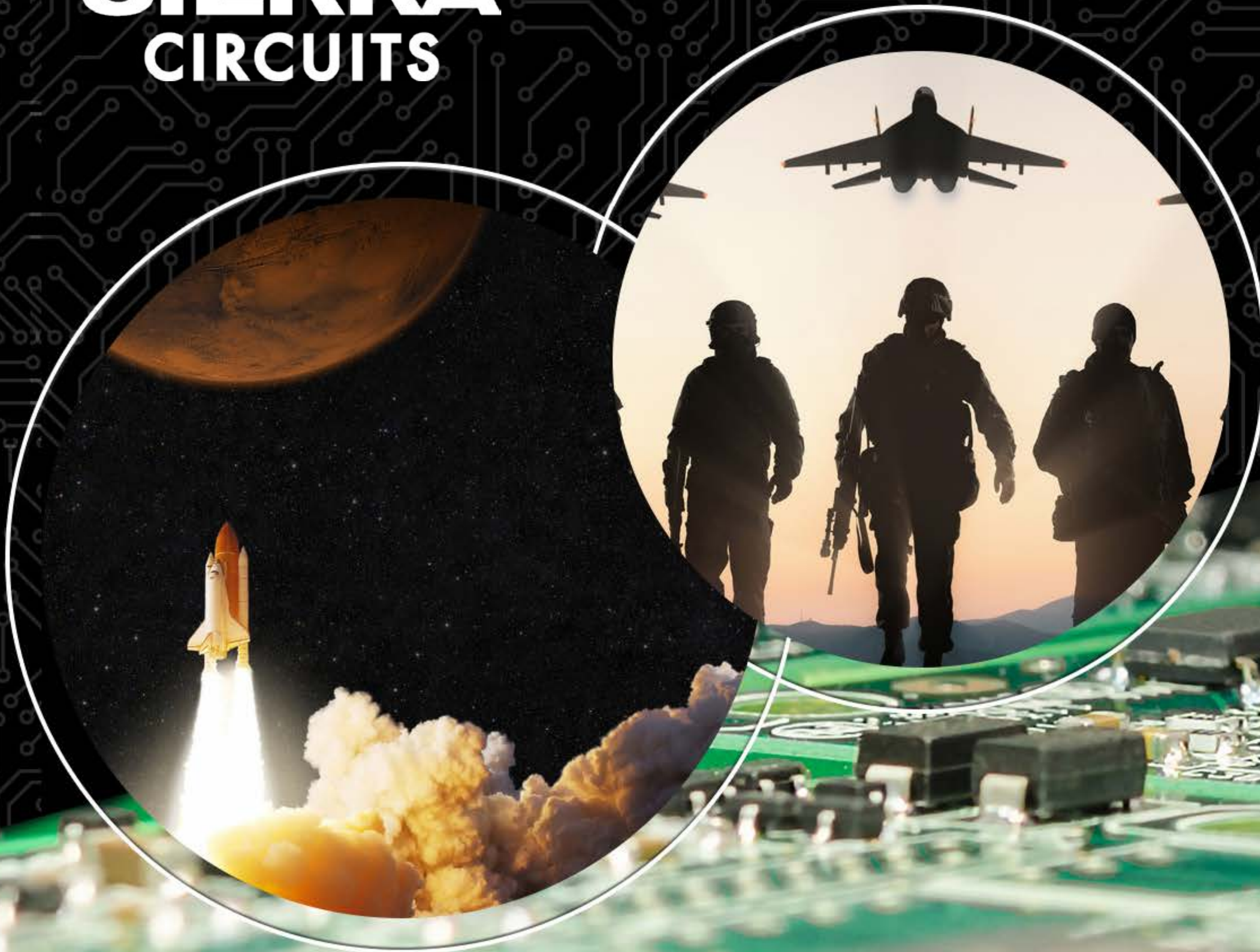
SIERRA CIRCUITS



IPC CLASS 3

DESIGN GUIDE

Table of contents

1. Introduction

PCB designers often wonder about the difference between the different IPC classes. It is always the end application of the PCB that determines the type of board to be used in that particular design.

When we talk about IPC classes like class 1, class 2, and class 3, we are speaking about the level of inspection that defines the manufactured board's precision and reliability. The three classes are categorized based on the criticality of the application, the tolerances to the harsh environment, and so on. In short, the three classes determine the quality of the board. With class 3 being the highest in quality and class 1 being the lowest. The other important thing that we would like to state here is that we cannot explain just about IPC class 3 without understanding the other two classes. Hence in this book, we mention class 2 and class 1 for a better understanding.

Before we dwell into the IPC classes, allow us to brief you about the IPC association.

2. What is IPC?

IPC is a global trade association for the electronic interconnection industry. Initially known as Institute for Printed Circuits, the organization changed its name to the Institute for Interconnecting and Packaging Electronic Circuits.

The organization publishes specifications on a regular basis.

The IPC standards are the most widely accepted rules by the electronic industry. This member-driven organization publishes standards for every stage of the electronic product development cycle, including design, purchasing, assembly, packaging, and more.

Adhering to the IPC standards will help in fabricating safe, reliable, and high-quality PCB products. Also, IPC compliance allows the designers and fabricators to be on the same page.

3. Class 1, class 2 and class 3 boards

The IPC-6011 describes the different classes for PCBs and the permitted defects for each board type. There are three IPC 6011 defined classes of electronic products with an addition of one IPC 6012 class 3/A standard:

Class 1 – General electronic products

Class 1 boards are assigned to general electronic boards with a limited life and a simple function. This class includes most typical everyday products. The class 1 boards allow various cosmetic defects as long as it doesn't affect the functioning of the board. The reliability of the product isn't a critical factor in these types of boards.

For instance, they can be found in TV remote controls, LED lights, kids' toys, etc. They are the most inexpensive boards to manufacture in the industry but they come with a limited life expectancy.

Class 2 – Dedicated service electronic products

Class 2 boards have higher reliability and extended life. They follow more stringent standards than class 1 but allow some cosmetic imperfections.

