VERGENT
PRODUCTS
Continually Elevating Return

Catch it early…
…and make it easier!

An overview of design practices that supports manufacturability of Printed Circuit Board Assemblies

Client Information and Design Rules for Product Designers

INTRODUCTION:

In the world of electronics, there are four distinct viewpoints as follows:

1) the inventor
2) the designer
3) the manufacturer
4) the end-user

One must consider the requirements of all four when striving to develop the most capable product with minimal cost, as all four contribute significantly to the successful outcome of the finished good.

From a contract manufacturer's perspective, variation exists in the design of a printed circuit board, or PCB, and a printed circuit board assembly, also known as PCBA. This variation can either 'make or break' the success of consistently building a quality product. Although many guidelines and standards do exist to help define designs that are easier to manufacture, awareness of their existence and the nature of their intent may not be fully or consistently understood. It is not uncommon for PCBs and PCBAs to be designed by individuals with mixed or minimal experience in circuit board design or the manufacturing processes used to build the actual product. This fact alone serves to explain the variation found in what contract manufacturers often see in product designs.

The purpose of this document is to bring awareness to a variety of issues that are commonly found during manufacturing, many of which can make it more difficult and more expensive to build certain printed circuit board assemblies (PCBAs). It is written for an audience with mixed PCBA experience. What is 'mixed PCBA experience' you might ask? Well, it's vague at best, so please ask us if you are uncertain of a recommendation or particular situation.

We have tried to create and maintain a relaxed presentation of information that uses the same syntax throughout, specifically to:

1) identify the issue
2) establish importance
3) offer solution

It is highly recommended that designs be reviewed with regard to the following list of concerns and that any discrepancies be resolved prior to having the product built.

Remember that designing a product that is easier to build supports the success of that product regardless of who manufactures it.

This is considered a 'living' document. Vergent Products welcomes your thoughts and comments for making it better. Please forward any improvement ideas that you may have for review and inclusion.

Now let's get started…

Fiducials inadequate or missing?

Now everybody knows that, or, wait a minute… Not everybody knows that automated manufacturing equipment requires reference points, called fiducials, to accurately register and orient assemblies during production. Each panel should contain at least three fiducials, two of which are diagonal and as far apart from each other as possible. The third should be in the same plane as at least one of the other two fiducials. Each fiducial should be finished copper measuring .040” in diameter with a .100” keep-out area that is relieved of copper and free of any silkscreen, attribute, or other material that would jeopardize the contrast required for successful automated registration. In simple terms, think about a donut and maybe even draw one. The donut hole represents the .040” finished metal fiducial, the actual donut is the keep out area and the outer circle would represent the outside perimeter of that keep-out area. Fiducials need to be inboard of the panel edge by at least .200” to ensure they are not covered by any clamping mechanisms inherent to the automated manufacturing equipment. Additional panel material will be required if these fiducials are too close to the edge. Furthermore, it is recommended that each PCB within an arrayed panel employ this three-fiducial format.

Polarity marks exist and are visible after component is installed?

All polarized locations should have corresponding polarity marks that are either silkscreened onto the PCB or are included in the copper layer. Having these polarity marks visible after the component is installed is essential for prompt and accurate post-install verification.

Is diode orientation identified clearly?

The variety of diode designs supports the need to clearly identify their orientation. Tradition symbols such as ‘+’ or ‘-’ tend to be ambiguous when working with diodes and should be avoided, as a result. Ask for exactly what you want by putting a ‘C’ or a ‘K’ on the cathode side and an ‘A’ on the anode side or use the schematic diode symbol for clarity. This works well as long as the designators for the diode locations are also clearly identified, (D1, D2, D3, etc.) Whether these marks are silkscreened on the board or introduced within the copper layer, always make sure they are still visible after the parts have been loaded.

Minimal Lead/Hole-size relationship?

Adequate space in the lead-to-hole-size relationship of through-hole applications is very important. It needs to be accounted for when the PCB is designed. This can directly affect vertical fill of solder and top-side wetting on through-hole designs by improving capillary action. It can also directly affect the success of automated insertion. The <u>finished</u> hole-size should be at least .019” larger than the diameter of the lead that accompanies it. Universal Instruments® also recommends a .019” delta when through-hole components are inserted with high-speed insertion equipment. In this model, the diameter across a square lead would be the distance across opposite corners (hypotenuse.)

