South Bay Circuits

Manufacturability Guidelines

For

Printed Circuit Board

Assemblies

Revision History

REV	DESCRIPTION	ORIGINATOR	REVISION DATE
A	Initial Release	Tom Tindale	03/12/02

Table of Contents

Introduction

The purpose of this document is to provide our customers with a set of guidelines that will allow them to design products that are both manufacturable and testable. Assemblies that are designed for manufacturability and testability will be easier (and faster) to produce, require less rework/repair, and generate less scrap. All of which will result in a more cost effective product for our customers. These guidelines are based on industry standard specifications, with the exception of those that pertain directly to equipment used at South Bay Circuits, Inc. Refer to Appendix A "South Bay Circuits, Site Capability Matrix", for specific capabilities within the facility.

Please note: For maximum impact, DFM reviews must be accomplished during the design stage, when changes are easy and cost effective.

1 References

1). ANSI/J-STD-001, Requirements for Soldered Electrical and Electronic Assemblies
2). IPC-T-50, Terms and Definitions for Interconnecting and Packaging Electronic Circuits
3). IPC-D-275, Design Standard for Rigid Printed Boards and Rigid Printed Board Assemblies
4). IPC-D-325A, Documentation Requirements for Printed Boards, Assemblies, and Support Drawings
5). IPC-A-610, Acceptability of Electronic Assemblies
6). IPC-R-700, Suggested Guidelines for Modification Rework, and Repair of Printed Boards and Assemblies
7). IPC-SM-782, Surface Mount Design and Land Pattern Standard
8). IPC-SM-786, Procedures for Characterizing and Handling of Moisture/Reflow Sensitive IC's
9). Motorola Semiconductor Technology Data – PBGA, AN12310

2 Workmanship Standards

1). All Electronic and Electro-Mechanical assemblies will conform to IPC-A-610, Acceptability of Electronic Assemblies, Class 2. for Dedicated Service Electronic Products, unless otherwise specified.
2). Dedicated Service Electronic Products includes: "communications equipment, sophisticated business machines, and instruments where high performance and extended life is required, and for which uninterrupted service is desired but not critical. Typically, the end-use environment would not cause failures."
3). Exceptions to Classification 2 level of assembly require management approval and will require a different response to an RFQ (Request for Quotation).

3 Hierarchy of Documents

In the event of conflicting requirements between applicable documents, the documents will take precedence in the following order:
1). The Purchase Order - Highest Priority
2). Applicable assembly and detail drawings including BOM Notes and Specifications
3). The customer standards manual if specified.
4). IPC-A-610, Acceptability of Electronic Assemblies guidelines

4 Documentation Requirements

4.1 Format

All documentation must be 1:1 copies of the original. Size “C” reductions to Size “A”, and faxes are not acceptable for quotation, or the planning/programming phase of the project. Electronic files over 2 megabytes should be transferred to SBC via the FTP site. Electronic files are preferred in all cases.

4.2 Assembly Drawings

The assembly drawing should include, but not be limited to, the following information:

1). Assembly Number and Revision
2). ECO History
3). Applicable Notes
4). Components with Reference Designators
5). Exploded views/details where required (including hardware stacking requirements, and label placement
6). information)
7). Dimension Tolerances if required.
8). Customer acceptance criteria.

4.3 PCB Fabrication Drawings

The drawing should include, but not be limited to the following information:

1). PCB Number and Revision
2). ECO History
3). Dimension Tolerances
4). Drill Chart - Identifies the hole legend, hole diameter, hole quantity, and plating requirements.
5). Tooling Hole Locations
6). Datum References
7). The theoretical-exact point, axis or plane that is the origin from which the location of geometric characteristics of features, of a part, are established.
8). PCB Material Requirements (i.e. laminate, copper, soldermask, surface finish etc.)
9). Layer Count
10). Layer Stack Configuration
11). Applicable Notes

4.4 Parts List/Manufacturing Bill of Material

The Parts List/Bill of Materials should include but not be limited to the following information:

1). Customer Component Part Number
2). Component description, including package information
3). Manufacturers Part Number and/or Approved Vendor List
4). Quantity per assembly, including unit of measure
5). Reference designators
6). Component revisions (in the case of programmed parts or PCB’s)

4.5 Gerber Files

Gerber files are used to create Printed circuit boards and Stencils. The following should be included and/or considered when generating Gerber files.

1). Drill files and drill guide must be included.
2). At least one layer must accurately depict the board outline.
3). Solder paste layers are to have 1:1 apertures with their corresponding pad layers. They should not be reduced or enlarged to compensate for fabrication or assembly limitations. South Bay Circuits’ stencil vendors will adjust apertures as per SBC’s stencil design guidelines.

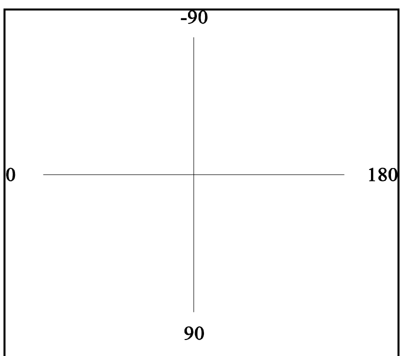
4). Fiducials should be included in the solder paste layer.
5). All corrections such as: soldermask relief, silk screen removal from pads, and trace routings, should be made in the CAD database prior to Gerber generation. Leaving these corrections for the PCB fabricator, may lead to delays and/or defects.
6). Netlist in IPC-D-356 Format for boards of 8 layers or more to facilitate Netlist testing
7). Aperture list, if not using 274X format.

4.6 CAD Data

CAD data is used to generate pick and place as well as other automated assembly programs. CAD data is also used to more efficiently generate internal assembly and inspection aids.

Centroid Data may be used in lieu of CAD data, provided the following information is included:
1). Reference Designator
2). Package Description (0805, SOIC16, SOT-23, etc.)
3). Part Number
4). X ,Y Data (SMT centroids, PTH pin 1 location)
5). Theta Rotation (see Detail A below)
6). Top or Bottom Identification

Files may be provided on 3.5" floppy, CD ROM, or transferred via e-mail or South Bay Circuits, Inc.'s FTP server.

Detail A


CAD Theta Orientation

4.7 Schematics and Test Specifications

Used for the development of In-Circuit and Functional Test Fixtures. Files may be provided on 3.5" floppy, CD ROM, or transferred via e-mail or South Bay Circuits, Inc.'s FTP server.

4.8 Engineering Change Orders (ECO's, ECN's etc.)

ECO's should include, but not be limited to the following information:

1). Applicable part number
2). Revision Level, from/to
3). Implementation date/orders effected
4). Part changes
5). Rework instructions
6). Disposition of material "on hand" (at customer sites and SBC)

5 PCB Layout and Design

NOTE: This section is intended to provide a generic overview of

PCB fabrication guidelines. It was compiled using common capabilities of several fabricators. Keep in mind that limitations to the assembly process may or may not match those in the fabrication process. Please see South Bay Circuits' "Manufacturing Guidelines for Printed Circuit Boards" for more detailed information regarding capabilities and guidelines for boards fabricated at South Bay Circuits.

5.1 References

1). IPC-2221, Design Standard for Rigid Printed Boards and Rigid Printed Board Assemblies
2). IPC-SM-782, Surface Mount Design and Land Pattern Standard
3). IPC-D-300G, Printed Board Dimensions and Tolerances
4). IPC-4101, Specification for Base Materials for Rigid and Multilayer Printed Boards

5.2 PCB Size

Overall PCB sizes vary due to customer design requirements. All PCB sizes within the following parameters can be accommodated by SBC for assembly.

Minimum board size: 4" X 4"
Note: With the use of breakaway rails, and multi-up arrays, minimum size constraints are rarely a concern

Maximum board size: 20" wide X 20" long
Note: This maximum size must include any necessary breakaway rails

5.3 PCB Thickness

Single layer, Double sided, and Multi-Layer boards from .024" (.6mm) to .120 (3 mm) can be accommodated. Special consideration should be given to PCB's larger than 8" X 10". A finished thickness of .062" or less can lead to warpage and/or defects related to excessive board flex. Defects such as fractured solder joints and cracked barrels, have been experienced.

5.4 PCB Shape

PCB's must have 2 parallel sides (preferably the 2 longest sides) for process through SBC's automated assembly equipment. Breakaway rails and/or panelization can be used to accommodate a wide variety of odd board shapes. (See section 5.11)

5.5 Layer Count and Construction

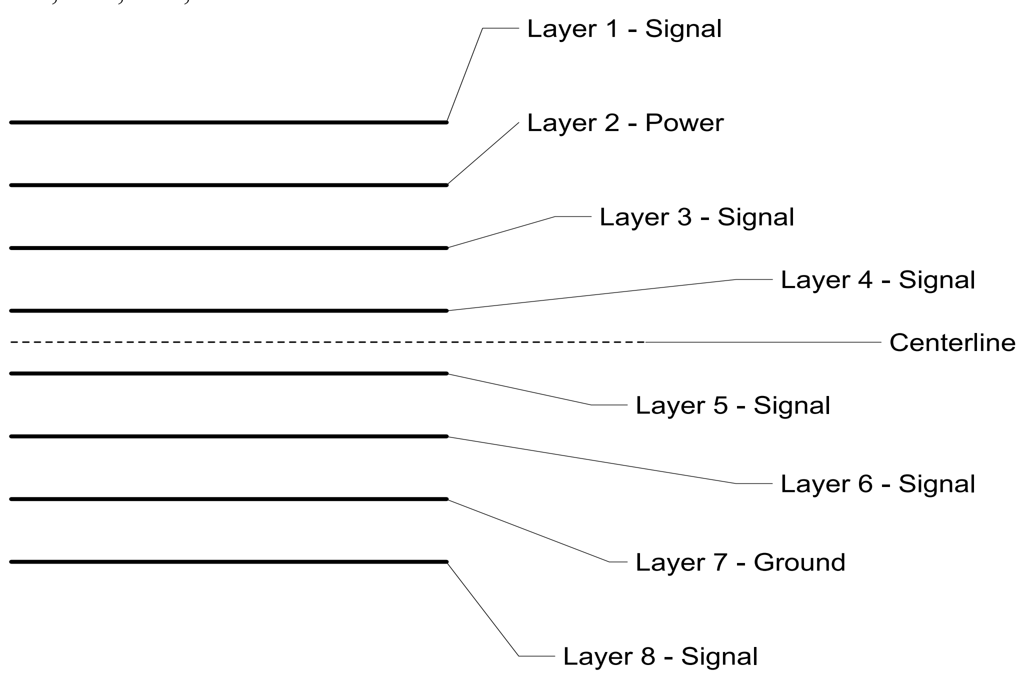
Multilayers that have an odd number of layers will often lead to warpage problems. Adding a non-functional layer to this type of multilayer is an approach that will minimize the potential for warp. A non- functional layer may be a plane layer with the copper relieved from all holes (no interconnects). This extra layer can also be a simulated signal layer with no connection to any holes. Consideration to the electrical performance should also be considered to avoid signal interference.
Layer stackup should be symmetrical with regard to dielectrics, copper weights, and layer design. Asymmetrical designs are a primary cause of warped boards.

Internal, external signal and ground layers should be designed so that every square inch of the layer has approximately the same percentage of retained copper. This will enable the fabricator to hold tighter tolerances on internal dielectrics and can help keep warpage to a minimum. This can be accomplished by adding non-functional pads and traces to the design. Often, non-standard constructions are chosen and are not necessary for meeting final product performance. This can add considerably to the board cost and diminish quality due to dimensional instability. Non standard designs should be reviewed with the fabricator for potential cost saving options.

Internal and external layers should have all copper relieved a minimum of .050" from board edges to accommodate panelization requirements.

Proper Layer Construction

1=8, 2=7, 3=6, 4=5



5.6 Copper Weights/Trace Widths/ Trace Spacings

5.6.1 Inner Layers – Internal layers should be designed with ½ oz copper foil, where current handling capacity permits. Where higher copper weights are required, 1oz is preferred over 2 oz. (2 oz copper can add as much as 20% to material costs).

Trace widths must be:

.0061" or greater for 2 oz copper
.0051" or greater for 1 oz copper
.0031" or greater for ½ oz coppers

Minimum Trace Spacing - .004"

5.6.2 Outer Layers - External layers are more effectively produced on ½ oz copper. The following guidelines should be used when choosing external layer copper.

Trace widths must be:

.0081" or greater for 1oz copper
.0050" or greater for ½ oz copper

Minimum Trace Spacing - .004"

5.7 Drill

5.7.1 Locating References

Of prime importance in PCB design and construction is the positioning of accurate reference points (datums) to which all holes are drilled. The references are used to position the PCB onto the workboard holders of the automatic insertion machines. It is recommended that the distance between the locating references be the maximum separation permitted by the length of the board. Also it is recommended that the locating reference hole diameter tolerance be held to ± 0.001" (0.025mm). The lower left reference hole is generally given the location terminology of X_0, Y_0.

5.7.2 Aspect Ratio

Aspect ratio is defined as the board thickness divided by the smallest drilled hole (typically via's). Higher aspect ratios increase the probability for voids in the holes. Aspect ratios higher than 6:1 should be confirmed with the PCB fabricator.

5.7.3 Hole Tolerancing

5.7.3.1 Plated Through holes

- A 6 mil window is preferred. 6 mil window is defined as +/- .003" or +.004"/-.002", etc.
- Plated through holes are normally drilled 4-6 mils over the nominal finished hole size. Copper and solder plating generally reduce the hole diameter by approximately 3-5 mils.
- Plated holes over .100" located at the edges of the board should have large outer layer pads to prevent them from plating too fast. Typically a pad .200" larger than the hole will help

5.7.3.2 Non-Plated Through Holes

- Non Plated holes require a 4 mil window. Outer layer pads for these should be removed.

