following documents will help to ensure a successful manufacturing transition:

- Assembly drawings and/or specifications
- Work Instructions
- Panelization drawing
- Schematic (for assemblies requiring test)

3.6 Special Requirements;

The customer should specify if there are any special or unique assembly techniques required to process their product. It is suggested that written assembly instructions be provided to detail such techniques.

3.7 Applicable Workmanship Standard;

The customer should specify whether their assemblies are subject to IPC-A-610 Class 1, 2, or 3 requirements, or other workmanship standards. Unless otherwise specified, MSI will manufacture to Class 2 requirements.

3.8 Test Requirements;

The customer should specify the level of testing (if any) required on their products. All processes and details are to be discussed and managed through the Test Manager.

3.9 Sample Assembly;

If available, a mechanical sample of the completed product should be provided as a sample assembly. This is invaluable in answering assembly questions and issues. A ‘golden sample’ (working and tested sample provided by the customer) should be supplied as a reference for tested products.

3.10 Consigned Kits;

- Loose components can not be placed by machine
- Discrete components and smaller IC’s should be supplied on tape and reel. Larger IC’s should be supplied either on tape and reel or on trays
- Discrete components must be supplied in minimum quantities of 100 pieces, as one continuous strip
- Discrete components require 6” leader of empty tape
- A 3% scrap factor is required for all discrete components (i.e. resistors, capacitors, diodes, etc.)

4. General PCB Considerations

4.1 Detailed PCB Specification;

The PCB design parameters with the greatest impact to manufacturability have been included below.

4.2 PC Board or Panel Size;

The maximum and minimum dimensions of a single board or panel are dependent on the equipment used in the manufacturing process:

Process	Maximum (Width x Length)	Minimum (Width x Length)
SMT	15” x 18”	3” x 5”
Thru Hole	18” x 18”	3” x 5”
Wave Solder	18” x 18”	3” x 5”
AOI	13.5” x 17”	3” x 5”

Oversize or undersize boards / panels may be possible to run. Please consult the MSI Production Supervisor.

4.3 PC Board Thickness;

Standard board thickness is 0.062”. Boards thicker than 0.062” (i.e. 0.093”, 0.124”) with PTH may require parts with longer leads in order to allow lead protrusion through the PCB. Boards that are 0.093” thick or greater may also require special tools or processes. Boards thinner than 0.062” should be panelized in such a way that board flex in minimized. Fixtures (carriers) will likely be required to provide adequate support on the conveyor line.

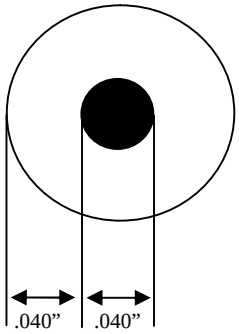
4.4 Fiducial Marks;

Refer to the IPC-7351 - Generic Requirements for Surface Mount Design and Land Pattern Standard, Section 3.4.4 for a full explanation of fiducials as well as specifications for their design. Fiducials are marks on the PCB used by vision systems of assembly equipment to accurately determine the position of the board (land patterns) at each step of the assembly process. Fiducials increase the accuracy of screen printing, component placement, and automated inspection by accounting for any offsets (x, y or theta) or nonlinear distortion of the PCB.

A minimum of three global fiducials are required on the panel. They should be located in a triangular position as far apart as possible. Global fiducials cannot be placed in locations that will be obscured (underneath a part). Global fiducials must be clear of the board edge by 0.250”. Global fiducials must be located on both top and bottom sides of the PCB if there are SMT components on both sides.

A minimum of two local fiducials is required for each fine pitch component of 20 mil pitch or less. These should be located diagonally opposite each other within the perimeter of the land pattern and not under the component. The minimum diameter of the fiducial mark is 0.040” (1.0mm) with equal radius solder mask clearance around the mark. A round plated circle is the best shape for a fiducial mark (please see IPC-7351). Other shapes are possible to use (please consult the MSI Production Supervisor).