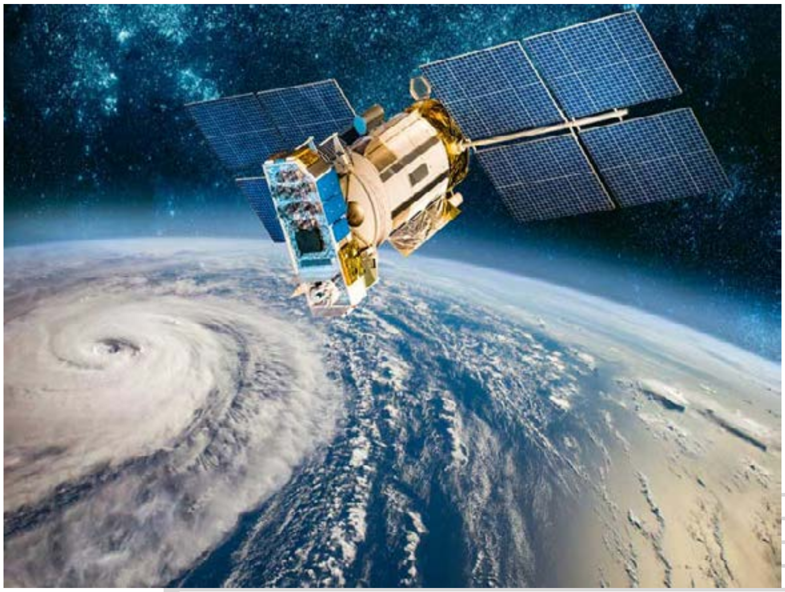
Here, uninterrupted service is preferable, but not critical. The class 2 products aren't exposed to extreme environmental conditions. The board is expected to run continuously but its operation is not extremely critical. These kinds of boards are implemented in your laptops, smartphones, tablets, communication equipment, etc.

Class 3 – High-performance electronic products

Class 3 boards must provide a continued performance or performance on demand. There can be no equipment downtime, and the end-use environment may be exceptionally harsh. High levels of inspection and testing are performed on these boards with stringent standards. This makes the class 3 boards highly reliable. This category includes critical systems such as life support systems, military equipment, electronic monitoring systems, automotive, etc.

IPC 6012 class 3/A

The IPC-6012 class 3/A is relatively a new class that includes space and military avionics. This is the highest class for printed circuits. The class 3/A boards call for very stringent manufacturing criteria since the boards should remain operational in critical conditions such as Outerspace, etc. These boards are expensive to manufacture compared to the other classes since they need to be close to perfection. They are found in aerospace, military airborne systems, and missile systems.



The major difference between all these classes is the degree of inspection. The classes define the permissible defects while manufacturing the boards.

The class 3 and 3/A boards are majorly implemented in critical military and aerospace equipment due to their reliability. That being said, there is a misconception that class 3 and 3/A are used specifically for aerospace. These standards can be used for any kind of application like the ones mentioned in class 2. But they become uneconomical due to the effort that goes into the manufacturing and inspection process.

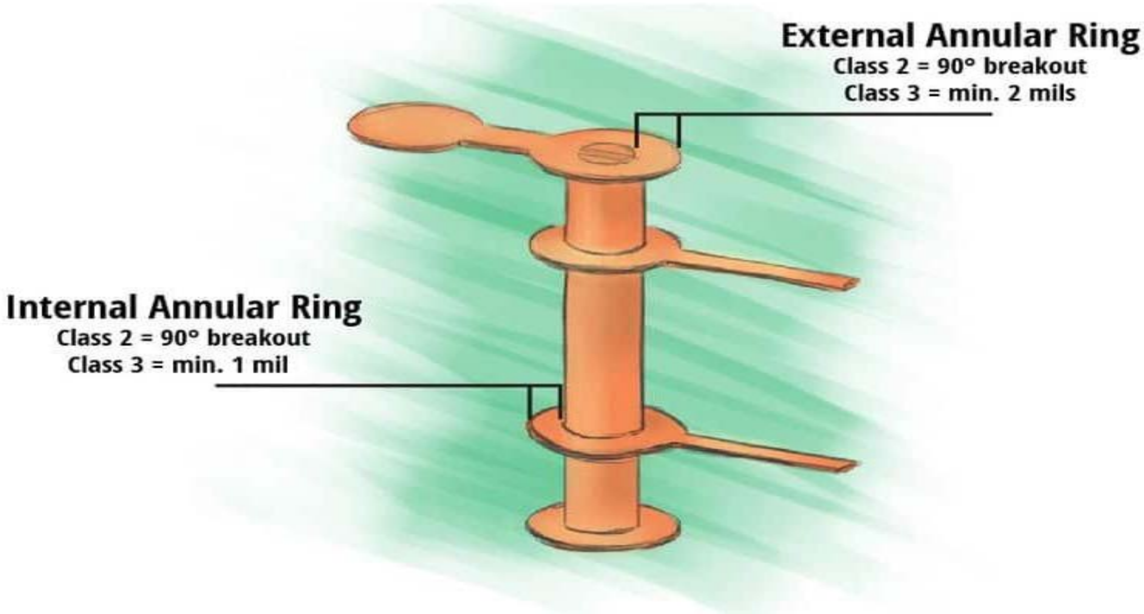
The designer can choose the life of his/her end product by choosing the right class. Sometimes class 2 fulfills all the criteria required for the end product and thus it can turn out to be more economical. But if the board is required to serve a critical application and is also expected to last for more than 15 years, then class 3 would be the right choice.

The environment in which the electronic product will be operational should also be considered since it decides the reliability segment of the design.

4. IPC guidelines for manufacturing defects

A board can have several faults in it and the IPC standards define the acceptable defects. Some defects might hamper the performance of the board, some are purely cosmetic imperfections and will have no impact on the board's consistent performance.

4.1 IPC standards for annular rings

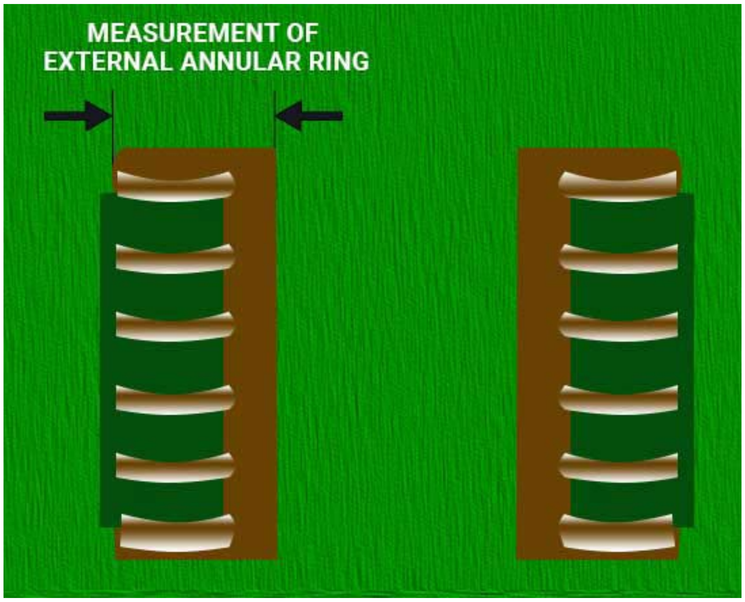


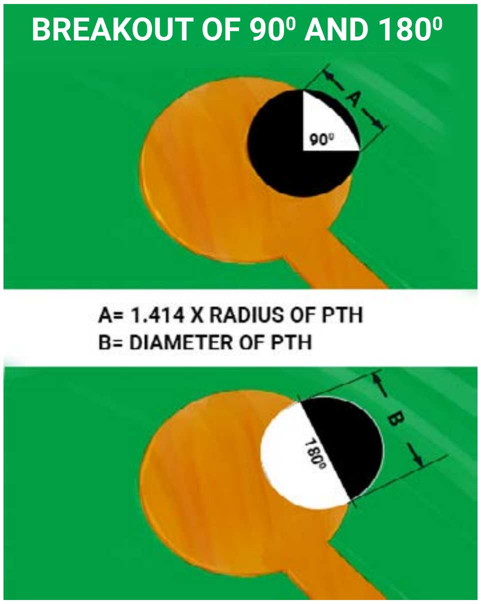
The IPC standards define the position of the holes on a landing pad and the width of the outer ring after a hole is drilled on to it.