5.7.4 Annular Ring

Annular ring is the annular solderable area remaining between the edge of a drilled hole and the edge of the associated pad. Designed annular ring is the difference between a pad and the corresponding drilled hole, divided by 2. By maintaining .005” annular ring in design, IPC requirements can be met despite drill wander. When minimum annular ring design is violated, PCB fabricators may incorporate a “tear drop” pad design to ensure sufficient annular ring at the trace/pad junction.

5.7.5 Pad Sizing

The following formula can be used for determining pad sizes for plated through holes.

PAD SIZE = FHS + .013” + (2 x A/R)

FHS – nominal finished hole size; A/R – finished annular ring requirement (.002” typical)
Smaller pad sizes may be possible, but can result in reduced producibility for the fabricator.

5.7.6 Standard Drill Sizes

The following shows a typical list of Standard Drill sizes common at many PCB fabricators.

Standard Drills Available

Size	Diameter	Size	Diameter	Size	Diameter	Size	Diameter
#97	.0059	#52	.0635	3.15mm	.124	#11	.191
#96	.0063	1.65mm	.065	1/8	.125	#10	.1935
#92	.0079	#51	.067	3.20mm	.126	4.95mm	.1949
#87	.01	1.75mm	.0689	#30	.1285	#9	.196
#83	.012	#50	.07	3.30mm	.1299	5.00mm	.1968
#80	.0135	1.80mm	.0709	3.35mm	.1319	#8	.199
#79	.0145	#49	.073	3.40mm	.1339	#7	.201
#78	.016	1.90mm	.0748	#29	.136	5.15mm	.2028
#77	.018	#48	.076	3.50mm	.1378	#6	.204
#76	.02	1.95mm	.0768	3.55mm	.1398	#5	.2055
#75	.021	#47	.0785	#28	.1405	5.25mm	.2067
#74	.0225	#46	.081	3.60mm	.1417	#4	.209
#73	.024	#45	.082	3.65mm	.1437	5.35mm	.2106
#72	.025	2.10mm	.0827	#27	.144	#3	.213
#71	.026	2.15mm	.0846	3.70mm	.1457	5.45mm	.2146
#70	.028	#44	.086	#26	.147	5.50mm	.2165
#69	.0292	2.20mm	.0866	#25	.1495	5.55mm	.2185
#68	.031	#43	.089	#24	.152	#2	.221
#67	.032	2.30mm	.0906	#23	.154	5.65mm	.2224
#66	.033	2.35mm	.0925	3.95mm	.1555	5.70mm	.2244
#65	.035	#42	.0935	#22	.157	5.75mm	.2264
#64	.036	2.40mm	.0945	#21	.159	#1	.228
#63	.037	#41	.096	#20	.161	5.85mm	.2302
#62	.038	#40	.098	4.15mm	.1634	5.90mm	.2323
#61	.039	#39	.0995	4.20mm	.1654	A	.234
#60	.04	2.55mm	.1004	#19	.166	6.00mm	.2362
#59	.041	#38	.1015	4.25mm	.1673	B	.238
#58	.042	2.60mm	.1024	#18	.1695	6.10mm	.2402
#57	.043	#37	.104	4.35mm	.1713	C	.242
1.15mm	.0453	#36	.1065	11/64	.1719	6.20mm	.2441
#56	.0465	2.75mm	.1083	#17	.173	D	.246
1.20mm	.0472	7/64	.1094	4.45mm	.1752	6.30mm	.248
1.25mm	.0492	#35	.11	#16	.177	1/4	.25
1.30mm	.0512	#34	.111	4.55mm	.1791	6.50mm	.2559
#55	.052	2.85mm	.1122	#15	.18	F	.257
1.35mm	.0531	#33	.113	4.60mm	.1811	G	.261
#54	.055	2.90mm	.1142	#14	.182	6.70mm	.2638
1.45mm	.0571	#32	.116	4.65mm	.1831	6.75mm	.2657
#53	.0595	3.00mm	.1181	#13	.185	6.80mm	.2677
1.55mm	.061	#31	.12	3/16	.1875		
1/16	.0625	3.10mm	.122	#12	.189		

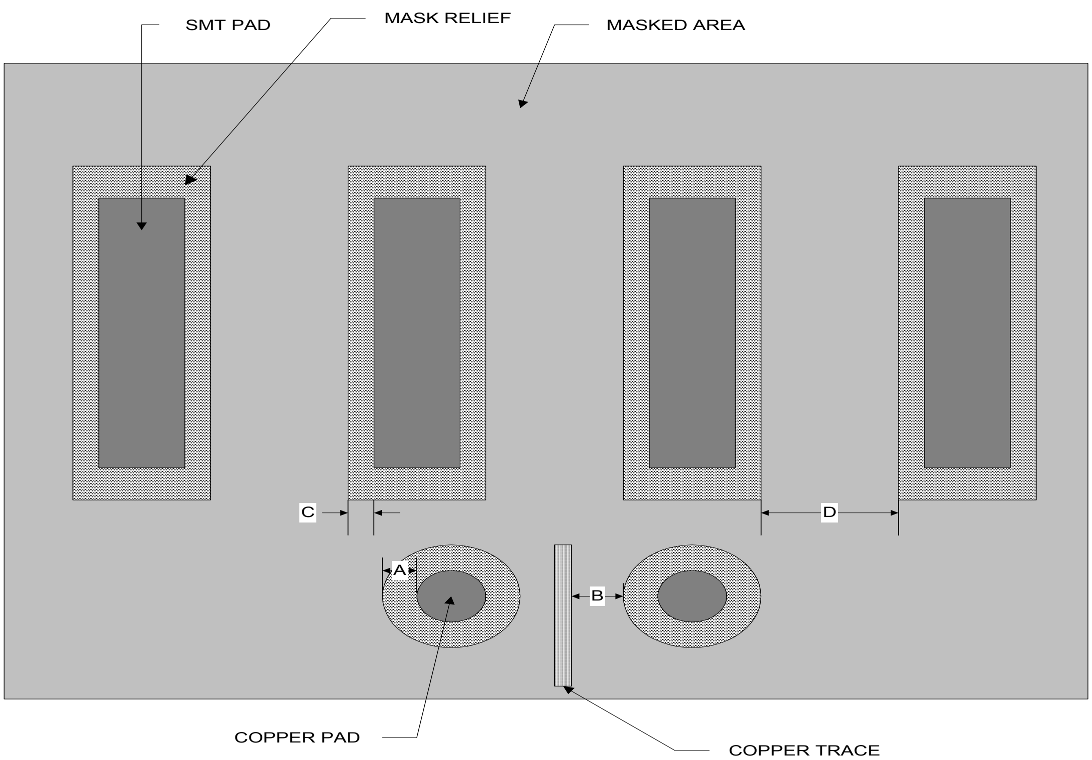
5.8 Soldermask

5.8.1 Types

1). Wet Screened Mask
- Manual application
- Uses woven mesh screens
- Thermal Cure
- Typical thickness .0006” - .0012”

2). Liquid Photoimageable Soldermask
- Automated operation
- Photo defined image
- Thermal cure
- Curtain coated
- Typical thickness - .0007” - .001”

5.8.2 Soldermask Clearances



	Preferred	**Minimum Acceptable**
A	.003” min	.002” min
B	.004” min	.003” min
C	.003” min	.002” min
D	.005” min	.004” min

5.8.3 Soldermask Dams

Soldermask dams between pads are required to minimize solder bridging in the Surface Mount process. As seen in the previous drawing, .005” dam is preferred with .003” being achievable by some fabricators.

Dams ARE achievable on ALL fine pitch devices down to .0197” pitch.

Example: .0197” pitch - .011” pad = .0087” space between pads - .005” (.0025” soldermask relief on either side) = .0037” solder mask dam.

5.8.4 Via Masking

There are several methods used for treating vias with soldermask.

1). **Covered vias** – soldermask is allowed to encroach on via pad. The mask relief is slightly larger than the via hole.

2). **Flooded vias** – Soldermask is allowed to completely cover the pad, and flow into the via hole. A meniscus may or may not be present after cure.

3). **Plugged vias** – Same mask relief as Covered via, but with a secondary plugging process. Can fill up to 2/3 of the barrel. Should be used where vacuum draw is needed for in-circuit test. To prevent bridging or thieving, covered or flooded vias may be used.

4). **Tented vias** – Created only with dry film soldermask.

Constraints:

1). Can only be accomplished with vias .018” (finished hole size) or less.

2). To avoid entrapped contaminants, boards should never be plugged from both sides.

NOTE: SOUTH BAY CIRCUITS, INC. RECOMMENDS LPI SOLDERMASK APPLIED DIRECTLY OVER BARE COPPER FOR ALL APPLICATIONS.

5.9 Surface Finishes

South Bay Circuits, Inc.'s assembly process can accommodate a variety of surface finishes. The most common are detailed below. Please contact South Bay Circuits, Inc. Engineering if using a product that is not listed.

Coating Process	**Advantages**	**Limitations**	**Environment Concerns**	**Cost**	**Shelf Life**
HASL	Solderability	Planarity	High	Low	Long
Electroless Ni/Immersion Au	Solderability, good for fine pitch	Poor Au bonding, solder pot contamination	Med	Med	Long
Immersion White Tin	Solderability, planarity, good wear	none	Med	Med	Long
Hard Au, Electrolytic Au	Good wear, solderability	Resist breakdown	Low	High	Long
Organic Solder Preservative (OSP)	Fine Pitch, Solderability,	Special Handling required	Low	Low	One Year

5.10 Silk-screened Legends

Legends are manually silk-screened using non-conductive epoxy ink. This process is not very accurate as far as registration and resolution. Legibility is dependent on character size. Minimum character height should be .040" with .007" spacing between characters. Optimum line width is .007". If spacing is less than .007", then the height should be increased to .045".

Other considerations:

- Legend should be clipped from all pad surfaces. Do not rely on the PCB fabricator to fix this.
- Component polarity markings should be outside the body when possible to allow inspection.
- Legend over covered or plugged vias should be avoided underneath components. The build up of material can effect component coplanarity.

5.11 Panelization

Panelization is required when one or more of the following conditions exist:

- The PCB shape does not have parallel edges along the 2 longest sides for conveyor transport. (See section 6.1 for detail)
- The PCB does not have sufficient clearance between components and the edge of the board. (See section 6.3 for detail)
- The size of the PCB does not meet the minimum requirement for the assembly equipment (See section 5.2 for detail)
- Facilitate processing and handling of higher volume assemblies.

In most cases, SBC receives one-up data from our customers, and designs the panels in house, to ensure compliance with assembly equipment requirements. In the case of a customer supplied panel design, SBC requests to review and approve, prior to PCB fabrication. Breakaway rails may contain holes ≥ .250", plating marks, clamp marks, etc, as long as fiducial and hole keep out areas are maintained.

5.11.1 Size and Shape

Overall panel dimensions must not exceed those for individual boards. (See section 5.2) Shape should be square or rectangular. The longest edges must have sufficient clearance or breakaway rails to accommodate conveyorized handling. If the short sides are used for transport, increased warping will occur during the process. Large gaps in the transport sides must be filled with breakaway material.

5.11.2 Array Design

Array design should take into account board size, orientation, run volumes,
and PCB fab master panel utilization. The following guidelines should be used:

1). Array size should not cause overall panel design to approach maximum board handling limits.

2). Panels will become less stable as array size increases.

3). Keep array numbers comparable with average lot requirements. If average lot size is 24, an 8 up panel will work better than a 10 up configuration.

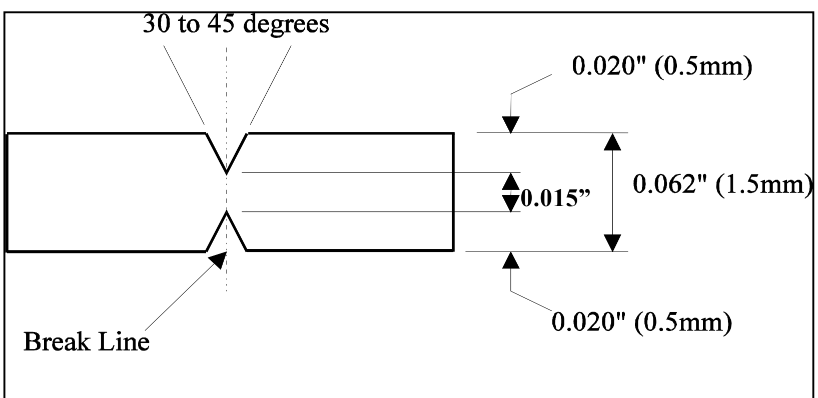
4). Unused material in the fabricator's master panel will increase board cost.

5). Assemblies with overhanging components must be panelized with sufficient space between boards.