Tabs/mousebites?

Tabs are used to connect PCBs within a routed panel. Mousebites are the perforations or holes that accompany these tabs to allow the cleanest results upon separation. General specifications for a .063” thick PCB with average component density might be .050” wide tabs placed approximately every 2” with mousebites that consist of .018” holes placed on the center of the PCB edge at .028” pitch. Larger, thicker and heavier designs will certainly benefit from wider tabs. Remember to relieve internal copper from the PCB edge and distance any nearby traces from these tab/mousebite locations to prevent exposed copper or damaged traces upon separation. Most contract manufacturers are willing to design the panelized array for you. Vergent Products prefers doing it because it ensures the panel is optimized for our processes.

Panel breakaways interfere with overhanging components or features?

Panel design should allow adequate relief for anything that might overhang the edge of the PCB, such as a connector or other feature. Tabs/mousebites should not be placed under any physical attribute that may interfere with the de-paneling process. Fortify the PCB Gerber data with relief information, as well as any 'keep-out' areas that would clearly define locations that a tab/mousebite cannot go, such as under an overhanging connector or near a trace that simply cannot be moved inward from the PCB edge. This information is also helpful when scoring is used.

Components too close to edge?

Manufacturing equipment clamping mechanisms require un-obstructed room to grab the PCB effectively. Will the PCB require extra panel material that is removed once manufacturing is complete? Delicate components, such as ceramic capacitors, are at risk of cracking when they are placed too close to the edge especially if the panels are segregated by snapping them apart manually.

Dry parts exist?

Many contract manufactures use automated aqueous wash processes to thoroughly clean assemblies. Devices such as tactile switches, buzzers, speakers, batteries, and loosely wound inductors are but a few that may not be wash-friendly. Washing these devices may cause damage or they might be difficult to dry, thus requiring them to be installed using more expensive manual methods after the wash process is complete. Research sealed alternatives that can survive the wash. Communicate any components that cannot be washed. Think about a no-clean process if the balance of ingredients truly warrants doing so, but don't lose site of the no-clean flux residue that is left behind.

Are there vias in pads? Are un-masked vias far enough away from the solder site?

Vias tend to scavenge solder during reflow leaving insufficient solder joints. In many cases finding these insufficient joints depends on inspection and fixing them requires time to manually rework, clean, re-inspect and the additional heating cycle that is required to do so. Please, no vias in pads and always design with at least .015" or more of solder mask to serve as a solder dam that separates the pad from the un-masked via.

Un-masked vias or inadequate solder dams under BGA devices?

As mentioned earlier, un-masked vias tend to scavenge solder from the finished solder joint. BGA solder ball terminations that are uniform in size and shape is considered ideal per the Acceptability of Electronic Assemblies specification, known as IPC-A-610. (IPC— the Association Connecting Electronics Industries® is an organization that serves the standardization and connection of electronics industries.) Scavenging under a BGA affects uniform size and shape and can also affect functionality. Reworking BGA devices is tedious, time-consuming & costly. Mask the vias directly under BGA devices.

Solder mask at fine pitch locations?

Fine pitch devices are defined as those having a lead pitch, or the distance between the centers of two adjacent leads, of less than or equal to .025" (.635mm). Solder mask between the pads of fine pitch devices is recommended to reduce the chance of bridging defects, but first be sure you know what your PCB supplier requires for successful adhesion. Most require the design to deliver at least a .004" wide resist for adequate adhesion of the resist to the PCB laminate.

Different pad sizes for BGAs?

Remember that BGA solder ball terminations that are uniform in size and shape are considered ideal. Different pad sizes under the same BGA device can negatively affect this uniformity. Be sure to design with uniformity.

Mixed SMD and NSMD approaches to BGAs?

Solder Mask Defined and Non Solder Mask Defined applications should not be shared under the same device. Uniform size and shape is at risk. NSMD is preferred over SMD for the sake of reliability per IPC-7095.

Lead-free solder balls on BGAs in a leaded design?