Minimum Clearance



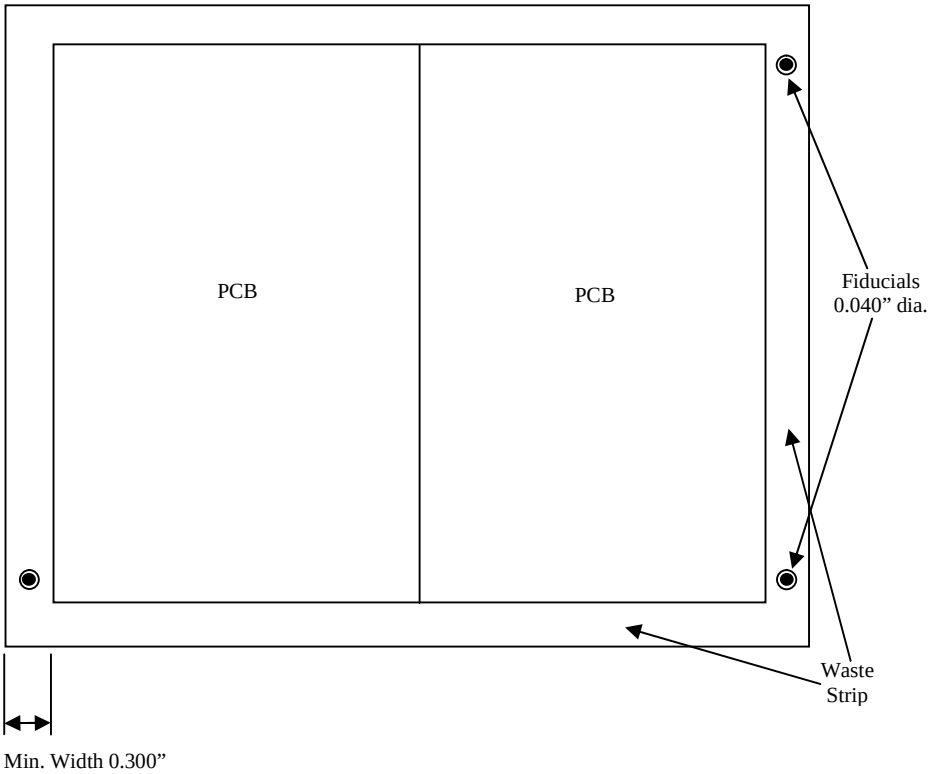
Fiducial Design

4.5 Panelization;

In general, panelizing smaller boards reduces assembly cost by increasing machine efficiency. Target panel size is 10” x 12” for boards that are 0.062” and thicker. Target panel size for boards thinner than 0.062” is 5” x 8”. Panelization (or a set of customer board carrying fixtures) is required for automated assembly if any of the following conditions apply:

- The board is not square or rectangular
- The board has notches along the edges which would interfere with sensors or the conveyors
- The board is smaller than 3” x 5”
- The board has components within 0.125” of the board edge along the two longest sides

If panelization is required, it is preferred to have 0.300” waste strips on all 4 sides of the panel. The panelization drawing should be provided to MSI Production and Design personnel for review prior to ordering the PCBs. Alternately, MSI can create a panel that will optimize both machine efficiency and the PCB supplier’s material usage in order to achieve the lowest overall product cost.



4.6 Depanelization Methods;

The panel can be designed with either 'V'-grooves or routed breakaway tabs between PCB images and between the PCB and the waste strips. 'V'-groove is the preferred method for depanelization as it is more efficient and less stressful to the product. 'V'-grooves should not be located closer than 0.200" from each other. 'V'-grooves are not recommended underneath parts which overhang the board edge. Breakaway tabs should use a routed path width of 0.100" to allow for easy depanelization using the nibble tool.