Annular ring breakout

An annular ring breakout is a condition where a via/hole is not completely surrounded by land/copper.

Annular ring breakout measurement





Conductor to land junction area

The conductor to land junction is the 90⁰ area positioned around the point of contact between the conductor and the land. This area is specifically considered for annular ring breakouts.

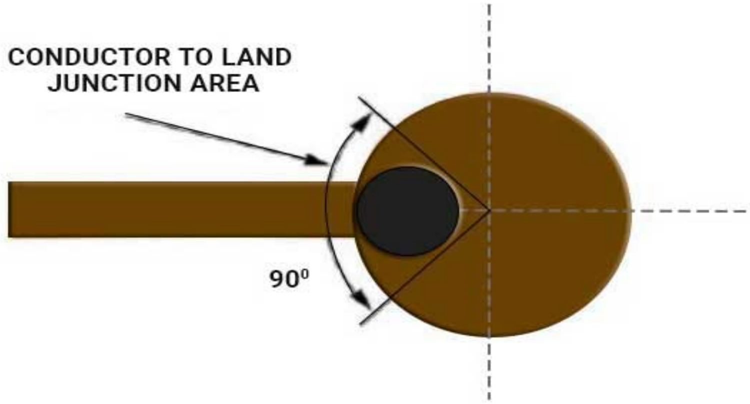


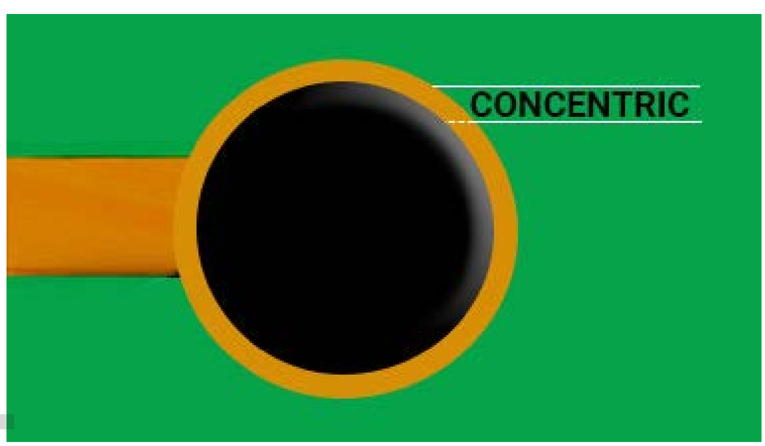
Table 1: IPC annular ring acceptance criteria

Feature	Class1	Class2	Class3
Plated-through hole	**180°**annular ring breakout from the land is acceptable provided the minimum lateral spacing is maintained. The land/conductor junction should not be reduced by more than **30%** of the minimum conductor width.	**90°** annular ring breakout from the land is acceptable provided the minimum lateral spacing is maintained. The land/ conductor junction should not be reduced by more than **20%** of the minimum conductor width. The conductor junction should not be less than 0.05mm or the minimum line width, whichever is smaller.	The minimum annular ring should not be less than 0.05mm. The minimum external annular ring may have a **20%** reduction of the minimum annular ring.

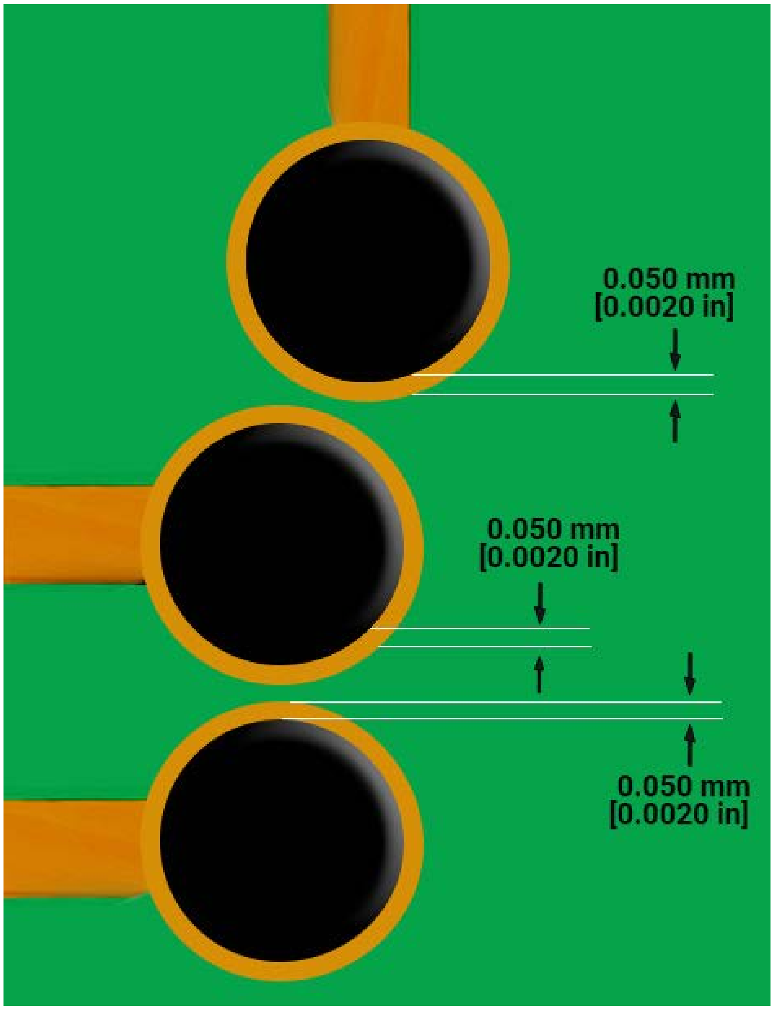
Illustration of acceptable annular rings