Different solder alloys often require different thermal profiles for successful reflow. Leaded designs that incorporate BGAs with lead-free solder balls require a modified leaded solder profile to ensure the collapse of the lead-free balls. Inadequate collapse can certainly become a reliability issue. Always communicate when lead-free BGAs are used in a design that requires leaded solder.

Large deltas in thermal mass across assembly?

Large deltas in thermal mass, across components and internal to the PCB itself, can make it more challenging if not impossible to reach thermal equilibrium before the liquidus solder event occurs. Keep in mind that different thermal masses reach preferred temperatures at different rates. This can lead to tombstones, laminate bow and twist, failure to collapse on BGA devices, inconsistent solder quality, etc.

Thermal relief?

Vertical fill and top-side wetting in a wave solder process is dependent on heat. Thermal relief improves the ability to reach the temperatures required to achieve quality solder joints on through-hole devices and can also be very useful to balance surface mount land patterns associated with smaller packages. Its importance increases as thermal mass increases - the result of heavier ground planes and components, more layers and designs that use two ounce or greater copper. Ensure that adequate thermal relief exists throughout the design.

Traces that promote natural bridging?

Short inter-pad traces are not recommended because they promote natural bridging. Natural bridges may be acceptable electrically but they create confusion and error in both automated and manual inspection processes. Keep in mind that bridges are often removed manually using solder wick and additional heat. This process will have a tendency of disjoining the bridge unless it is actually a natural bridge. Regardless of the outcome, another heating cycle has occurred now and more time is required to clean and re-inspect the rework. Inspectors also become de-sensitized to these anomalies and bridges that join non-common conductors are missed as a result. Remember, “No features that facilitate natural bridges!”

Oversized geometry on thermal lugs?

Devices with large center areas requiring solder may be prone to bridging especially if the perimeter connections are very close to the center lug. A custom aperture design may be required here to reduce the amount of solder paste if you are designing the SMT solder paste layer.

Do wire modifications exist?

Wire modifications, especially those that use light gauge wire such as Kynar, are both tedious and time consuming to do well and they tend to be less reliable than if the modification was actually designed into the PCB itself.

Do packages fit the pads?

Components that do not fit the pads do not support design for manufacture. Poorly spaced pads can lead to defects such as tombstone, drawbridge, no component and poor heel fillets. Adhesive may be an option for securing devices in place on a mismatched pad layout, but this additional step takes time and resources and that can lead to higher costs.

Really small components, really fine pitch, leadless devices?

Really small components, parts with really fine pitch and leadless devices tend to be more expensive and more difficult to assemble. Sure, they have their purpose, but it is not practical to use them when the application does not truly warrant their presence.

Courtyard issues, are devices to close?

Extremely tight courtyards can push expectations past what is considered acceptable per IPC. They also make rework very difficult. It is recommended to leave at least .150" of open space around BGAs or any other devices that require hot air for rework, and more open space whenever possible. Close proximity of high profile components such as electrolytic 'can' capacitors can also make it impossible to achieve adequate reflow temperatures for those devices without over-exposing the rest of the assembly. Spread things out or revise the layout to support the ability to solder the entire assembly.

Devices that are difficult or cannot be reworked post assembly?

Not really common, but take heed as a defect that cannot be fixed becomes scrap. Components that are too small, courtyard and other proximity issues, and pads that are too small to access with a soldering iron can make rework challenging if not impossible. Design with the ability to re-work in mind.

Is no-clean flux residue acceptable?

Some designs, high impedance critical ones for example, are affected by flux residue, regardless of its chemistry. What is the customer's expectation for cleaning? Is no-clean flux residue acceptable as IPC-A-610 states? Please communicate your requirements regarding no-clean flux residue.

Are trace cuts required?

Trace cuts are tedious and time consuming to do well. This manual modification uses very sharp or high speed tools that intentionally damage the PCB to fulfill the requirement. It is best to design the lack of continuity into the PCB itself.

SMT verses PTH mix?

The mix of surface mount technology and through-hole technology should be reviewed to ensure the best approach. Vergent Products recommends SMT whenever possible. Avoid extreme situations of few PTH parts on an otherwise all SMT design or vice-versa whenever possible. Employing both technologies drives higher assembly costs.

Is the product controlled by RoHS? Can all of the ingredients survive the higher temperatures associated with lead-free solder processing?