4.7 Board Edge Clearance;

It is recommended that all PCB's / Panels have a waste strip (min. width of 0.300") around the entire outer edge (see above figure). The design of the waste strip must be reviewed by the Production Supervisor to ensure that:

- All edge connectors that hang over and below the edge of the PCB are accounted for
- Process fixtures / pallets can be ordered and/or modified to fit the proposed panel layout

All components should be a minimum of 0.100" from the edge of the PCB to reduce the chance that the solder joints may be cracked during depanelization. All traces, holes, etc. should be a minimum of 0.040" from the edge of the PCB to allow for proper depaneling. If waste strips are not used, the minimum clearance for parts from the edge of the PCB is 0.125".

4.8 Via Guidelines;

Refer to IPC-7351 (section 3.4.6) for guidelines on 'via' design. 'Vias' should be designed in such a way to prevent solder migration from the component land down the 'via', which would result in the component having insufficient solder. All 'vias' under BGA components are required to be covered or plugged with solder mask.

4.9 PCB Finish;

Electroless nickel immersion gold (ENIG) is the preferred board finish and the only finish currently approved for MSI's lead-free assembly process. Contact Design personnel of Production Supervisor if an alternate finish is desired.

5. SMT Considerations

5.1 Land Pattern Design;

Refer to IPC-7351 for recommended land pattern designs. The component manufacturers typically also provide recommended land patterns for their devices.

5.2 Density Level;

For determining the land patterns and courtyard boundaries for SMT components, IPC has defined 3 density levels:

- Density Level A: Maximum (Most) Land Protrusion - used for low density applications; this level provides the largest land patterns and the largest courtyards around components.
- Density Level B: Median (Nominal) Land Protrusion - used for moderate density applications; this level provides median land patterns and median courtyards around components.
- Density Level C: Minimum (Least) Land Protrusion – used for high density applications; this level provides the smallest land patterns and the smallest courtyards around components.

To improve manufacturability, MSI recommends the use of density level A or B (in that order). Refer to IPC-7351 for a full explanation of density levels (section 1.4) and courtyard determination (section 3.1.5.4).

5.3 SMT Component Clearances;

To allow for reworking large SMT components on our automated rework station, the following clearances are requested:

- 5.5mm for components larger than 15.0 x 15.0mm
- 2.0mm for components equal to or smaller than 15.0mm x 15.0mm

If these clearances cannot be accommodated, it may be necessary to remove surrounding components prior to rework.

6. PTH Considerations

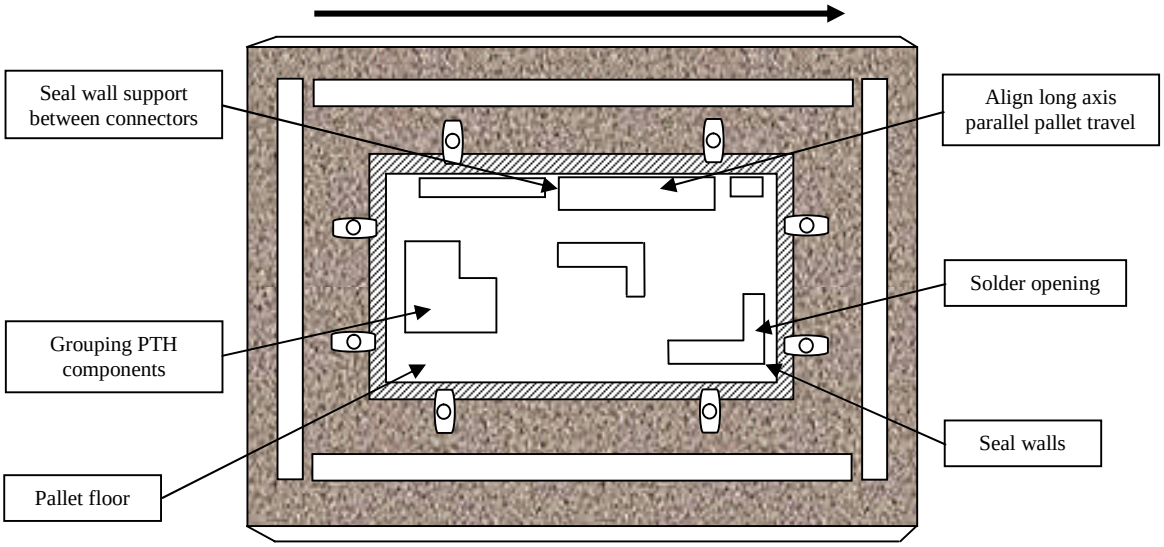
MSI typically uses a selective wave solder pallets for assemblies that have PTH components in addition to SMT components on both sides. Following the guidelines below will maximize the number of parts which can be wave soldered using custom selective solder pallets, which in turn will decrease assembly cost and increase quality levels.