5.11.3 Retention Methods

5.11.3.1 V-Groove Scoring

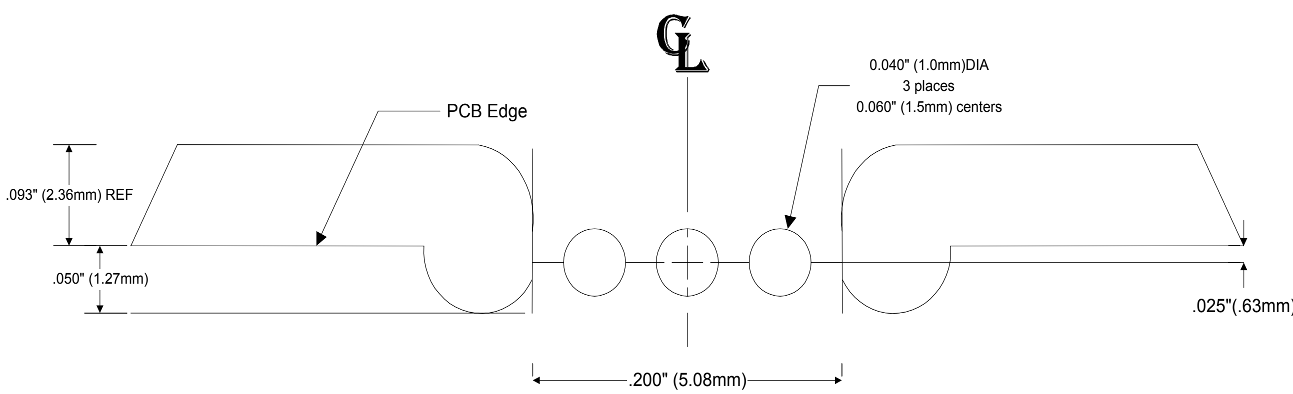
V-groove scoring is generally provided on both surfaces of the board, and only in a straight line. A small cross section of board material is retained at the break line. An allowance for the scoring angle must be maintained. Features that are located too close to the score groove may be exposed or damaged. As a general rule, .035"-.050" should be allowed. For most fabricators, the remaining material, or the "web thickness" is the controlling dimension. The web thickness should be based on depaneling method, panel size and rigidity, as well as proximity of components to the edge. Web thickness tolerance is +/- .005"

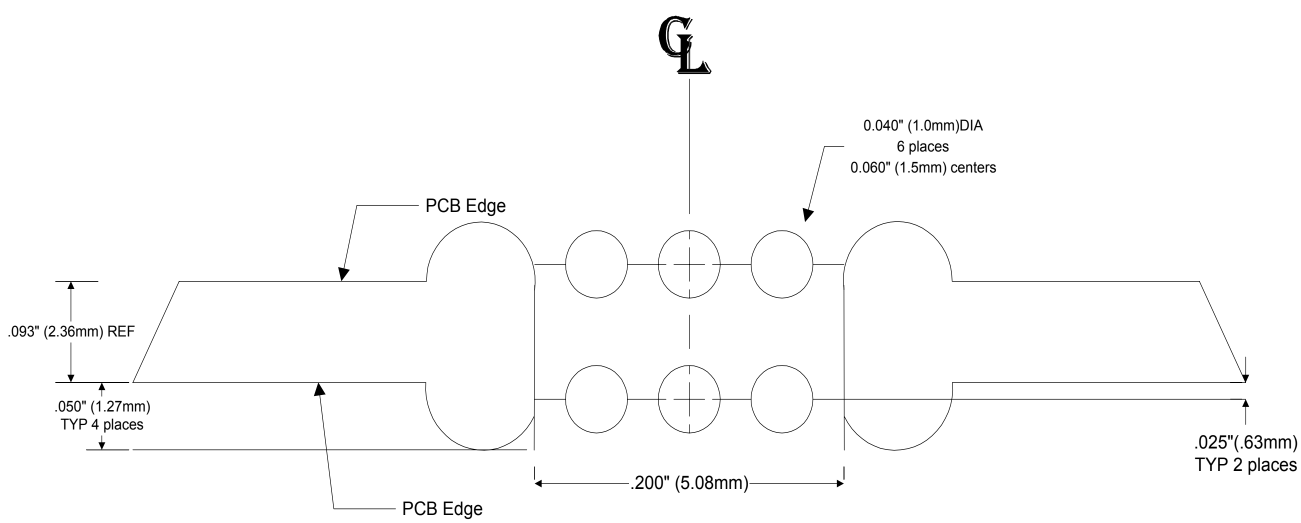


V-Groove Scoring

5.11.3.2 Tab/Routing

The routed slot and tab pattern is widely used for panel construction and break-away tab extensions. Routing is more precise than scoring, and edge surfaces are smooth, but the break-away "tab" points will require consideration. Tabs can be cut and ground flush with the board edge or pre-drilled in a pattern. The drilled pattern furnishes a low stress break point on the "tab". If the hole pattern is recessed within the board edge, secondary sanding or grinding can be bypassed. However drilled tabs are inherently less stable than solid tabs, and careful consideration should be given to the array size and area available per board to locate the tabs. Solid tabs due however require a secondary sanding or grinding processes that increases the labor content of the assembly. Typical residual material is .020"-.040". Sanding is not necessary if there are no clearance issues. The below diagram is for .062" thick boards only. Please consult South Bay Circuits, Inc. Engineering for recommendations on other fab thickness breakaways.





5.11.4 X-outs

X-outs are individual boards in a panel array that fail electrical test at the board fabricator. While not allowing x-outs will raise fab costs slightly, the impact to assembly is far greater. Panels with X-outs require special marking so that the placement equipment will skip them during the placement cycle. In some cases allowing x-outs is acceptable, with prior SBC Engineering authorization. Contact SBC engineering for further information regarding the use of X-outs.

6 Surface Mount Assembly

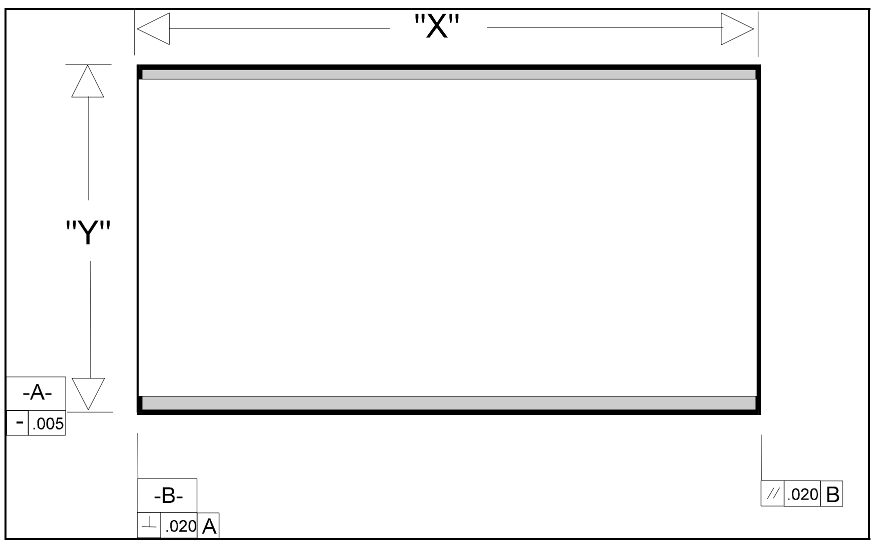
6.1 PCB Size/ Shape

SBC incorporates automated PCB handlers on all SMT lines. This produces a set of parameters that must be followed for proper conveyorized handling

1). Minimum PCB size – With the use of panelization, there really is no minimum board size constraint. Smaller boards can be panelized into a useable size array. The smallest array or individual board that can be handled is 4” X 4”

2). Maximum PCB size - Varies by line and facility. Please see Appendix “A” for details.

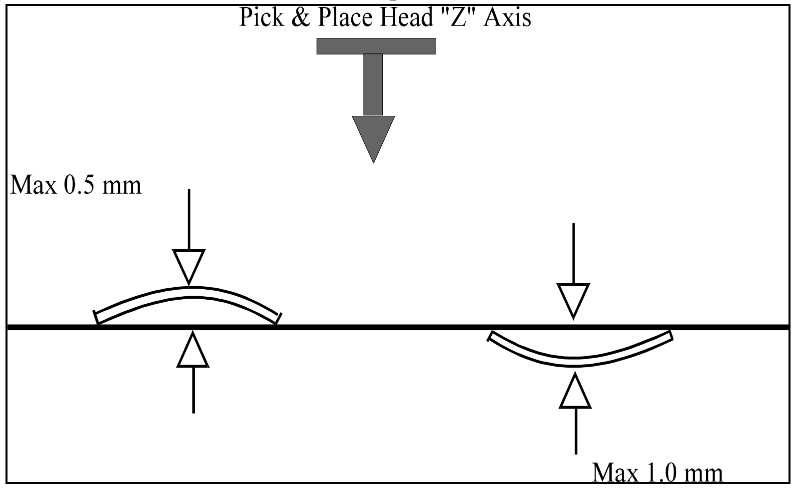
3). Edge Parallelism and Squareness



1. Straightness to be within 0.005” (0.127mm)
2. Parallelism with respect to datum A - 0.020” (0.5mm)
3. Parallelism with respect to datum B - 0.020” (0.5mm)
4. Perpendicularity with respect to datum A - 0.020” (0.5mm).

6.2 Bow and Twist

Bow is defined in IPC-T-50 as "The deformation from flatness of a board characterized by a roughly cylindrical or spherical curvature such that if the board is rectangular its four corners are in the same plane."

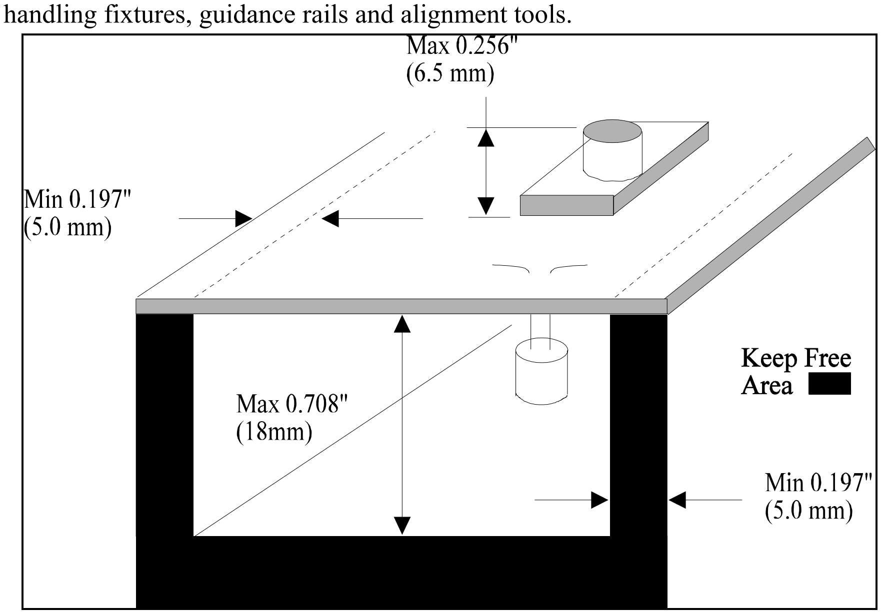


An "upward bow" greater than 0.020" (0.5mm) will cause the PCB to deflect downward as the device is being placed. When the pick and place head retracts the nozzle after placement the board will flex back to its original bowed state causing parts to be scattered across the PCB surface.

In the event that a "downward bow" exists, the opposite is true. The pick and place nozzle is extended to place the device and since the maximum extension is limited by a mechanically fixed limit, the device will be dropped onto the surface of the PCB. In most all cases the device is never near its placement location.

6.3 Board/Panel Clearances

Components can be mounted on individual boards or on boards that are still organized in panel form. Boards or panels that are to be handled by automatic board handling equipment or are to pass through automated parts placement, soldering, cleaning, etc., steps must have areas along the sides kept free of parts or active circuitry. Special tooling and fixturing holes are generally located within the edge clearance areas. The clearance areas are needed to avoid interference with board handling fixtures, guidance rails and alignment tools.



Typically a strip width of 0.118" (3mm) on the primary side and 0.197" (5mm) on the secondary side must be allowed along the transport sides for the clearance. The required clearance width is dependent upon the design of the board handling and fixturing equipment. The PCB design should be reviewed with Manufacturing Engineering to minimize the impact to production and to ensure the panelization and tooling requirements are understood early in the design phase. Failure to do so can cause lengthy delays with the introduction into manufacturing and may require costly tooling.

6.4 SMT Panelization guidelines

6.4.1 Breakaway rails

When required, they should be located on the longer axis of the array and should be .500” Rails should also include Fiducials and .125” tooling holes

6.4.2 Retention Method

This will be based on PCB shape, size, array number, and proximity of features to the edge of the board. See section 5.11 for more details.

6.5 Fiducials

6.5.1 Definition

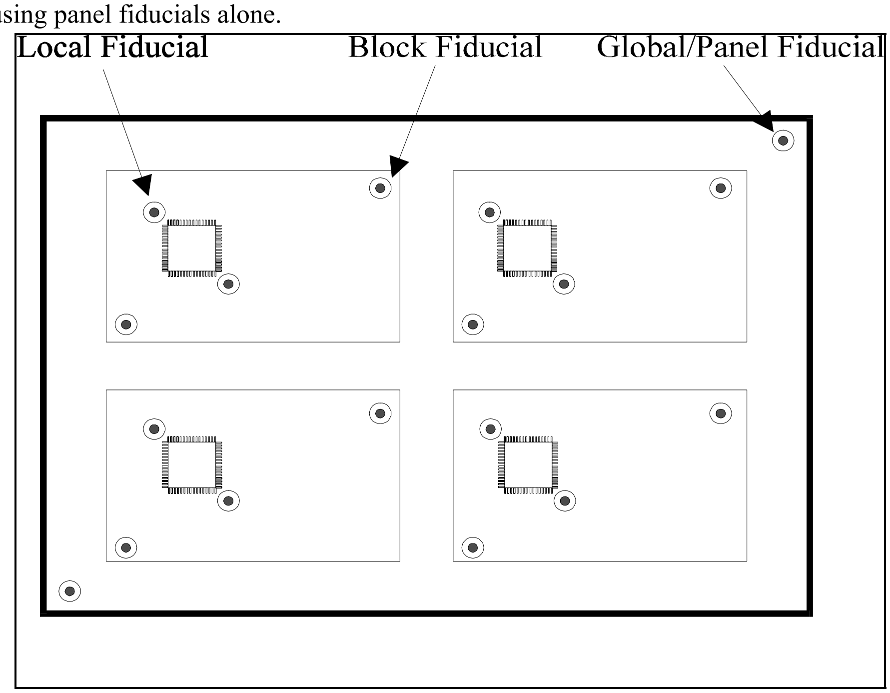
A Fiducial Mark is a printed artwork feature which is created in the same process as the circuit artwork. The fiducial and a circuit pattern artwork must be etched in the same fabrication step. Fiducial alignment is used to compensate for X, Y, and theta offsets in the position of the circuit pattern relative to the expected position. The fiducial alignment system is an opto-electronic system which performs geometric measurements of fiducial marks on the PCB in order to calculate the deviations from their expected positions. The system needs minimally two fiducials per board, or panel. For optimum correction, they should be diagonally opposed and as far apart as possible on the circuit or panel.

