



# **Design for Manufacturing (DFM) Guidelines**

## **Design for Manufacturing (DFM) Guidelines**

**Why DFM?** The purpose of designing a part for manufacturing is to improve the product along three dimensions: quality, delivery, and cost. Utilizing sound DFM practices will ensure quality, reduce delivery lead-times and provide a reduction in the product cost by carefully selecting the best component for your application. In this day of global competition where cost, quality and speed to market are the key to a successful product, too often we witness very simple mistakes that are costly for our customer that could have been easily corrected during the design stage.

Many errors are systemic across different customers and industries. Some of the most common errors:

- 1) Missing or invalid manufacturing part numbers
- 2) Selection of components that are at the end of life or not available through distributors
- 3) No allowances for alternates on components
- 4) Selection of components that cannot withstand process temperatures (reflow)
- 5) The design requires unnecessary "extra" manual assembly effort
- 6) Distance between plated through hole parts (PTH) and surface mount technology (SMT) components is inadequate
- 7) Connectors or other components too close or overhanging edge of board
- 8) Incorrect pad size for the part geometry
- 9) PCB specifications not adequately detailed (e.g. board material, thickness)
- 10) PCB layout provides poor access for testing
- 11) Via holes & board pad proximity too close
- 12) Pin spacing on PTH components wrong for machine placement, PTH holes incorrectly sized for part

Our general guidelines for PCB development:

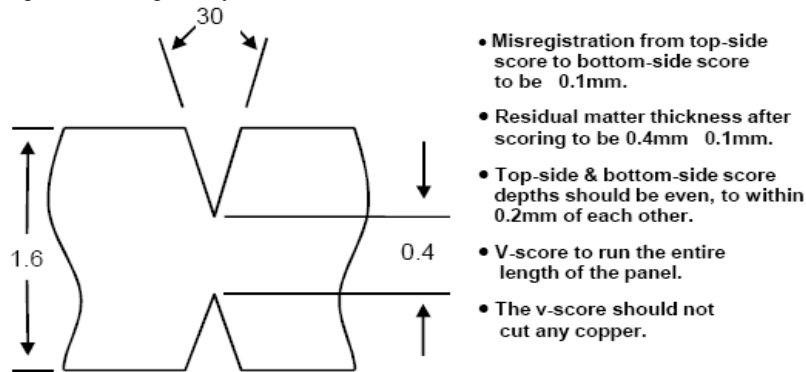


## **Bill of Material (BOM) creation and Component Selection**

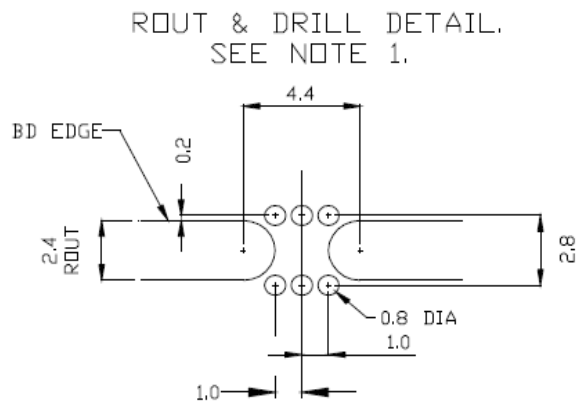
- Select a manufacturing part number for each and every item on the BOM.
- Review end of life, lead-times and availability of components selected. Select the best component that is not near its end-of-life, the component whose lead-time is within your delivery window and the component that is widely available in distribution. All too often it is only after the design is completed and implemented do you realize a difficult or long lead-time item has been unnecessarily designed in the product.
- Select components with machine placeable features and packaging options. This will avoid driving additional costs by incurring additional manual handling of components.
- When possible, list alternate components to allow us to find the best price/lead time part.
- For specialized components (e.g. ICs) or high value items, register the parts with the manufacturer and negotiate special pricing upfront. Once designed in, you will have little control on pricing or delivery as the manufacturer knows they have you “locked in”. Communicate pricing and the distributor to us.
- Select components where the required Minimum Purchase Quantity best matches your annual demand. Expected demand of 1000 units of a component that is only offered in 2500 unit reels multiples creates obsolescence and waste.
- On SMT assemblies use headers that can be placed with automatic pick & place equipment. Avoid selecting components that need to be hand placed as this increases costs and the opportunity for errors.
- Select parts that can withstand the temperatures of the reflow ovens and wave solder equipment. Restrictions on Hazardous Substances (RoHS) parts need to withstand 260° Celsius as a maximum temperature.
- It is unadvisable for Ball Grid Arrays (BGA) RoHS components to undergo the traditional Tin-Lead process; however, they can be processed but with higher reflow oven temperature profile. The key is temperature; the temperature must be high enough to allow the balls to fully melt.
- Wires need to be specified for Gauge, Length, Colour, and UL rating. Remember to include manufacturer’s part number.