Target annular ring

Holes are centred with perfect annular rings

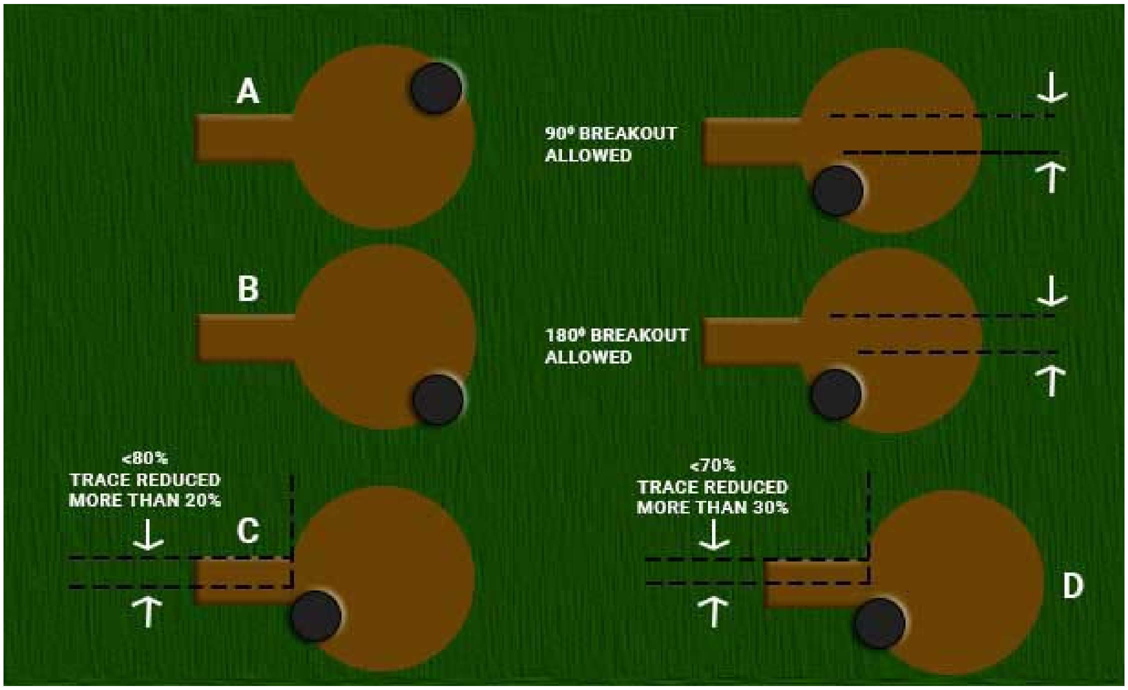


Class 3 acceptable annular ring conditions



- Holes are not centered but the annular ring area measures at least 0.05mm.
- The minimum external annular ring may have 20% reduction of the minimum annular due to defects.

Class 2 and class 1 acceptable annular ring breakout conditions



Class 2 annular ring criteria

- As depicted in diagram A, 90º breakouts or less than that are accepted.
- In situations where the breakouts occur at the conductor to land junction area, the conductor junction area should not be reduced by more than 20% of the minimum conductor width. Also, the conductor shouldn't be less than 0.05mm or the minimum linewidth as shown in diagram C.

Class 1 annular ring criteria

- As depicted in diagram B, 180º breakouts or less than that are accepted.
- In situations where the breakouts occur at the conductor to land junction area, the conductor junction area should not be reduced by more than 30% of the minimum conductor width as shown in diagram D.

4.2 Design rules for annular rings

To achieve acceptance for class 2 and class 3, follow the tables below published by Altium. The first one gives the annular ring requirements for mechanically drilled blind, buried, and through holes on ½ oz copper:

Table 2: IPC class 2 drill & pad diameter for 1/2 oz copper

Drill	Pad	Anti-Pad	PCB thickness	Aspect ratio
0.006"	0.016"	0.026"	Up to 0.039"	6.5:1
0.008"	0.018"	0.028"	Up to 0.062"	7.75:1
0.010"	0.020"	0.030"	Up to 0.100"	10:01
0.012"	0.022"	0.032"	Up to 0.120"	10:01
0.0135"	0.024"	0.034"	Up to 0.135"	10:01

Table 3: IPC class 3 drill & pad diameter for 1/2 oz copper

Drill	Pad	Anti-Pad	PCB thickness	Aspect ratio
0.008"	0.023"	0.033"	Up to 0.062"	7.75:1
0.010"	0.025"	0.035"	Up to 0.100"	10:01
0.012"	0.027"	0.037"	Up to 0.120"	10:01
0.0135"	0.028"	0.038"	Up to 0.135"	10:01

Table 4: Various copper thicknesses

Drill & pad diameter	8 layers or Less	>8 layers
IPC Class 2	Pad Diameter Over Drill	Pad Diameter Over Drill
1/4 oz Copper	0.010"	0.010"
3/8 oz Copper	0.010"	0.010"
1/2 oz Copper	0.010"	0.010"
1 oz Copper	0.012"	0.012"
2 oz Copper	0.014"	0.014"
3 oz Copper	0.016"	0.016"
4 oz Copper	0.018"	0.018"

Drill & Pad Diameter	8 Layers or Less	>8 Layers
IPC Class 3A	Pad Diameter Over Drill	Pad Diameter Over Drill
1/2 oz Copper	0.013"	0.015"
1 oz Copper	0.015"	0.017"
2 oz Copper	0.016"	0.018"

Drill & Pad Diameter	8 Layers or Less	>8 Layers
IPC Class 2	Pad Diameter Over Drill	Pad Diameter Over Drill
1/4 oz Copper	0.013"	0.015"
3/8 oz Copper	0.013"	0.015"
1/2 oz Copper	0.013"	0.015"
1 oz Copper	0.015"	0.017"
2 oz Copper	0.016"	0.018"
3 oz Copper	0.019"	0.021"
4 oz Copper	0.022"	0.024"

5. IPC standards for assembly process

In the assembly process, the major differences between class 2 and class 3 are found in component placement for surface-mount components, cleanliness requirements based on residual contaminants on the assemblies, plating thicknesses as defined in plating through-hole and on the surface of PCBs.

During assembly, **surface-mount components** might be slightly placed off pad. This is what we call a visual defect since it does not usually affect the electrical and mechanical performance. It, therefore, does not matter for class 2 circuit boards. However, class 3 does not accept any imperfection and this type of assembly misstep will cause the circuit board to fail the inspection.

5.1 IPC standards for solder joints

The amount of barrel fill required for through-hole leads is 50% for class 2 and 75% for class 3. As it can be delicate to get the paste into small plated through-holes (PTH), our advice is to design your PTH 15 mils over the diameter of the lead. This way, you will have 7.5 mils on each side, which will make it easier for the paste to fill the barrel.