6.5.2 Global/Panel Fiducials

Fiducial marks that are used to locate the position of all features on an individual board or multi-image panel. In most cases a multi-up panel can be treated as one large circuit, and the panel fiducials are the only ones used for placement. This saves unnecessary fiducial recognition time and speeds machine placement.

6.5.3 Block Fiducials

Marks located on individual circuits within a multi-up panel. These may be used if accuracy problems are encountered using panel fiducials alone.

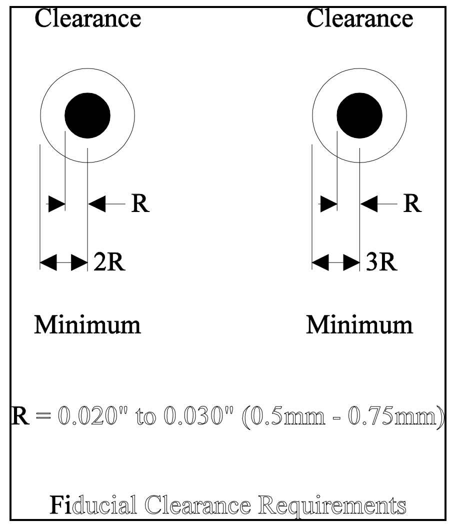


6.5.4 Local Fiducials

Marks that are used to locate individual components. These are primarily used for fine pitch devices, and may be located either diagonally along the outside perimeter of the component or centered within the body outline of the part. Due to increasingly tight design constraints, local fiducials are becoming harder to accommodate within a design. South Bay Circuits, Inc. recommends the use of local fiducials on components less than .5mm pitch.

6.5.5 Fiducial Mark Design Specifications

The Surface Mount Equipment Manufacturers Association (SMEMA) has standardized on design rules for fiducials. These rules are supported by the IPC and adopted by SBC and are consistent as to:



1). Shape: The optimum fiducial is a solid filled circle.

2). Size: The minimum diameter of the fiducial mark is 0.040” (1mm). The maximum diameter of the mark is 0.060” (1.5mm). 1 mm is preferred. Fiducial marks should not vary in size on the same PCB more than 0.001” (0.025mm)

3). Clearance: A clear area devoid of any other circuit features or markings shall exist around the fiducial mark. The size of the clear area shall be equal to the radius of the mark. A preferred clearance around the mark is equal to the mark diameter.

4). Material: The fiducial mark may be bare copper, gold, bare copper protected by a clear anti-oxidant, nickel or tin plated, or solder coated (HASL). The preferred thickness of plating or solder coating it 0.0002” to 0.0004” (0.0005 to 0.0010mm). Solder coating should never exceed 0.001” (0.025mm). If solder mask is used, it should never cover the fiducial mark or the clearance area. It should be noted that oxidation of a fiducial mark’s surface may degrade its readability.

5). Flatness: The flatness of the surface of the fiducial mark should be within 0.0006” (0.015mm).

6). Edge Clearance: The fiducial shall be located no closer to the PCB edge than the sum of 0.200” (5mm) (SMEMA Standard Transport Clearance) and the minimum fiducial clearance required.

7). Contrast: Best performance is achieved when high contrast is present between the fiducial mark and the PCB base material.

6.6 Component Placement

Although a wide variety of assembly types can be accommodated by South Bay Circuits, Inc., unnecessary labor costs resulting from additional processes can be avoided if assemblies are laid out with the following in mind.

6.6.1 Process Considerations

The following assemblies are listed in order from the least labor intensive to the most labor intensive.

1). Single Sided Assembly - All SMT and PTH components located on the same side of the board.
2). Double Sided Assembly – All SMT active devices and all PTH devices located on the topside, only SMT passives on the bottomside.
3). Double Sided Reflow – All SMT devices reflowed, and all PTH devices hand soldered.
4). Mixed Double Sided Assembly – SMT active devices and PTH devices located on both the top and bottomside of the assembly. This will require hand soldering and/or wave solder fixturing.

6.6.2 Reflow Considerations

- Avoid large components opposite each other on double sided boards.
- Avoid clustering large SMD's in one area as this will result in uneven heating during reflow
- Avoid fine pitch devices or BGA's on both sides of an assembly

6.6.3 Wave Solder Considerations

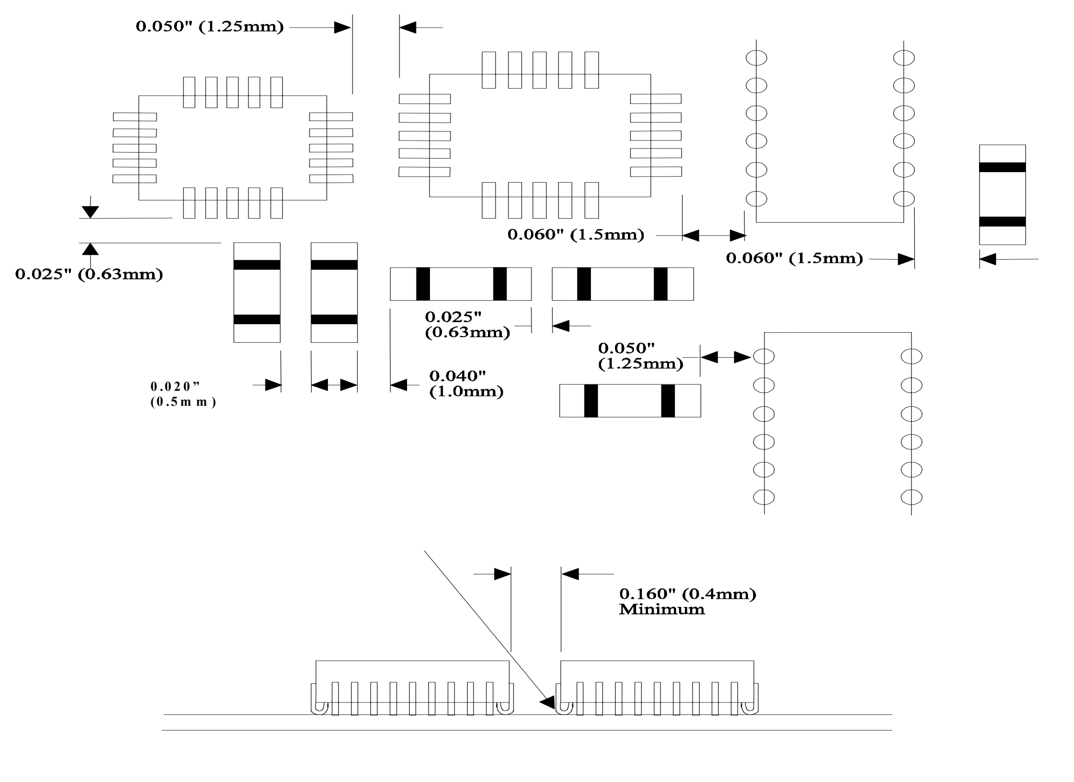
- No discretes or passives larger than an 1810 size package. This includes C size or large tantalum capacitors.
- No active components larger than 16 pin SOIC's
- See section 10 for proper wave solder component orientation guidelines.

6.6.4 Inspection Considerations

- Orient all polarity marking the same direction to facilitate identification of reversed components
- Keep all placement angles at 0/90/180/270

6.7 Component Spacing

Minimum component spacing specifications exist to ensure that assemblies will be manufacturable within an automated SMT process. Minimum spacing requirements also ensure that solder joints can be visually inspected, and if necessary, reworked or repaired.



Recommended Minimum Land-to-Land Clearances

6.8 Component Footprints

Properly designed component footprints are crucial to provide a manufacturable assembly, with minimum defects. Improperly designed footprints can lead to:

- Shorts
- Opens
- Solder Balls and Solder Beading
- Tombstoning
- Misalignments
- Insufficient Solder
- Decreased rework accessibility

6.8.1 Hierarchy of Pad Design References

1). IPC SM782 Surfacemount Design and Land Pattern Standard
2). Other industry accepted land pattern library such as SMT Plus
3). Manufacturers recommended pad layout.

6.8.2 Fine Pitch Layout

1). Soldermask dams are achievable down to .0197" pitch. Example - .0197" pitch - .011" pad width - .005" soldermask relief (.0025 each side) = .0037 soldermask dam.

2). Excessive joint length will not improve joint integrity, but will starve the solder joint by allowing solder to cover the pad and flow away from the joint.

6.8.3 BGA Layout

Ball Grid Arrays have gained significant popularity in recent years. The many benefits of using PBGA's over similar lead count leaded devices include:

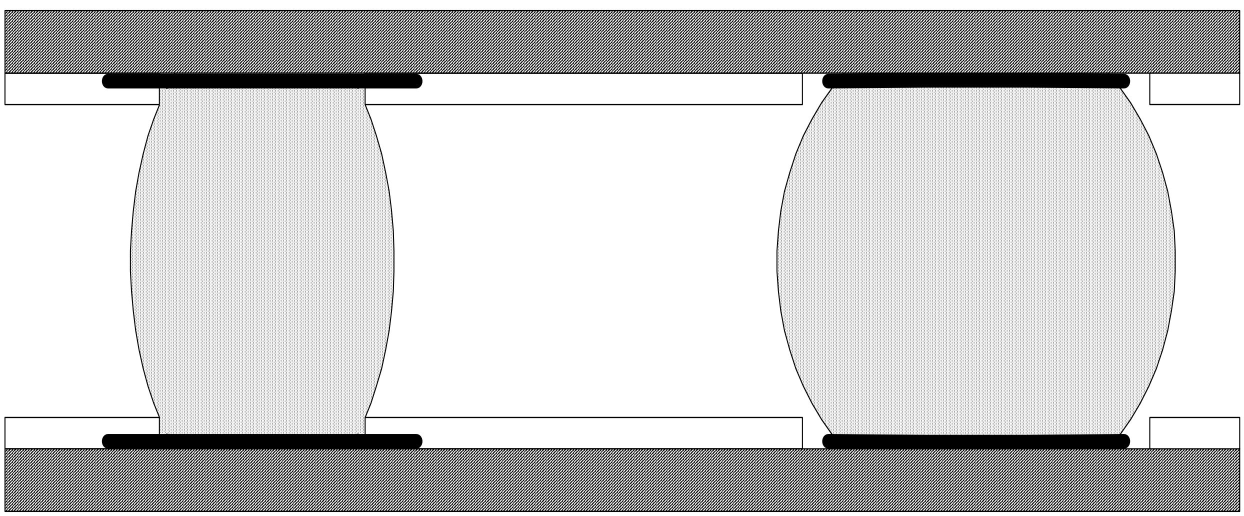
1). Board space efficiency
2). Excellent surface mount yields when compared to fine pitch leaded devices
3). Lower profile
4). Potential lower cost of ownership compared to leaded devices due to reduced scrap, and rework.

BGA's are by far a preferable package to high lead count, ultra fine pitch devices, if the following guidelines are followed with regard to pad geometry and escape routing.

6.8.3.1 Solder Pad Geometry

There are two basic types of solder pads commonly used with plastic ball grid arrays: copper or non-soldermask defined (NSMD) and soldermask defined (SMD). The NSMD is similar to a standard surface mount pad where there is clearance around the copper solder pad. The SMD type of pad has a larger copper than soldermask opening diameter. The NSMD pad type is recommended for most applications. It has the advantage that copper dimensions can typically be controlled more tightly than soldermask, and a HASL surface finish with better uniformity can usually be achieved. An additional advantage may be a lower stress concentration on the PBGA solder joint resulting in increased solder joint reliability. This advantage can only be realized when NSMD pads are used on both the PCB and the PBGA package. The SMD pad because of its greater copper area and soldermask overlap, has greater adhesion strength to the epoxy/glass laminate. This extra strength could be important in certain extreme bending and accelerated thermal cycling testing, where the pad to PCB adhesion is the weak link.