6.1 PTH Component Placement;

Where possible, place all PTH components on the same side of the PCB. Placing PTH components on both sides of the PCB will normally require hand soldering the PTH parts on one of the sides. Placing PTH parts on the same side as the SMT parts on single-sided SMT assemblies will allow for the use of a universal wave solder pallet thus avoiding the expense of customer selective solder pallets.

The PTH parts should be aligned such that their longest axis is in the same direction on the PCB. Maximum yields are achieved when running the PCB with the longest axis of the PTH parts parallel to the direction of travel of the PCB through the wave. This means that the longest axis of the PTH parts is normally parallel to the longest axis of the PCB. PTH parts should be grouped together in order to allow for large openings in the selective wave solder pallet. No SMT parts should be placed within these openings.

PALLET DIRECTION THRU WAVE SOLDER



6.2 Solder Side Component Height;

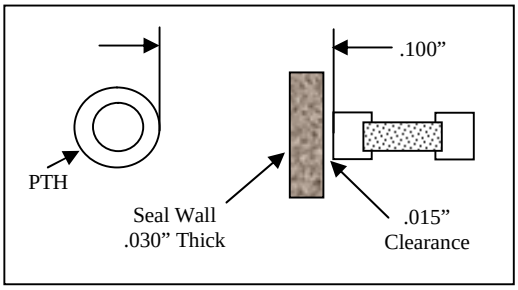
Avoid placing tall components on the solder side of the PCB, especially close to the PTH leads. The maximum height of any component on the solder side of the PCB is 0.250” measured from the surface of the PCB.

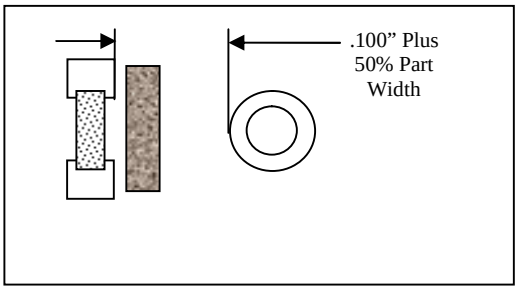
6.3 PTH Component Clearances;

6.3.1 PTH to SMT Components -

SMT components on the solder side (side exposed to the wave) need to be located far enough away from the PTH parts to allow the selective solder pallets to be manufactured with a large enough opening to allow solder penetration to the PTH lands plus enough distance for the thickness of the seal wall (which stops the solder from contacting the SMT parts). The recommended distances from the outside edge of the PTH annular ring to the nearest SMT components are:

- 0.100” to the short axis of the SMT pad layout
- 0.100” + ½ the SMT component body width to the long axis of the SMT pad layout





6.3.2 Clearance for Solder Openings in the Pallet -

The openings in the selective solder wave pallet require certain aspect ratios (length/width to height) in order to allow the solder to enter the opening and reach the bottom of the PCB. As the height of components on the solder side increases, the size of the opening in the pallet also needs to increase.

Component Height Immediately Outside Opening	Ratio X : Y	Min. X	Min Y
No components nearby	1 : 1	0.150”	0.150”
Max. height of 0.250”	3 : 1	0.450”	0.150”

If the dimension in the ‘Y’ direction is ≥ 0.300”, then an aspect ratio of 1 : 1 can be used for the ‘x’ and ‘y’ dimensions.

WAVE SOLDER TRAVEL DIRECTION