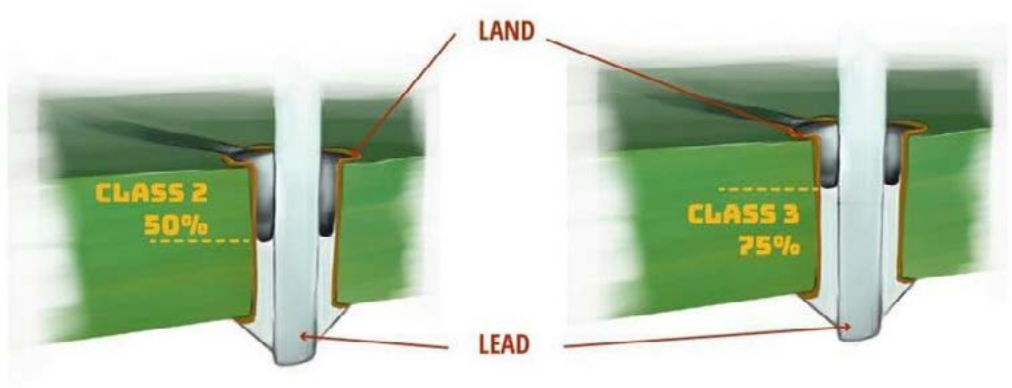
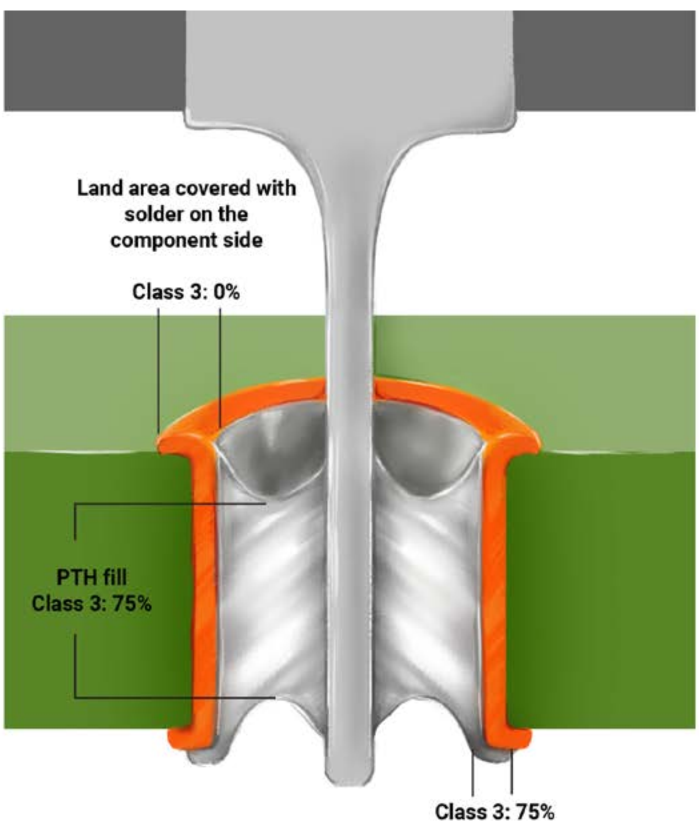


Table 5: Surface mount components and barrel fill

Factors	Class 2	Class 3
Surface-mount components	Can be slightly placed off pad. (Considered as a visual defect, doesn't affect the electrical and mechanical performance.)	Imperfections are not acceptable including visual flaws. This kind of imperfection will cause the circuit board to fail the inspection.
Amount of barrel fill	Throughhole leads 50%	Through-hole leads 75%

Solder coverage for joints is another factor addressed by the IPC standards. The acceptable IPC standards for soldering are described in the tables mentioned below.

Criteria for through-hole components



Land area covered with solder on the solder side

Circular wetting of solder of the lead and plated hole barrel

Class 3 component side: 270°
Class 3 solder side: 330°

Table 6: Acceptable solder criteria for through-hole components

Characteristics	Class 1	Class 2	Class 3
Circular wetting of solder of the lead and plated hole barrel on the component side	-	180^0	270^0
PTH fill	-	50%	75^0
Circular wetting of solder of the lead and plated hole barrel on the solder side	270^0	270^0	330^0
Land area covered with solder on the component side.	0%	0%	0%
Land area covered with solder on the solder side.	75%	75%	75%