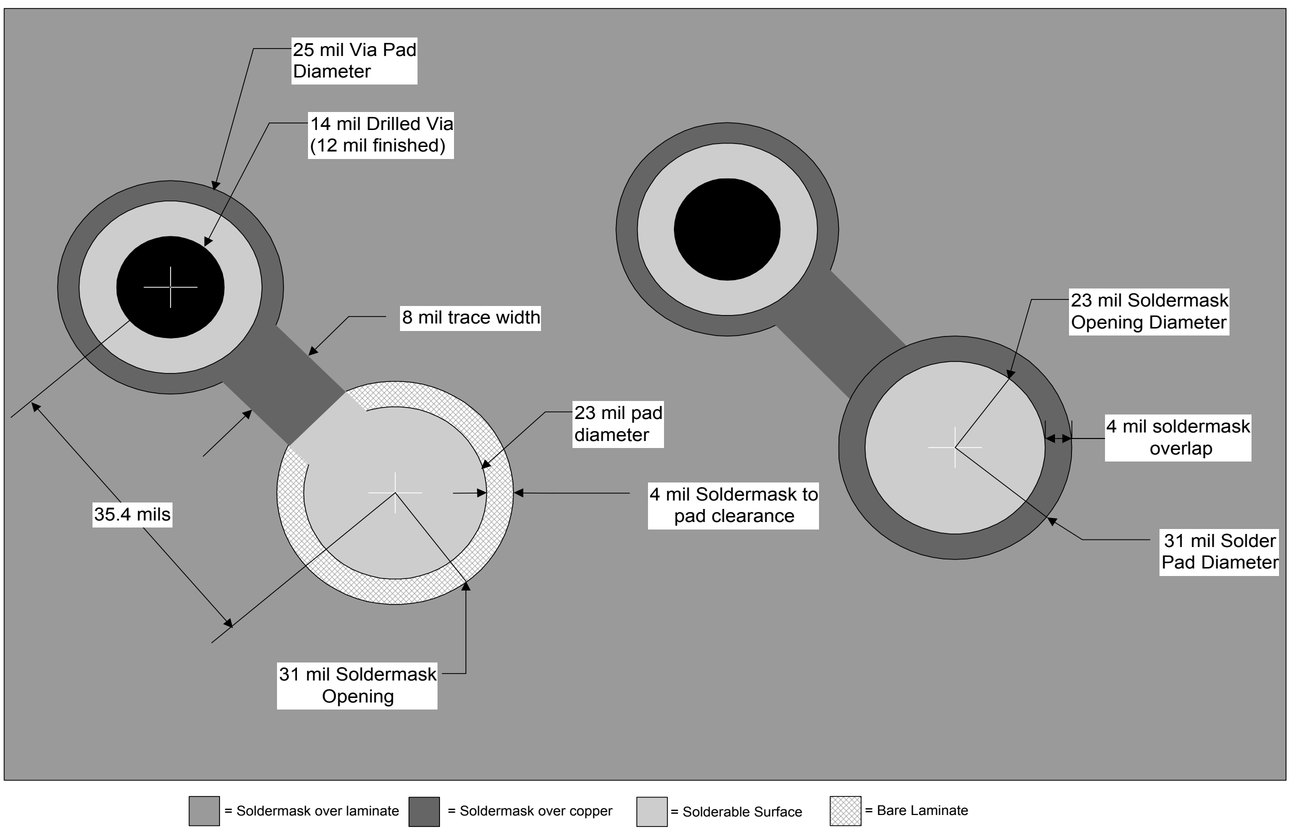
For either pad design, the goal is to have the pad diameter equal to that on the package.



Soldermask Defined **Nonsoldermask Defined**

The above drawing represents a comparison of PBGA solder joints with soldermask defined and conventional non soldermask defined pads. Note the greater solder volume and greater effective joint diameter for the non soldermask defined pad to achieve the same stand-off height.

The following diagram is an example of a 1.27mm pitch PBGA containing a 23 mil pad. In this figure, the via, via pad, and via soldermask clearance shown are recommended, but some variation is permissible. It is recommended that the width of the trace attached should not be much greater than the 8 mils shown. A trace much wider, will cause the pad to begin to resemble a mixture between NSMD and SMD. Also, a fillet should be present where the trace joins the solder pad, and no more than one trace should be joined to any NSMD pad.



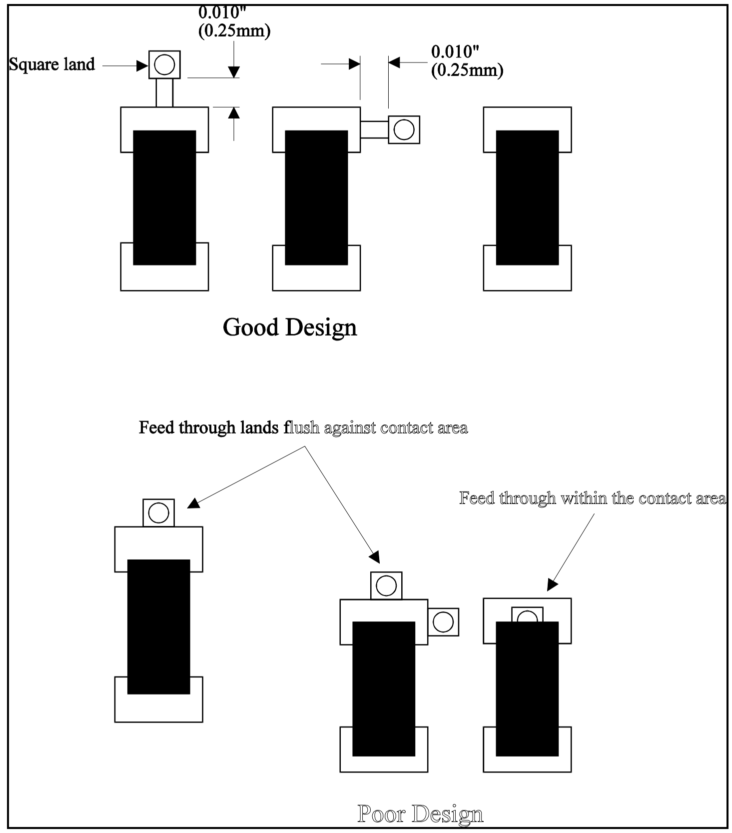
6.9 Vias

Size of the via holes should be selected based upon the PCB thickness vs. hole diameter or aspect ratio limits as defined by the PCB fabricator. Specific via lands and holes can also be accessed for automatic in-circuit test (ICT).

6.9.1 Vias within the Component Land Pattern

It is important to note that layout of the vias with respect to the land pattern on the PCB is important to good solder joint design. When the vias are located directly against or within the contact area of the land, solder thieving occurs during the reflow process, leaving insufficient/no solder fillets on the land and component. This condition can be prevented by providing a narrow bridge between the land and the via, therefore it is not recommended that via holes be specified as part of the component land pattern for any reason.

Specifying tented or filled vias will also reduce the propensity for solder migration on assemblies that utilize solder reflow processes. Typically, tenting is accomplished using dry film type soldermask unless the via is filled with a resin prior to liquid photoimageable (LPI) mask application. LPI masking is the preferred soldermask material for SBC's SMT manufacturing process due to its ability to be applied thinner than the dry film.



6.9.2 Vias Under Components

Via holes may be placed under SMD's (Surface Mount Devices) if they are to be solder reflowed. However, if the PCB is to be wave soldered (through hole devices on the primary side of the substrate),via holes underneath SMD's should be avoided or "tented" with solder mask, as solder may wick up the via and short to adjacent pads.

6.10 Advanced Technologies

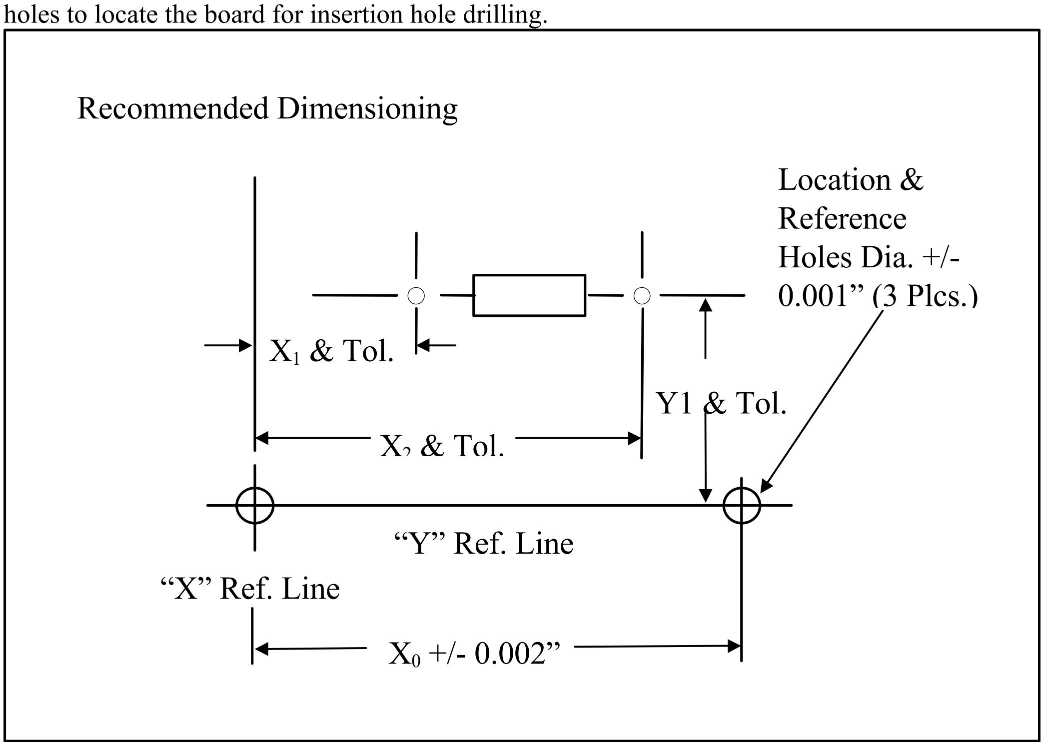
South Bay Circuits, Inc.'s Manufacturing Engineering Department is continuously developing advanced technologies and processes in keeping with those found in the Electronics assembly industry. See Appendix "A" for details.

7 Automated Through Hole Assembly

7.1 Insertion Hole Locations

The locations of all insertion holes should be defined relative to the board locating references. The accuracy of the hole locations with respect to the locating references will affect insertion reliability and hole diameter requirements. On drilled PCB's, the hole tolerance from the board references should be ±0.003" (0.076mm) or less.

Care must be exercised in the initial board layout to ensure that the insertion hole locations and the board locating references are only "one tolerance" apart. If this consideration is not met, tolerance build-up during insertion will reduce insertion reliability. The "one tolerance" requirement can easily be accomplished by establishing X-Axis and Y-Axis dimensional reference lines through the centerlines of the locating references and using these pre-drilled holes to locate the board for insertion hole drilling.



7.2 Component Location Objectives

The most efficient use of automatic insertion equipment and the greatest component population density can be achieved by inserting components in only one or two orthogonal axes. Use of only one axis minimizes the amount of board handling and machine operations. Two axes insertion is an acceptable and efficient way of inserting components. South Bay Circuits, Inc.'s insertion equipment utilizes automatic rotary tables to accomplish two axes insertion. However, the use of the rotary table contributes to a tolerance build-up which can lower insertion reliability. Applications which use more than two axes are generally inefficient and difficult to implement and are discouraged for use at South Bay Circuits, Inc..

7.3 Component Selection

Insertion machines allow a wide range of components to be inserted with minimum tooling changes. Careful component selection can provide improved automatic insertion reliability. It is wise to limit the range of component body sizes and lead diameters used within a given design application. Doing this will decrease the need for editing programs and reduce time consuming tooling changes.

7.4 Lead Hole Diameter Considerations

Each general type of component insertion has its own rule for establishing hole diameter. Hole diameter is basically a function of the machine, board accuracy, and lead diameter. For higher yield rates, it must be noted that component tolerances, board tolerances, and hole diameters play an important role. Pushing any or all of these to their limits will lower yields.

While some hole diameters may seem somewhat large in size, they take into account typical production floor conditions. They deal with nominal dimensions plus the tolerances of all elements of the insertion system regardless of type. It can be said that components can be inserted into smaller holes. It is difficult to accomplish it in a production environment. Taking this into account, the hole sizes established in these guidelines can generally be used without special considerations and with excellent results for production yield.

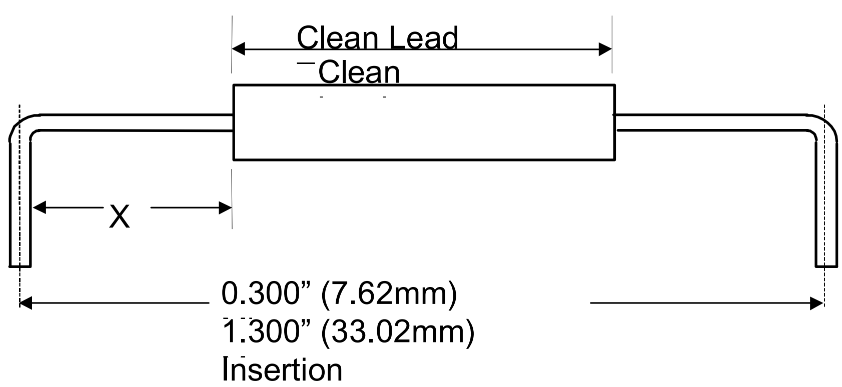
7.5 Location Considerations

Automatic insertion of PCB's requires that there be sufficient clearances around components and locating references to allow room for the insertion head and the cut & clinch tooling. The clearances required will vary with the type of insertion machine.

8 Axial Lead Component Insertion

8.1 Lead Form and Tooling

The VCD Head is tooled for the lead form as shown below/ The "X" dimension (end of component body to inside of form in lead) given is a function of the Driver Tips and Inside Formers (head tooling) selected. Typically, this is about 0.080" (2mm) minimum. Larger values of "X" will give more reliable insertion. With special attention to component selection and body centering during sequencing, the 0.075" (1.91mm) dimension can be used as the minimum.