## PCB Size and Mechanical Fit

- Maximum machine PCB Size placement we can process is 460mm x 400mm. The key is that size does matter and bigger is not always better. The larger the panel, typically the more difficult to process.
- PCB Shape must have two parallel sides (longest sides) to process through automation.
  - 0.125" keep out area along board edges or rails/break off tabs is required
  - 0.250" rail/break off tab minimum size
    - Let us optimize for lowest cost by having flexibility on panelisation
    - Panels become less stable as the array size increases. We will consider the lot size requirements when establishing the array size
    - Include information about overhanging parts (outline & keep out areas)
- V-Groove scoring is applied to both sides of the board. Components or other features should not be too close to the edge or damage may occur.



- Routing & Perforated Tabs.



- The location of the cut tab shall be specified, if critical as it may cause interference upon assembly into the application box or hardware.

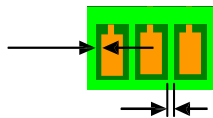
## PCB Design

- Layers of the PCB should be balanced in both layer count and across the board to avoid warp age.
- Specify surface finishes:
  - Horizontal Hot Air Leveling (HASL) is preferred for Tin-Lead applications or Lead Free Horizontal Hot Air Leveling (HASL) for non-leaded applications
    - Exceptions do apply for CSP, QFN, or ultra fine pitch parts where pad co planarity is critical. In these cases, HASL is not the preferred method due to pad doming
  - Electroless Nickel Immersion Gold (ENIG) is preferred for RoHS applications and fine pitch devices
    - ENIG has several advantages over more conventional (and cheaper) surface plating processes such as HASL, including excellent surface planarity, good oxidation resistance, and usability on untreated contact points

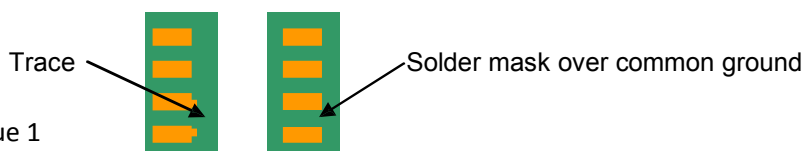
Lead Free Printed Circuit Board Finishes Comparison Chart (not inclusive of all features)

Finish	Shelf life	Solderability	Pro	Con
Electroless Nickel Immersion Gold (ENIG)	Excellent	Good	Excellent wetting Excellent Co planarity	Complex Process Not Reworkable
Hot Air Solder Leveling (HASL)	Good	Excellent	Excellent wetting Excellent Solder Joints	Poor Co planarity Dark Solder Joints
Organic Solderability Preservative (OSP)	Poor	Good	Very Low Cost Good Surface Oxidation	Degrades with High Temp Poor Wettability
Hard Gold	Good	Average	Good Wettability Good Co planarity	Brittle Solder Joints High Cost & Not Reworkable
RoHS HASL	Good	Good	Good Wettability Low Cost	Co planarity issues
White Tin	Average	Good	Overall good principles	Develops Tin whiskers Hand Solder Difficulty
Immersion Silver	Good	Good	Good Wettability Low Cost & Reworkable	Oxidation Issues Packed in Sulphur free paper

- Fiducials: 2 fiducials on corners of the board or panel. 0.050" round dot with 0.100" soldermask clearance.
- Fiducials must be at least of 0.190" from rail edges to prevent obstruction of clamps.
  - On larger arrays, add local fiducials near any fine pitch component
- Soldermask clearance from copper pad edges is typically 0.003" with 0.002" minimum.
- Use Soldermask dams between pads to minimize solder bridging – 0.005" is preferred but 0.003" is the industry minimum.



- Connect common pad sizes (IC & QFN) outside the pad area with a trace or dam with mask.