Criteria for chip components

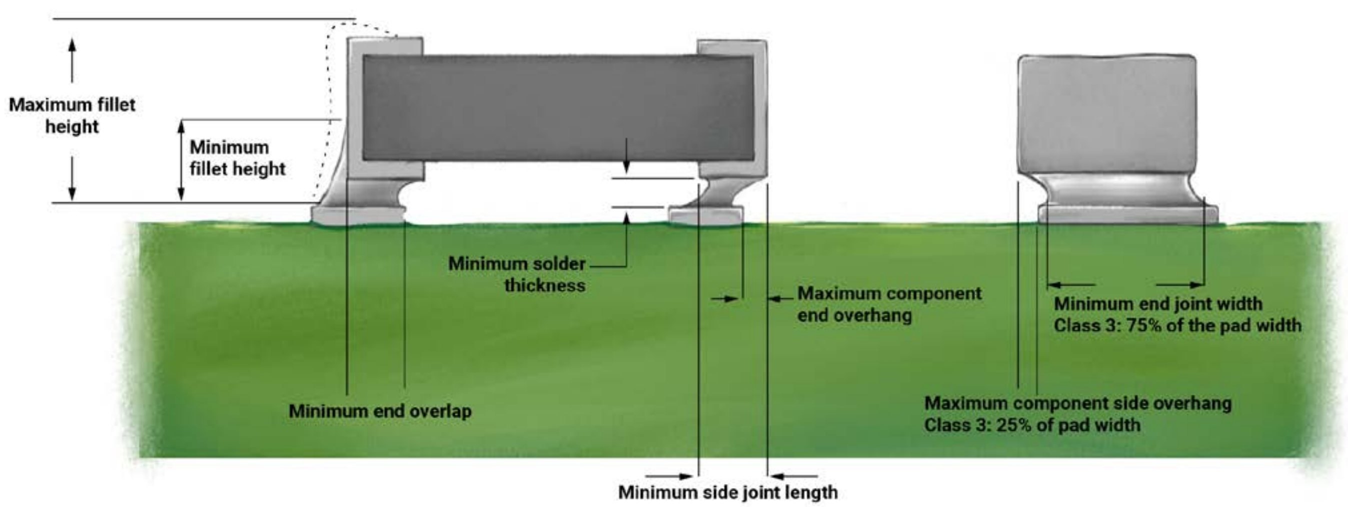


Table 7: Acceptable solder criteria for chip components

Characteristics	Class 1	Class 2	Class 3
Maximum component side overhang	<50% of component termination width or 50% of pad width	<50% of component termination width or 50% of pad width	<25% of the component termination width or 25% of pad width
Maximum component end overhang	Not acceptable	Not acceptable	Not acceptable
Minimum end joint width	50% of the component termination width or 50% of the pad width	50% of the component termination width or 50% of the pad width	75% of the component termination width or 75% of the pad width
Minimum side joint length	Proof of accurate wetting	Proof of accurate wetting	Proof of accurate wetting
Maximum side fillet height	Solder may overhang the pad but must not contact the non-soldered region of the component package body	Solder may overhang the pad but must not contact the non-soldered region of the component package body	Solder may overhang the pad but must not contact the non-soldered region of the component package body
Minimum fillet height	Proof of accurate wetting	Proof of accurate wetting	Equivalent to solder thickness+25%, or solder thickness +0.50 mm (.020”)
Minimum solder thickness	Proof of accurate wetting	Proof of accurate wetting	Proof of accurate wetting
Minimum end overlap	Proof of any overlap needed	Proof of any overlap needed	Proof of any overlap needed

Criteria for J lead components

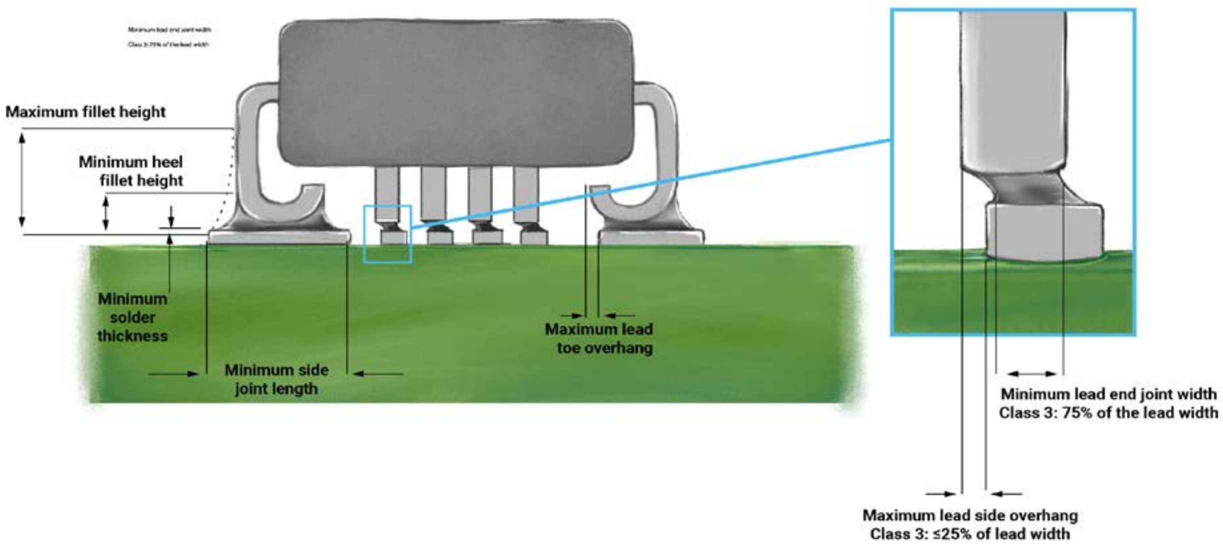


Table 8: Acceptable solder criteria for J lead components

Characteristics	Class 1	Class 2	Class 3
Maximum lead side overhang	≤50% of lead width	≤50% of lead width	≤25% of lead width
Maximum lead toe overhang	Not acceptable	Not acceptable	Not acceptable
Minimum lead end joint width	50% of the lead width	50% of lead width	75% of the lead width
Minimum side joint length	Proof of accurate wetting	Side joint length must exceed 150% of the lead width.	Side joint length must exceed 150% of the lead width.
Maximum fillet height	Solder must not touch the component package body.	Solder must not touch the component package body	Solder must not touch the component package body.
Minimum heel fillet height	Proof of accurate wetting	Equal to the solder thickness plus 50% of lead thickness.	Equal to the solder thickness plus 100% of lead thickness.
Minimum solder thickness	Proof of accurate wetting	Proof of accurate wetting	Proof of accurate wetting

Criteria for gull wing components

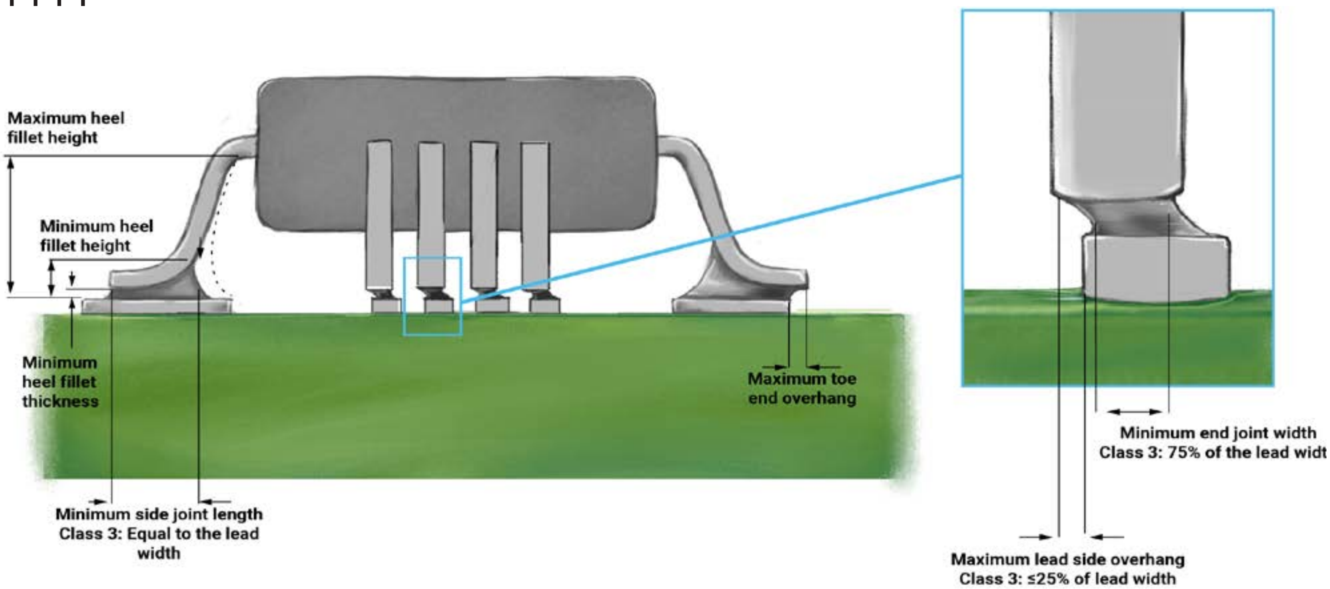
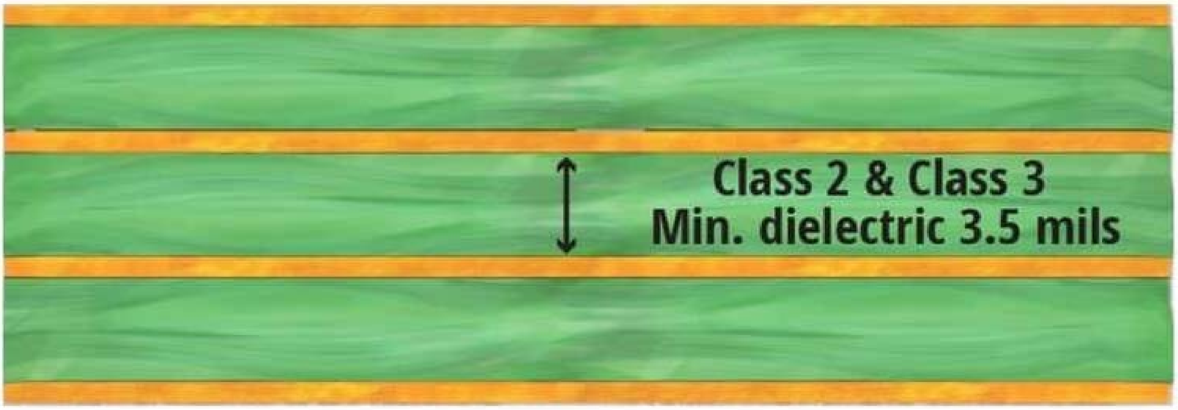