Note: A minimum "X" dimension of 0.075" (1.91mm) is possible for components with lead diameters up to 0.032" (0.81mm), providing the configuration allows normal centering during re-taping. For example, some coated components require more attention during

8.2 Hole Diameter Requirements

Hole sizes necessary for reliable component insertion is a function of the following:

1). Component lead diameter
2). Machine insertion Head tolerances
3). Table positioning accuracy
4). Board holder accuracy
5). PCB hole pattern and tooling references accuracy
6). Component lead protrusion (through the board)

For Axial Lead Insertion machines with rotary tables, the maximum machine tolerances for the head, X - Y table, rotary table and workboard holder is a true position accuracy of 0.005” (0.12mm). This value requires a starting hole diameter of 0.010” (0.25mm) larger than the lead diameter being inserted. The manufacturing tolerance of the PCB insertion hole pattern must be added top this hole diameter. The formula for determining hole diameter is:

Minimum Hole Diameter = Lead Diameter + 0.010” (0.25mm) + Hole Location Tolerance

The formula normally produces insertion reliability of 99.9 percent (0.999). Reduced clearance from wire diameter to hole diameter to as low as 0.005” (0.13mm), may be used with small, high accuracy boards and still maintain reasonable successful insertion. The insertion reliability will be reduced.

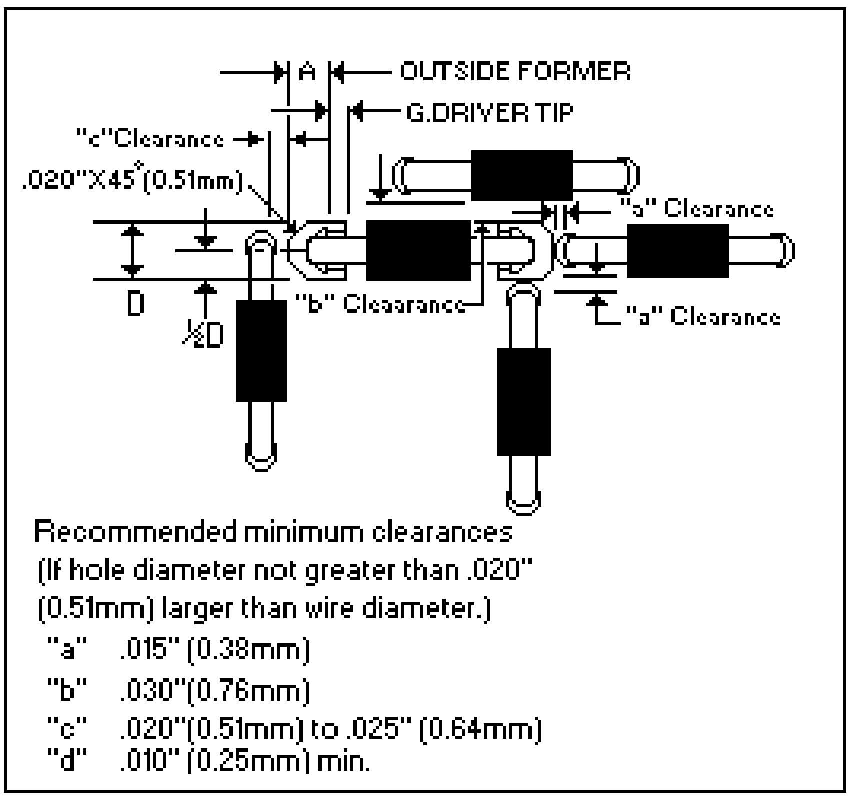
8.3 Component Body Configuration

Normally body configuration is not critical unless the extreme limitations of the equipment are being used. The components are handled by the leads so that component bodies may be round, oval, dog boned in shape, etc.

8.4 Location Considerations

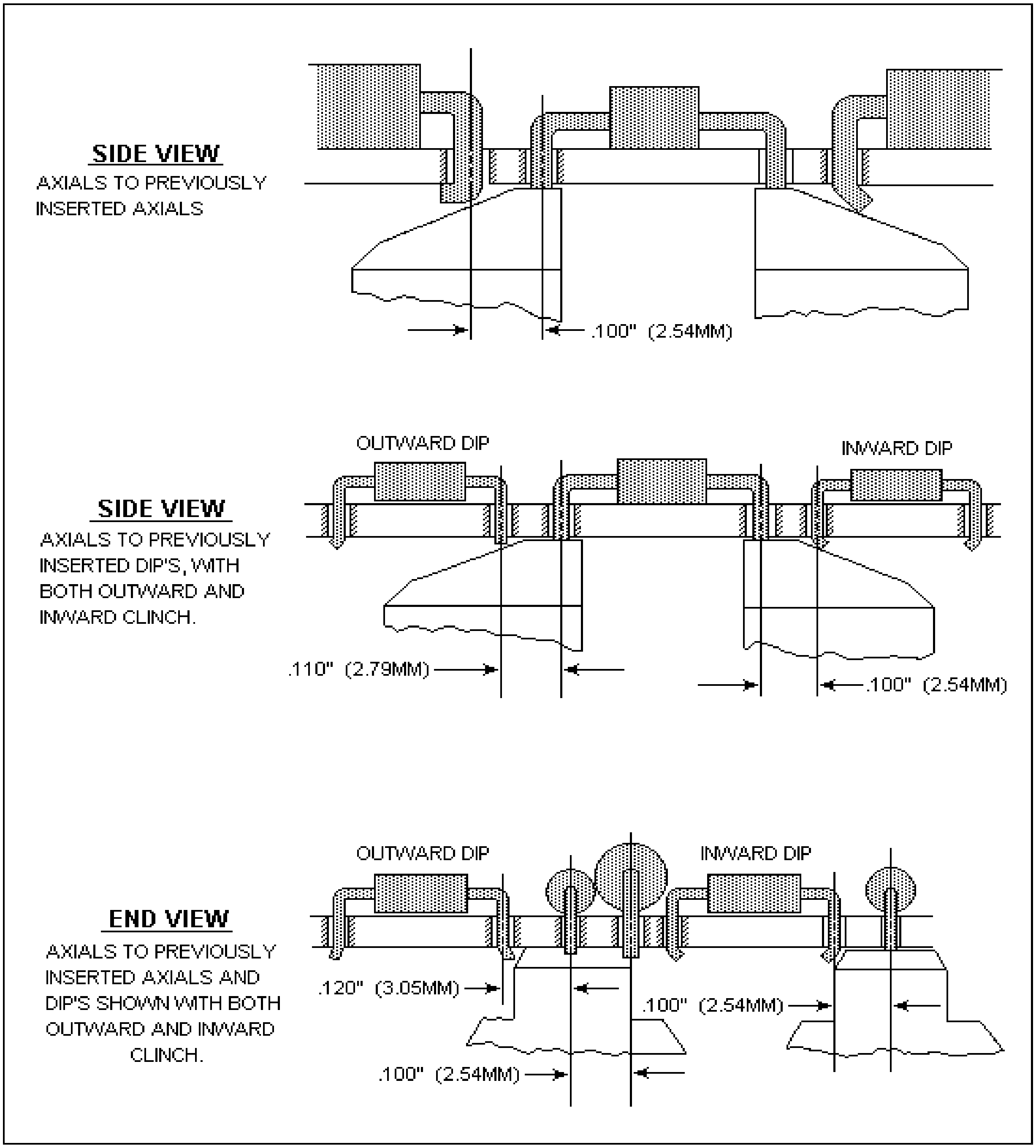
8.4.1 Above the Board

Axial Lead Insertion Heads are equipped with Outside Formers to guide leads to the point of insertion onto the PCB. Clearance around a given hole must be taken into considered to allow proper equipment function. The adjacent figure shows a top view of a standard Outside Former at the point of insertion, illustrating the clearances required between the lead being inserted and any adjacent component body or lead.



8.4.2 Below the Board

Below the board the cut and clinch is normally positioned 0.600” (15.4mm) to clear the workboard holder and rotary disk. When the insertion cycle begins, the Cut and Clinch Anvil is raised to 0.005” to 0.010” (0.12mm to 0.25mm) below the board to support it and provide lead clearances. At the bottom of the Head stroke, Cutters are driven upward to cut and clinch leads. See the figure below for minimum side-to-side and end-to-end clearances for Axial Lead Cut and Clinch to previously inserted axials / DIPs.

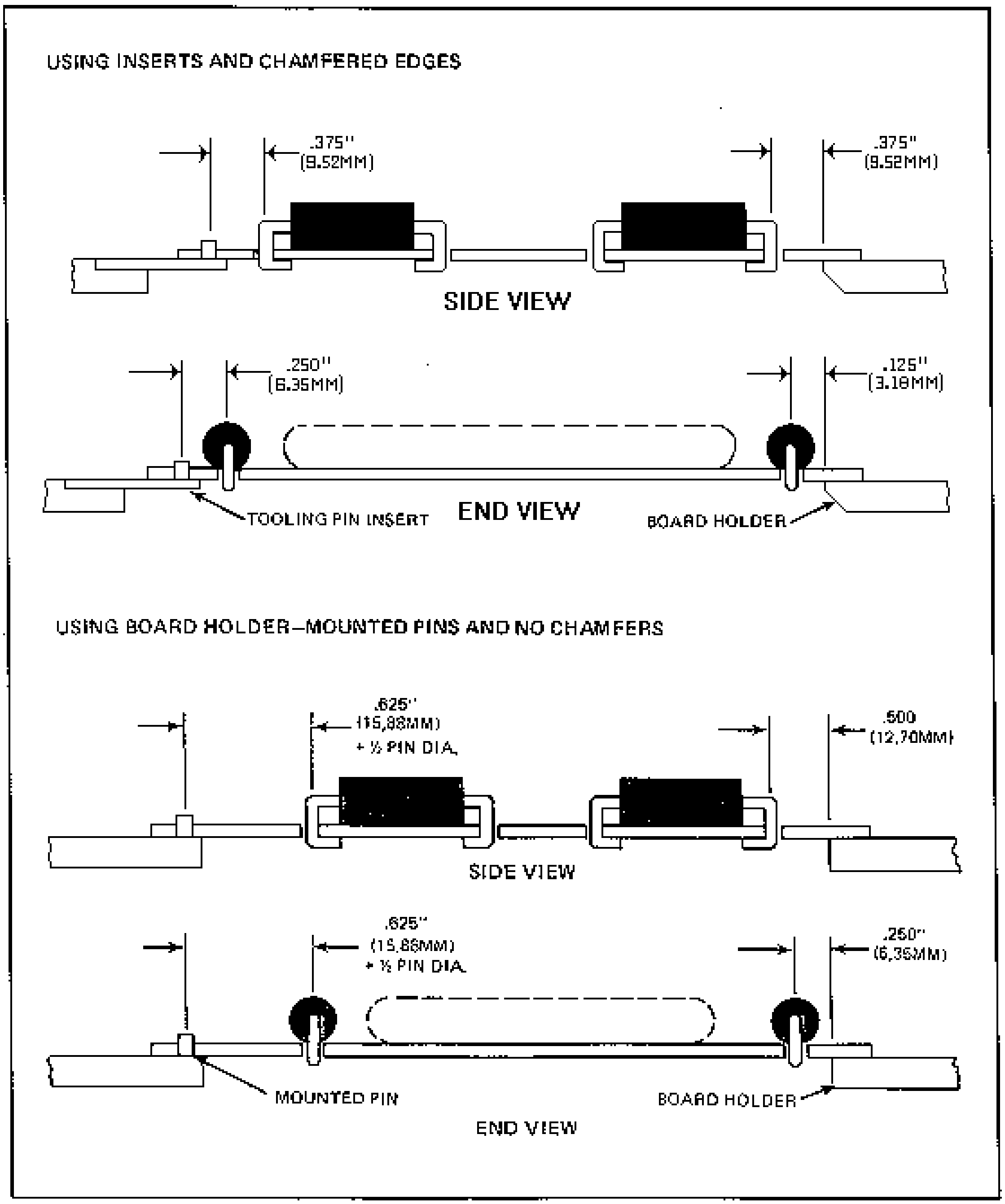


8.4.3 Uninsertable Area

The various methods used to locate boards for insertion exclude mounting of components, in most cases, in areas around reference points. This varies greatly between different board configurations; however, various techniques can be used to reduce this to an acceptable minimum.

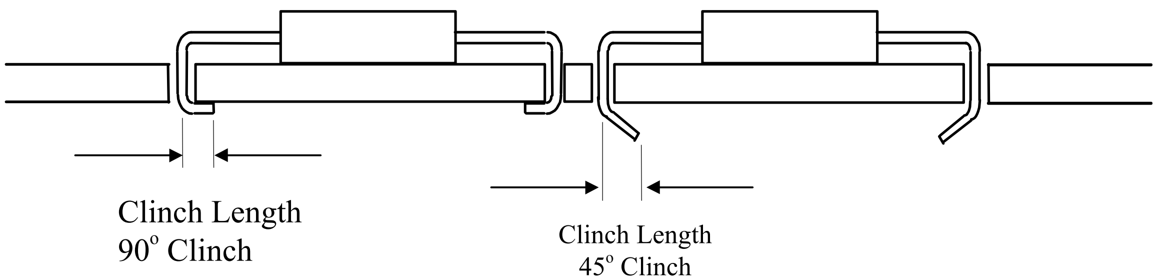
As a general rule, the minimum uninsertable area in the vicinity of any reference hole is approximately 0.5" (12.7mm) radius from the edge of the workboard holder which supports the locating pin. Edge locating and support methods usually require 0.5" (12.7mm) along the edge guides. With the use of inserts, these requirements can be reduced. See the figure below.