- Component spacing 0.060" clearance around BGAs for inspection and rework.
- Component footprints should be in accordance to the device part specification from the manufacturer and IPC-7351.
- For Quad Fine Pitch No Lead (QFN) components: keep the pad sizes consistent with each other to prevent tilting which can lead to an 'open' position. Extending the pad size beyond the shape of the component is recommended for inspection purposes.
- Via masking options: Covered, Flooded, Plugged or Tented.
  - Cover via to prevent solder from being lost down into the via and starving the joint
- Avoid vias within the land pattern, underneath components or closely connected to pads.
  - If vias are close to a pad, plug the via or provide a soldermask dam between the pad and via to prevent solder from wicking to via and robbing solder from the intended joint
  - Via size will drive the cost of the PCB 0.010" is standard dimension



- Clearly identify polarities on the silkscreen and on assembly drawings.
  - Indicate preference (or no preference) on Non-Polarized parts
- Place polarity consistently in one direction to reduce potential errors and inspection time.
- Radial through hole components should have hole spacing typically of 0.100", 0.200" or 0.300" on center.
- Dual in Line Package (DIP) components typically have a lead splay of 0.300", 0.400, or 0.600".
- Axial through hole components shall have center to center spacing of 0.300" to 0.800".
  - Avoid hand inserted "hairpin" axial device requirements
- Maintain 0.125" "Keep Out" area between the plated through hole (PTH) lead and the surface mount technology (SMT) components for ease of Selective Soldering or to allow the Selective Wave solder pallets to be used.
  - Locate all PTH on primary side of the board to avoid manual soldering
- Place all large mass parts on the Primary side of the PCB and items of lower mass on the Secondary side. This will help in ease of manufacturing.
- **DO NOT** mirror image Ball Grid Array (BGA) components on opposing sides. This makes inspection & rework very difficult.
- Minimize SMT component height on secondary side of the PCB to a maximum of 0.25".
  - Typically, Secondary side should contain mostly passive components and smaller / low mass devices. Large / high mass parts do not go on the Secondary side. Large parts placed on the Secondary side could overcome solder paste surface tension and fall off
- Design for testability: 0.030" – 0.040" test pads on 0.100" centers. Test point for each node.
  - Test pads to be 0.030" from other components. All test points on the Secondary side of the PCB
  - Provide test specification drawings to us. Define functional test methods
  - Define Environmental Test Screening (ESS) requirements if needed

## **PCB Drawing & Assembly Drawing Notes**

- Define PCB fabrications: Copper weights & Laminate type.
  - Copper thickness: common thickness for general use is 1 ounce (1.4mm or 0.0014")
  - Soldermask type: liquid photo imageable (LPI) green is most common
  - Silkscreen color: white is most common
  - Laminate type: FR4, RoHS. Laminate should be specified for RoHS applications
  - Board thickness: 0.0625" is typical. 0.031" & 0.093" are also common
- Define workmanship standard: IPC 610 Class 2 or Class 3.
- Specify conformal coating requirements, if needed Acrylic, Polyurethane, or Silicon.
  - Specify any "Keep Out" areas for coating (e.g. headers, connectors, etc)
- Specify any potting requirements.
- Specify any special adhesives, RTV or tamper seals, if needed. (Note on drawings)
- Define labeling requirements for board assembly and programmed parts.
- Note special packaging if required for finished goods.
- Specify any UL, CSA, ATEX or other Regulatory Agency requirements.
  - Provide a list of critical components
- Through holes that need to be "un-filled" or free of solder should be clearly identified.
  - Often "no stuff" parts are to be added on at the OEM and need to have holes open
- Define flux and cleaning requirements or provide an allowance to use "No Clean" flux (preferred).
  - Indicate if there are "non-hermetically" sealed components that cannot be washed
- Flying probe has "no-fly zones" which are tall components shadowing access points. This can hamper automatic inspection and test access. Call for advice on preferred test methods.
- Moisture Sensitive Device (MSD) handling must be considered for sensitive components and bare boards.
  - If consigning, keep materials sealed according to the manufacturers guidelines for MSD and audit process
- Components that need special forming or preparation should be specified. (e.g. Components required to be held to a certain height tolerance on the PCB).

While this is not an exhaustive list, the examples given should help reduce and in some cases eliminate waste and inefficient design choices.