Table 9: Acceptable solder criteria for gull wing components

Characteristics	Class 1	Class 2	Class 3
Maximum lead side overhang	≤50% of lead width or 0.5mm (.020")	≤50% of lead width or 0.5mm (.020")	≤25% of lead width or 0.5mm (.020")
Maximum toe end overhang	Acceptable provided it does not violate electrical clearance.	Acceptable provided it does not violate electrical clearance.	Acceptable provided it does not violate
Minimum end joint width	50% of the lead width.	50% of lead width	75% of the lead width
Minimum side joint length	Equal to Lead width or 0.5mm (.020")	Equal to the lead width or 0.5mm (.020")	Equal to the lead width or 0.5mm (.020")
Maximum heel fillet height	Solder should not touch the component package body	Solder should not touch the component package body.	Solder should not touch the component package body.
Minimum heel fillet height	Proof of accurate wetting	Solder thickness+50% of lead thickness	Equal to solder thickness+100% of lead thickness
Minimum heel fillet thickness	Proof of accurate wetting	Proof of accurate wetting	Proof of accurate wetting

PCB dielectric requirement

The minimum dielectric for class 2 and class 3 is 3.5 mils or as agreed between user and supplier.



6. Common differences between IPC classes

When designers implement a class 3 circuit board, it implies that the product has to be built according to the complete IPC criteria. The design and the manufacturing teams must take into account the laminate selection, plating thickness, annular ring requirements, manufacturing processes, material qualification, facilities arrangements, inspection criteria, etc. in order to produce the board that meets all the Class 3 requirements.

This table will provide a glimpse of the different class standards with their acceptable defects.

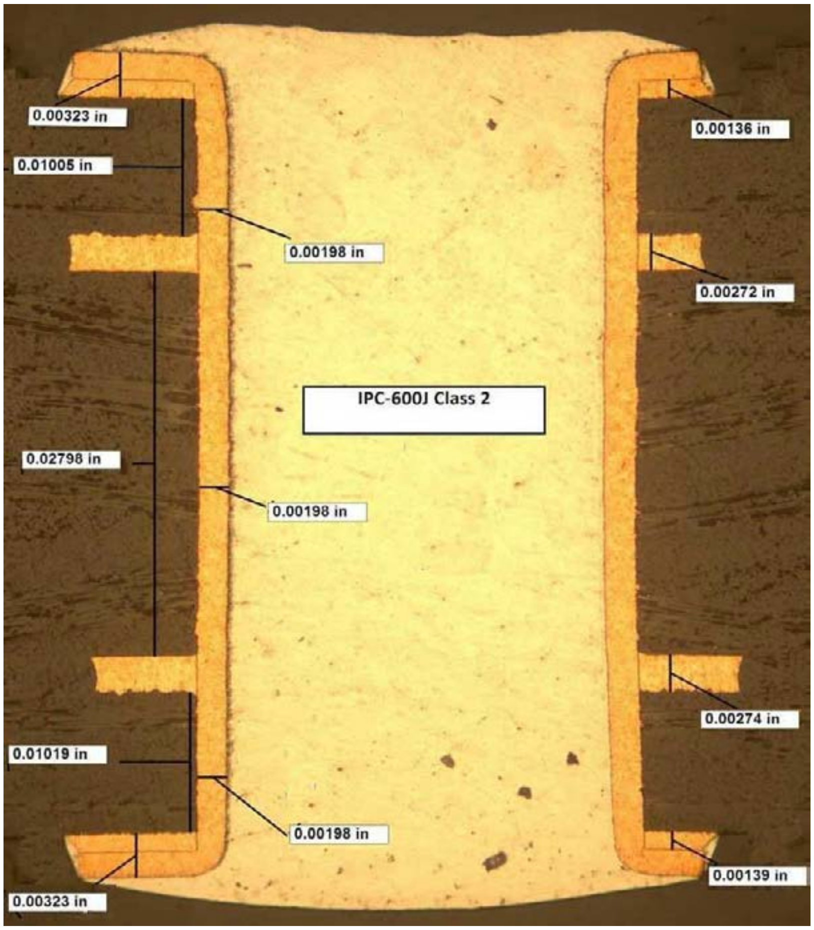
Features	Class 2	Class 3	Class 3A*
External annular ring PTH	90° breakout accepted provided minimum lateral spacing is maintained	50µm [0.00197 in.] minimum 20% isolated area reductions due to defects permitted	50µm [0.00197 in.] minimum 20% isolated area reductions due to defects permitted.
Internal annular ring PTH	90° breakout accepted provided minimum lateral spacing is maintained.	25µm [0.00098 in.] minimum	25µm [0.00098 in.] minimum
Annular Ring, unsupported hole	90° breakout accepted provided minimum lateral spacing is maintained.	150µm [0.00591 in.] minimum	150µm [0.00591 in.] minimum
Burrs and nodules	Allowed, provided the minimum hole diameter and copper thickness meet th requirements		
Bow and twist	0.75% maximum for SMT boards / 1.5% for the rest		
Laminate cracks	Within 80µm [0.00315 in]		
Barrel cracks	Not acceptable		
External foil cracks	Allowed, provided it does not extend into the plating		
Internal foil cracks	Not acceptable		
Dielectric thickness/spacing	90µm [0.00354in.] minimum		
Lifted land (visual)	Not acceptable		
Solder connection regions de-wetting	5% maximum	5% maximum	Not acceptable
	Conductors and planes are permitted		
Plating thickness, copper, through, blind, buried vias, >2 layers, average	20µm [0.00079in.] minimum	25µm [0.0009 in.] minimum	25µm [0.00098in.] minimum
Plating thickness, copper, through, blind, buried vias, >2 layers, thin areas	18µm [0.00071in.] minimum	20µm [0.00079 in.] minimum	25µm [0.00098in.] minimum *
Plating thickness, copper, through, blind, buried vias, >2 layers, wrap	5µm [0.00019in.]	12µm [0.00047in.]	12µm [0.00047in.]
Plating thickness, copper, blind and buried microvias, average	12 µm [0.00047in.] minimum	12µm [0.00047in.] minimum	20µm [0.00079in.] minimum*
Plating thickness, copper, blind and buried microvias, thin areas	10 µm [0.00039in.] minimum	10µm [0.00039in.] minimum	18µm [0.00071in.] minimum*
Plating thickness, copper, blind and buried microvias, wrap	5µm [0.00019in.]	6µm [0.00024in.]	6µm [0.00024in.]
Plating thickness, copper, buried via cores (2 layers), average	15µm [0.00059in.] minimum	15µm [0.00059in.] minimum	20µm [0.00079in.] minimum*
Plating thickness, copper, buried via cores (2 layers), thin areas	13µm [0.00051in.] minimum	13µm [0.00051in.] minimum	18µm [0.00071in.] minimum*
Plating thickness, copper, buried via cores (2 layers), wrap	5µm [0.00019in.]	7µm [0.00027in.]	7µm [0.00027in.]
Nicks and pinholes, planes	Maximum size is 1.0mm [0.0394 in] with not more than 4 per side, per 625 sqcm [96.88 sqin]		
Negative etchback	25µm [0.00098in.]	13µm [0.00051in.]	None allowed*
	Between 5µm [0.000197in.] and 80µm [0.00315in.] with preferred		Between 5µm [0.000197in.] and 40µm [0.001574in.] with