UNINSERTABLE AREAS — CLINCH TO WORKBOARD HOLDER, FOR VCD INSERTION MACHINES



8.4.4 Clinch Patterns

Lead forming for axial and other similar components is along the centerline of the component, parallel to the component leads, and inward toward the body. Although the component is cut from the tape by the Insertion Head before insertion, a second and more precise cut occurs beneath the board. The minimum length of the lead under the board is a function of lead diameter. Lead length is measured parallel to the board after the clinch. See the figure below.



WIRE DIAMETER	MAXIMUM CLINCH	MINIMUM CLINCH
90° CLINCH		
0.020” (0.51mm)	0.030” (0.76mm)	0.060” (1.52mm)
0.030” (0.76mm)	0.040” (1.02mm)	0.060” (1.52mm)
0.040” (1.02mm)	0.050” (1.27mm)	0.060” (1.52mm)
0.050” (1.27mm)	0.060” (1.52mm)	0.060” (1.52mm)
45° CLINCH		
0.020” (0.51mm)	0.020” (0.51mm)	0.042” (1.07mm)
0.030” (0.76mm)	0.028” (0.71mm)	0.042” (1.07mm)
0.040” (1.02mm)	0.035” (0.89mm)	0.042” (1.07mm)
0.050” (1.27mm)	0.042” (1.07mm)	0.042” (1.07mm)

8.4.5 Clinch Length

The clinch length is measured from the centerline of the lead as it extends through the hole in the PCB. Minimum clinch lengths for smaller wire diameters are somewhat dependent on the hole diameter in the board. To maintain these minimum clinch lengths, hole size must not be more than 0.015” (0.38mm) larger than the lead diameter. Clinch length is adjusted by changing anvil span relative to head span.

8.4.6 Clinch Repeatability

Once the Clinch Unit is set for a lead length, the repeatability is ± 0.005” (0.13mm) or less depending on the lead material consistency, with all other insertion parameters remaining constant.

9 DIP Component Insertion

General

This section has been prepared to assist in the design of printed circuit boards for use with Automatic DIP Insertion Machines. However, the data is generally also usable on other automatic, semi-automatic, and manual insertion operations.

The Automatic DIP Insertion Machines are capable of processing a combination of two of the following devices:

1). 0.300” (7.62mm) lead span: 6 to 20 leads
2). 0.600” (15.24mm) lead span: 22 to 40 leads

3). 2 and 4 lead DIP devices with 0.300” (7.62mm) lead span

9.1 Component Input Specifications

DIP Automatic Insertion machines can be tooled to insert DIP components with lead spans of 0.300” to 0.600”. Components for input to the machines are loaded in plastic or metal DIP sticks or tubes and are placed into the Input Magazines of the machine.

The Automatic DIP Insertion Machines can also be tooled to insert DIP sockets, with exception to 2 and 4 leaded devices, which meet the specification shown in Appendix 6.6 and which are loaded in carrier sticks as shown in

Optional tooling, permits longer devices to be automatically processed by Automatic DIP Insertion Machines, but unless otherwise specified herein all dimensions, clearances, and references shown apply to the devices listed in the General Section.

The guidelines stated herein will provide optimum reliability in the production environment. As stated previously, more generous clearances will improve insertion reliability; pushing dimensions to the lower limits will reduce insertion reliability. Other configurations and dimensions are often possible. Consult your Manufacturing Engineer for any deviations.

9.2 Hole Diameter Considerations

When determining lead hole diameters, two primary factors must be considered: the first being that the holes are large enough to consistently accept component lead insertion; the second being that the holes are small enough to assure a secure lead clinch.

The minimum component lead hole size required for reliable DIP component insertion is a function of:

1). Component lead cross-sectional area and lead-end configuration.
2). Machine tolerance and accuracy.
3). Workboard holder accuracy.
4). Hole location tolerance.

The accumulated tolerances of the insertion tooling, Positioning System and the design of the Workboard Holder provides a positioning accuracy of ±0.005” (0.12mm), relative to true position, for a machine that is equipped with a standard, rotary positioning system. In order to determine the minimum acceptable lead hole diameter, this positioning accuracy dictates a starting diameter of 0.010” (0.25mm) larger than the Effective Lead Diameter of the component to be inserted. To this must be added the manufacturing tolerance of the PCB hole pattern (Hole Location Tolerance).

1). This is expressed as: Effective Lead Diameter + Hole Location Tolerance + 0.010”

2). Effective Lead Diameter is a function of the tooling lead slot [0.022” (0.056mm)] and component lead thickness where: $\text{Effective Lead Diameter} = \sqrt{(0.022)^2 + (\text{lead thickness})^2}$

3). The minimum component hole diameter should not be less than 0.037” (0.94mm), except as described in “Component Lead Considerations” for pointed or tapered leads, nor should the maximum hole diameter exceed 0.041” (1.04mm).

4). Whenever insertion into PCB holes smaller than those specified herein, the Manufacturing Engineer should be consulted.

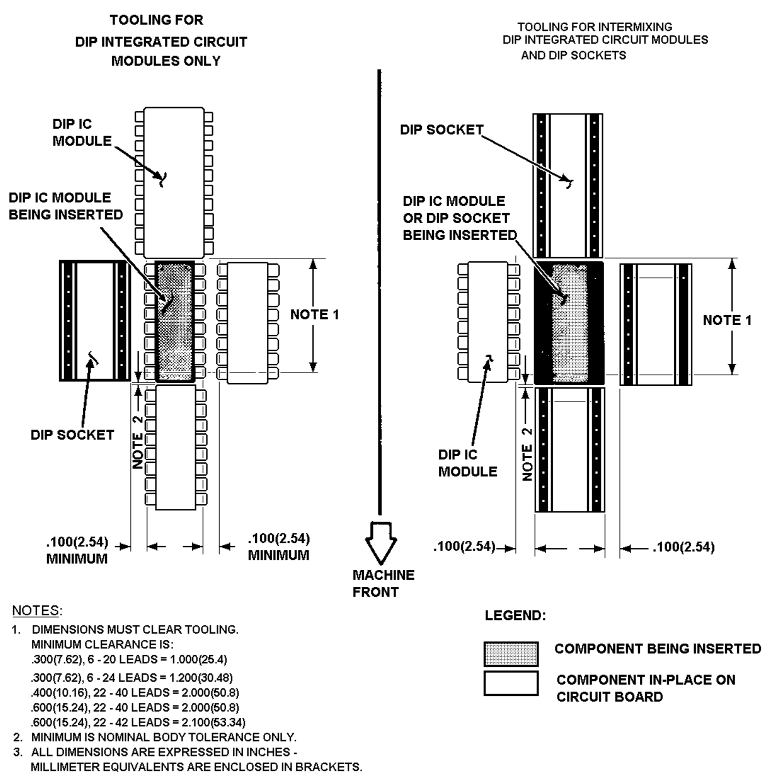
9.3 Component Lead Considerations

Insertion reliability for DIP components is also influenced by the condition of the component leads as well as by the lead tip configuration. Sharp bends in the leads will prevent their being inserted. Leads terminated in a point will enhance insertion reliability and can effectively reduce the required hole diameter by 0.003" (0.08mm) to 0.006" (0.15mm) depending upon the degree of taper, length of taper, lead material, and PCB hole plating.

9.4 Component Mix

Ideally, the best production rates are obtainable when using components all of which have identical physical characteristics and tolerances. Because this is not always possible, it must be noted that the greater the range and the greater the variety of deviations there are between the component types being processed, the greater the probability that stated insertion rates and reliability will be adversely effected. Where either or both of these conditions are anticipated, the Manufacturing Engineer should be consulted to assure that the best ratio between component mix and productivity / reliability is obtained.

COMPONENT-TO-COMPONENT CLEARANCE FOR DIP INSERTION TOOLING



9.5 Location Considerations

9.5.1 Above the Board

In general, where axial lead components and DIP components are intermixed on the same board, DIP components should usually be inserted first and the spacing rules for axial leads should be applied, provided the axial lead component height is not greater than 0.250" (6.35mm) at the point of clearance.

9.5.2 Below the Board

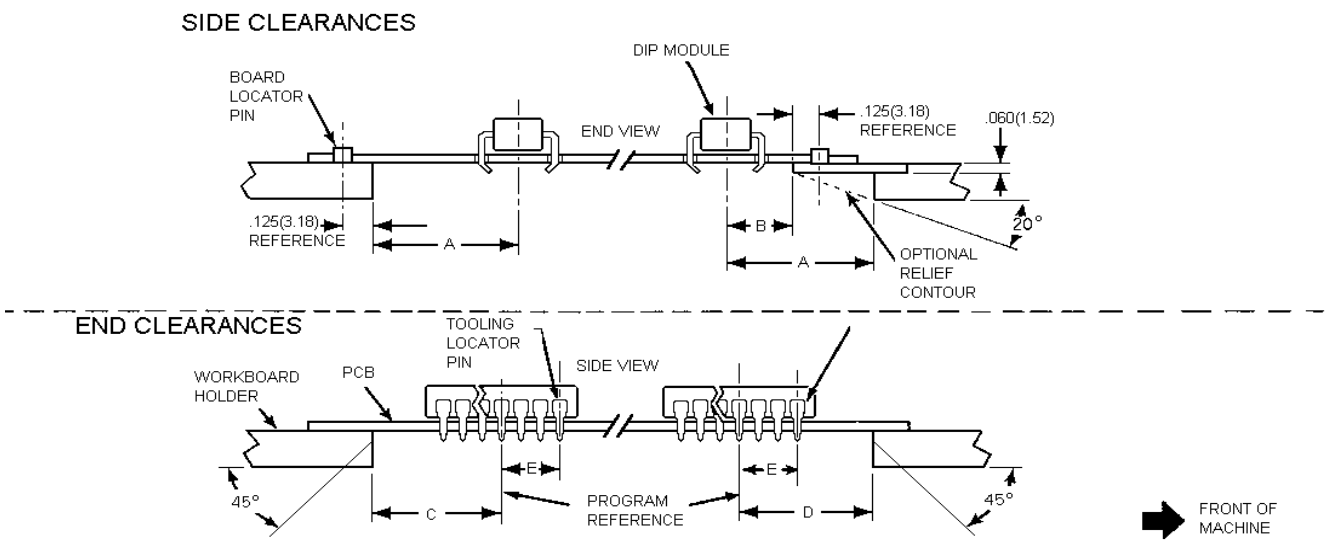
The clinch down position is below the work board holder providing clearance for positioning movement. As the Insertion Head begins the insertion cycle, the Cut and Clinch Anvil is raised to support the PCB during the component lead insertion. At the bottom of the head stroke and after the insertion plunger is extended, the cutters are driven in or out, depending upon pattern program selection, to cut and clinch the component leads. The workboard holder edges may not be positioned so as to obstruct the operation of the Cut and Clinch Assembly.

9.5.3 Uninsertable Area

The various methods used to locate PCB's on the workboard holder generally do no permit components to be located in areas around board reference points. Uninsertable area varies greatly between the different workboard holder configurations, however, various design techniques can be used to reduce this uninsertable area to an acceptable minimum.

As a general rule, the minimum uninsertable in the vicinity of any reference hole location is approximately a 1.0" (25mm) radius. Edge locating and support methods usually require a minimum of 1.0" (25mm) clearance along the guide edges. Use of inserts can reduce these requirements, when this condition is anticipated, the Manufacturing Engineer should be consulted to assure that optimum results are obtained.

UNINSERTABLE AREAS



DIP LEAD INSERT SPAN	DIMENSIONS						
	A	B	WITHOUT 45° RELIEF C	WITHOUT 45° RELIEF D	WITH 45° RELIEF C	WITH 45° RELIEF D	E
.300(7.62)	1.550(29.37)	.940(23.88)	.980(24.89)	.680(17.27)	.744(18.9)	.444(11.28)	.300(7.62)
.400(10.16)	1.600(40.64)	.990(25.15)	1.480(37.59)	1.180(29.97)	1.244(31.6)	.944(23.98)	.800(20.32)
.600(15.24)	1.700(43.18)	1.090(27.69	1.480(37.59)	1.180(29.97)	1.244(31.6)	.944(23.98)	.800(20.32)

ALL DIMENSIONS ARE EXPRESSED IN INCHES.
METRIC EQUIVALENTS ARE BRACKETED.

9.5.4 Clinch Patterns

Multi-Center Cut and Clinch (Inward)
The Multi-Center Cut and Clinch Unit will cut all leads from the inserted DIP component.

Multi-Center Cut and Clinch (Outward)
When used in the outward clinch configuration, the Multi-Center Cut and Clinch Unit will trim and clinch all leads. The outward clinch configuration tends to provide better component seating by taking advantage of the natural outward spring of the leads. It also permits easier removal of components from the PCB in the event replacement is required.

The outward clinch pattern is not recommended for DIP socket component insertion.

10 Press Fit Component Consideration

General – Most mid plane and back plane designs utilize "press fit" connectors when interfacing with the PCB. This cost effective design allows connectors to be placed on both sides of the board, without the added labor of hand soldering. The following guidelines should be followed when designs incorporate "press fit" connectors.

10.1 Hole Size

Hole size is critical in creating a successful press fit interconnection. A drill size of .0453" with a finished size of .040"+.002" - .001" must be used.

10.2 Component Clearances

Press fit components require back up support fixtures for proper insertion. Adjacent components should be kept a minimum of .100" away from all press fit components on both sides of the board.

11 Wave Solder

General – Designing for Wave Solder is a critical step in producing a lower cost assembly. Poorly designed panels will result in higher defect rates and/or increased labor costs due to increased hand soldering and wave solder fixturing.