The Restriction of Hazardous Substances (RoHS) Directive requires that certain electronic products meet certain restrictions based on the legislation. Not only does the design have to meet the limitations dictated by RoHS, it also must be able to survive the increased temperatures associated with lead-free solder processing. Peak lead-free solder temperatures could achieve 260°C, so all components in a RoHS design need to account for this if automated manufacturing processes are to be used.

Is the PCB finish clearly identified?

PCB finishes vary in their ability to solder, flatness, storage life and cost. Remember that Hot Air Solder Level (HASL) is often the least expensive but Electroless Nickel Immersion Gold (ENIG) and Immersion Silver (ImAg) are more flat. Designs that incorporate fine pitch devices, BGAs, QFNs, or other packages with small or hidden terminations require a flat finish. Preferred flat finishes in order of preference are ENIG and ImAg. ENIG tends to be more expensive than ImAg but it also has a longer shelf life without the special packaging requirements of ImAg. Organic Solder Preservative (OSP) and Immersion Tin (ImSn) are not preferred finishes and must be avoided in Vergent Products processes. Clearly identify the surface finish that you require, for instance:

- Electroless Nickel Immersion Gold (ENIG) finish

Or is it…

- Lead-free RoHS-compliant Hot Air solder Level (HASL)

Is the laminate stack clearly defined?

PCB's vary in the number of layers, copper weight and overall thickness. These variables must be defined to ensure the PCB is manufactured correctly.

Is 'trace & space' and smallest hole diameter identified?

Minimum trace width, minimum distance between traces and the smallest hole diameter is critical when determining who can manufacture the PCB and how expensive it will be. Communicate these specifications in the fabrication notes.

Are build standards identified?

It is important that workmanship specifications be clearly identified. Clearly communicating any and all expectations is essential to the success of the finished product. Keep in mind that IPC—Association Connecting Electronics Industries® has a tiered class of standards to govern the acceptability of electronic assemblies. IPC-A-600 covers the raw PCB & IPC-A-610 regards the finished PCBA. The tiered structure is:

Class 1 – General Electronics Products, basically the device has to function.

Class 2 – Dedicated Service Electronic Products, products where continued performance and extended life is required but uninterrupted service is not critical.

Class 3 – High Performance Electronics Products, high performance products where continued or on-demand functionality is critical, such as life support systems.

Clearly identify the level of workmanship and any exceptions as notes within the fabrication notes.

Maximum potential voltage identified?

IPC provides specifications for minimal electrical clearance that are determined by the amount of voltage inherent to a specific product. Manufacturing cannot use those specifications effectively if the greatest potential voltage difference is unknown. Identify the greatest potential voltage difference that is specific to the PCBA. For example:

- Greatest potential difference on the PCBA is 50 Volts.

PCB base-material laminate specs missing or inadequate?

It is essential that base material specifications identify material that will survive the manufacturing processes specific to the product. This becomes absolutely essential in RoHS compliant designs simply because of the higher temperatures required to process the lead-free solder alloys. A minimum glass transition temperature or Tg ≥ 170°C is required for lead-free solder processes. The IPC-4101 standard identifies these laminate specifications in greater detail.

Does a location exist for a label?

Product identification and traceability is an ISO requirement in manufacturing. Room for the Vergent Products standard label is recommended. This label requires 1.0” x .25” of un-populated real-estate and is best identified with a box in the silkscreen. Smaller labels are possible but remember cost increases when the standard is not used.

Does a legend exist for Gerber layer files?

There is no standard for naming Gerber layer filenames or extensions. Navigating Gerber layers can be challenging without an understanding of how these filenames and extensions were derived. Include a legend that decodes Gerber layer filenames and extensions. This legend is best located in the Gerber data but may also reside in a README.TXT file that accompanies the data.

Does the product have Regulatory Requirements?

Regulatory requirements are serious business. Always clearly identify all regulatory requirements.

Multi source AVL/AML?

Component availability can be a moving target. Lead times may vary and components become obsolete. Multiple sources for components can be very helpful with regard to availability, lead-time and cost. At least two sources for each component are preferred whenever Approved Vendor List (AVL) and Approved Manufacturer List (AML) source control is required. Furthermore, all sources must be approved if the assembly is governed by a regulatory agency, such as UL, ETL, CE, or others.

Torque specification identified?

Tightening individual fasteners to the proper torque assures product consistency and reliability. Please include torque specifications in the assembly documentation for any fasteners that might exist.

Clearly defined fabrication notes exist?

We have recommended that several specifications be included in the fabrication notes. These fabrication notes are best kept in the PCB Gerber data whenever possible or in an appropriate assembly or fabrication document. It is important that they exist, are clearly and thoroughly defined and that they are accessible. The following list is an example of what a set of fabrication notes should look like:

- PCB manufactured per IPC-6012 current revision Class 2 and meet IPC-A-600 current revision Class 2 inspection requirements.
- PCBA finished assembly manufactured per IPC-A-610 current revision Class 2.
- PCB & PCBA must be RoHS complaint.
- PCB must be able to withstand the temperatures associated with lead-free solder processes.
- PCB laminate Tg ≥ 170°C.
- Electroless Nickel Immersion Gold (ENIG) finish.
- Green solder mask.
- White silkscreen.
- PCB is six layers, 1oz. outer copper layers, 0.5oz. inner copper layers, .063” ±10% thick.
- Trace & space = .008” minimum.
- Smallest hole size = .018” diameter.
- Greatest potential difference on the PCBA is 50 Volts.
- No upside down passive SMT chip components.
- No-clean flux residue visible at 4X magnification must be thoroughly removed from the PCBA.
- Transducer @ PSS1 cannot be washed. It must be installed with no-clean flux chemistry.

Electronic data?

Having the necessary data is considered essential when introducing new products to manufacturing. It is also very helpful in the maintenance of existing products. Electronic data in the correct format supports the manufacture’s ability to process a new order quickly and accurately.

1) Gerber data with PCB fabrication notes
2) CAD data in ASCII format
3) Centroid data with X/Y/theta information
4) Bill of Materials (BOM) with reference designators in Microsoft Excel or ASCII format

In addition, schematics are also required for test development and troubleshooting.

Any existing assembly instructions or inspection documentation is also valuable.

A known good finished assembly or ‘Golden Board’ is very helpful when quoting a product and setting up the manufacturing process. A ‘Golden Board’ is preferred with photographs being the next best alternative, whenever possible.

Closing

Thank you for taking the time to read this document. We hope you find the content both helpful and valuable. Remember that Vergent Products welcomes your thoughts and comments for making this a better document. Please feel free to contact us with any suggestions for improvement or questions that you may have.

Design Checklist

Done	*Attribute*
	Fiducials inadequate or missing?
	Polarity marks exist and are visible after component is installed?
	Is diode orientation identified clearly?
	Minimal Lead/Hole-size relationship?
	Tabs/mousebites?
	Panel breakaways interfere with overhanging components or features such as connectors?
	Components too close to edge?
	Dry parts exist?
	Are there vias in pads?
	Are un-masked vias far enough away from the solder site?
	Un-masked vias or inadequate solder dams under BGA devices?
	Solder mask at fine pitch locations?
	Different pad sizes for BGAs?
	Mixed SMD and NSMD approaches to BGAs?
	Lead-free solder balls on BGAs in a leaded design?
	Large deltas in thermal mass across assembly?
	Thermal relief?
	Traces that promote natural bridging?
	Oversized geometry on thermal lugs?
	Do wire modifications exist?
	Do packages fit the pads?
	Really small components, really fine pitch, leadless devices?
	Courtyard issues, are devices to close?
	Devices that are difficult or cannot be reworked post assembly?
	Is no-clean flux residue acceptable?
	Are trace cuts required?
	SMT verses PTH mix?
	Is the product controlled by RoHS?
	Can all of the ingredients survive the higher temperatures associated with lead-free solder processing?
	Is the PCB finish clearly identified?
	Is the laminate stack clearly defined?
	Is ‘trace & space’ and smallest hole diameter identified?
	Are build standards identified?
	Maximum potential voltage identified?
	PCB base-material laminate specs missing or inadequate?
	Does a location exist for a label?
	Does a legend exist for Gerber layer files?
	Does the product have Regulatory Requirements?
	Multi source AVL/AML?
	Torque specifications identified?
	Clearly defined fabrication notes exist?
	Electronic Data?