11.1 PCB Size

Minimum board size is 4" X 4", Maximum board size is 18"W X 20 "L

11.2 Clearances

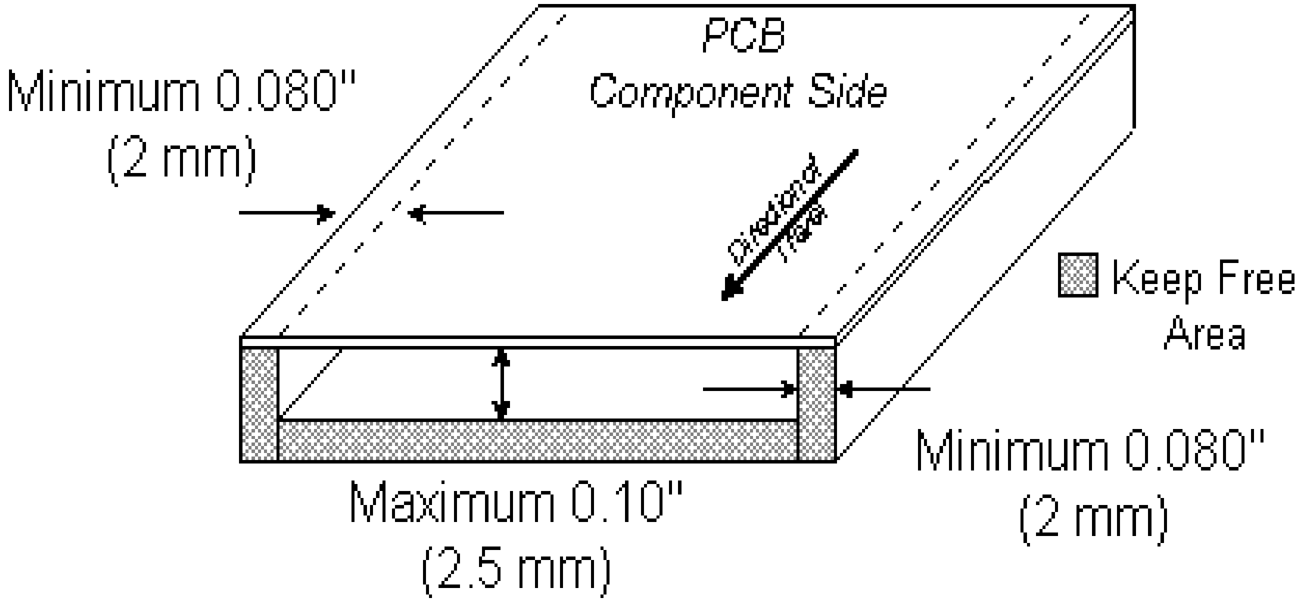
Panels that meet the following criteria may be wave soldered without the use of additional tooling:

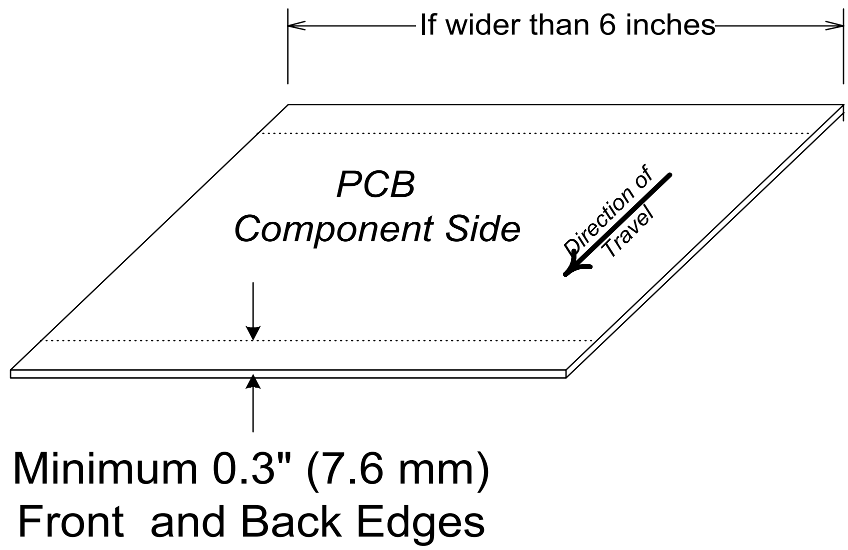
1). The PCB/Panel must be rectangular, with 2 parallel sides along the longer axis

2). These parallel sides must have a minimum of .080" (2mm) clearance to any component leads or bodies.

3). Under board clearance for leads or previously mounted SMT components must not exceed .100"

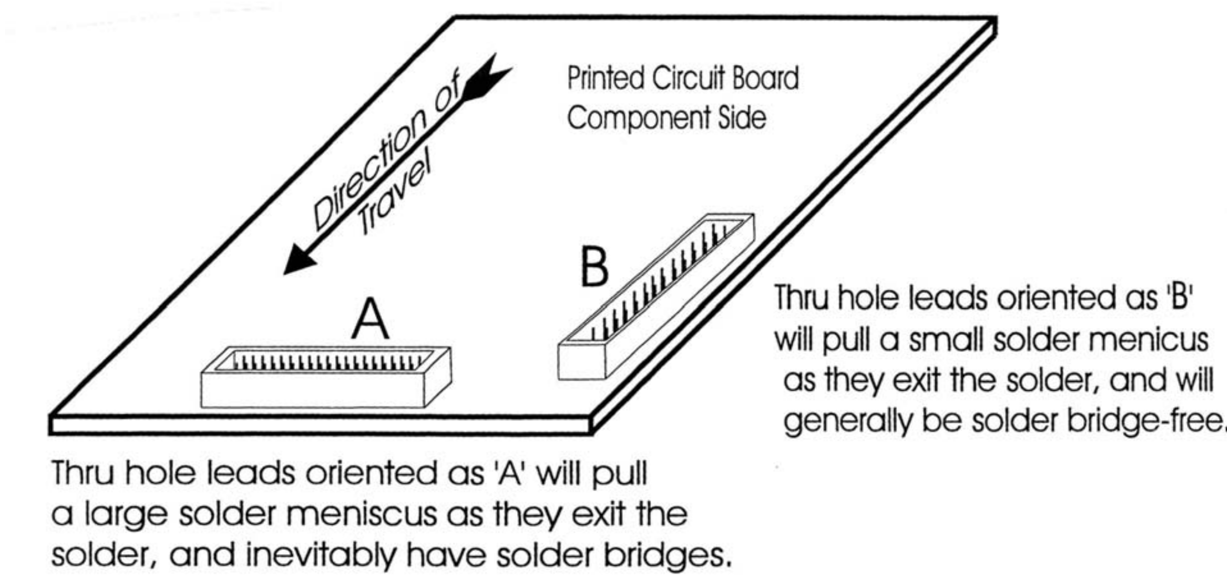
4). PCB's/Panels exceeding 6" in width, must have additional keep out areas of at least 0.3" (7.6mm) along the front and rear edges to accommodate PCB stiffeners



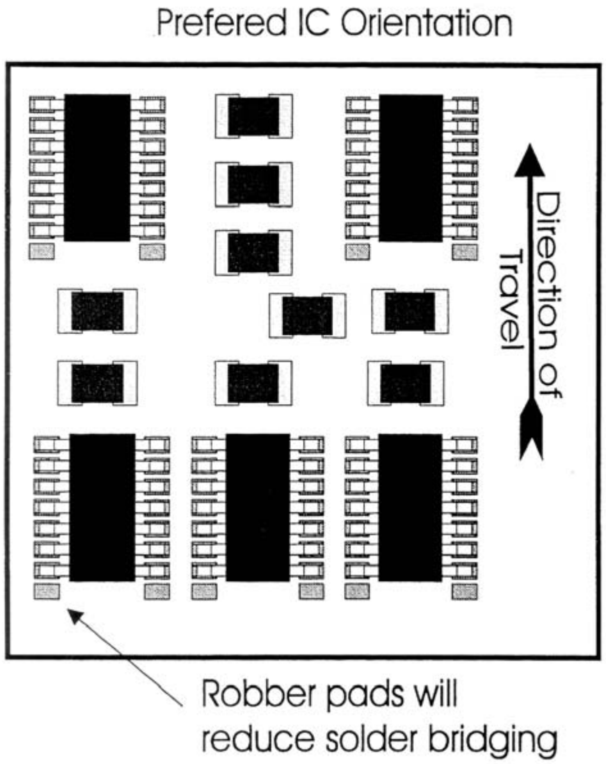


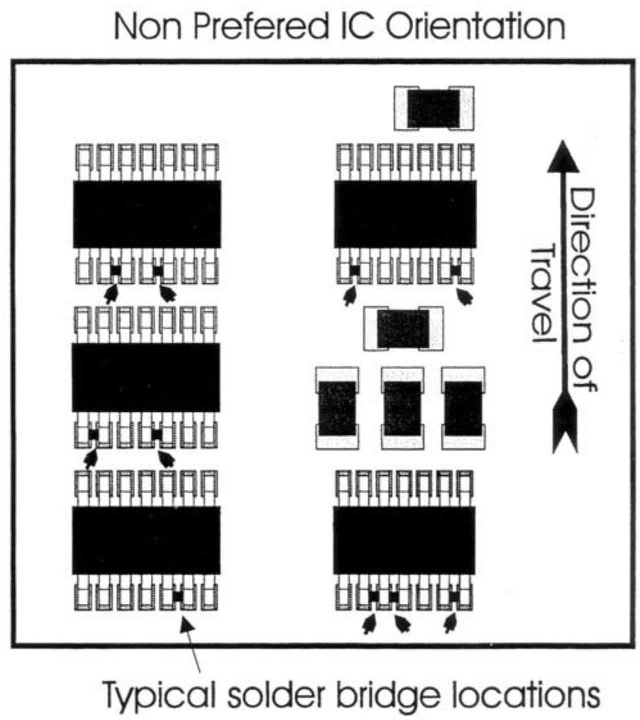
11.3 Component Orientation

11.3.1 DIP/Connectors



11.3.2 Surface Mount





11.4 Component Footprint Design

11.4.1 Solder thieves

Solder Thieves are additional pads located at the trailing edge of IC's as they pass over the wave. These additional pads reduce bridging between pins of the component. These pads should be located on the same center to center pitch as the component pads. They should be the same length , but double the width of the component pad. (See drawing above)

11.4.2 Vias

All non test point vias should be masked to minimize solder migration to the topside of the board. Spacing guidelines should be the same as for SMT.

11.4.3 Pad Sizes

Pads should be large enough to allow proper fillet formation and access for rework and repair. In some cases, IPC guidelines will callout a slightly different pad layout for wave solder. Consult the IPC-SM-782 for details.

11.4.4 Passive components

Passive components whose terminations do not hit the wave simultaneously will be subject to thermal shock and potential component failure.

11.5 Non-waveable Components

- Passives, especially ceramics, larger than 1810 size package
- Tantalum Capacitors
- PLCC's
- QFP's
- Fine Pitch SOIC's
- Parts that cannot exceed 250 degrees Celsius
- Parts that cannot be exposed to an aqueous cleaning cycle

11.6 Wave Solder Pallets

In some cases, pallet use is unavoidable due to component use, board shape, etc. Wave solder pallets are designed to mask SMT devices while allowing PTH devices to be wave soldered normally. The key to a successful pallet design is to allow as much area as possible around the PTH leads. Keeping SMT components at least .150" away will allow proper pallet routing.
Pallets always increase the cost associated with the assembly. Pallets generally cost $500 to 1,000 each and depending on volume may require 15-20 to be effective.

12 Materials

Whenever possible, SMT devices should be selected from standard configurations. The standard components will be available from multiple sources and will usually be compatible will all assembly processes. For those devices developed to meet specific applications, standard packaging is often available. Select a package type that will be similar in materials and plating of standard device types when possible.
Illustrations of component dimensions beginning in each component section of IPC-SM-782 are accompanied by table of figures for each of the different part numbers, as taken from EIA-PDP-100, JEDEC-95 and other world wide component standards. EIA-PDP-100 is a catalogue listing of outline drawings illustrating the dimensions of supplier registered passive components; JEDEC-95 is the outlining document for solid state products. At times, the component tolerances or component gauge requirements do not necessarily reflect the exact tolerance on a manufacturers data sheet. Usually, this occurs when industry component specification ranges are so broad that they defy good surface mount design principles.
Refer to IPC-D-275, Design Standard for Rigid Printed Board and Rigid Printed Board Assemblies, Section 8.2 Material Selection.

12.1 Form Factor

Automated placement of SMT components requires proper packaging and presentation to the machines. Components packaged in either Tape and Reel or Matrix tray configurations are preferred. Tubes are acceptable for low volume applications. Bulk components should be avoided whenever possible. Additional costs for outside tape and reel sources can be avoided if components are specified correctly for automated placement.

12.2 Terminations

Whenever possible standard tin/lead alloy terminations should be specified. Other alloys containing silver, gold, or palladium may require special processing, increasing assembly costs.

12.3 BGA's

Whenever possible Overmolded plastic BGA's should be specified as opposed to the Encapsulated type. Encapsulated BGA's allow more of the BT substrate to be exposed, resulting in increased flex during heating cycles. While normal reflow is possible, rework or replacement of these parts becomes more difficult.

12.4 Sealed Components

South Bay Circuits, Inc. incorporates an aqueous cleaning process for surface mount and wave solder. Components unable to withstand the wash process are hand soldered, resulting in a higher assembly cost. Whenever possible sealed components should be specified.

12.5 Moisture Sensitive Components

Customers should be aware of and follow the component manufacturer's recommended guidelines for the handling and storage of moisture sensitive devices. Moisture sensitive components consigned to South Bay Circuits, Inc. should be shipped in sealed moisture barrier bags with sufficient amounts of desiccant and humidity indicator cards enclosed.

13 Appendix Listing