Surface mount lands	Imperfections along edge of land not more than 20%; internal defects not more than 10%		
	Faults internal to the land remain outside central 80% of the diameter.		
Plating separation	Not acceptable		
Laminate voids	Voids within and outside the thermal zone not more than 80µm [0.00315 in.]		
Voids, copper in holes (visual)	One per hole in not more than 5% of the holes	None allowed	None allowed at 3 diopters *
Voids, final finish plating (visual)	Three per hole in not more than 5% of the holes	One per hole in not more than 5% of the holes	None allowed at 3 diopters *
Nail heading	Allowed		
Inner layer inclusions/separations	Not acceptable		
Cap plating of filled holes (visual)	Unless covered by soldermask, plating voids exposing resin are not acceptable		
Cap plating of filled holes (minimum)	5µm [0.00019 in.]	12µm [0.00047 in.]	12µm [0.00047 in.]
Cap plating of filled holes depression (maximum)	127µm [0.005 in.]	76µm [0.0030 in.]	76µm [0.0030 in.] general areas 50µm [0.0020 in.] in BGA areas
Cap plating of filled holes protrusion (maximum)	50µm [0.0020 in.]	50µm [0.0020 in.]	50µm [0.0020 in.]
Electrodeposited copper tensile strength	> 248 Mpa [36,000 PSI]	> 248 Mpa [36,000 PSI]	> 275.8 Mpa [40,000 PSI]
Electrodeposited copper elongation	> 12%	> 12%	> 18%
*Requirements for Class 3A products are referenced in IPC-6012DS.			

7. PCB cross-section verification

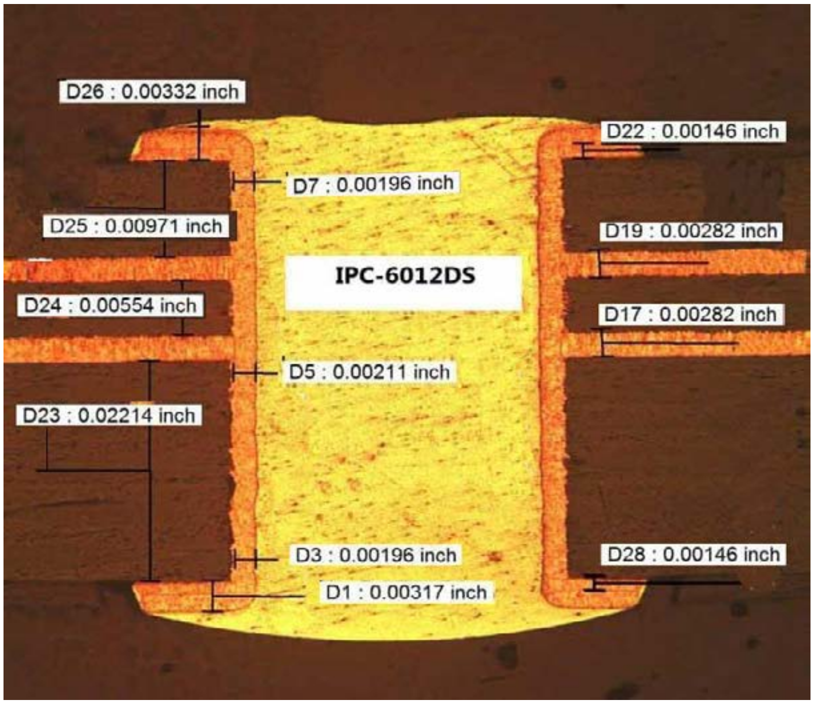
Just the visual and X-ray inspections techniques are not always enough to ensure the integrity of a board. To verify if your PCB manufacturer has met your requirements, request a cross-section analysis. The cross-section analysis is a destructive technique that verifies the PCB internal structure, mostly using a microscope. The test can check for various aspects, such as cracks, voids in solder joints, through-hole filling, etc.

Below is a cross-section of a class 2 circuit board:



Cross-section of a via in a 4-layer board

And this is a cross-section of a class 3A board:



Cross-section of a via in a 4-layer board

At Sierra Circuits, we do in-process cross-sections for every circuit board we manufacture at each step of the building process. We check for dielectric, plasma etch, solder mask thickness, copper, plating, etc. And if we don't meet the customer's requirements, we reject the board and build another one. If you need a report, you can ask for a final cross-section. We will send you a document with everything we have tested with the results.

8. IPC documents to set the level of acceptance criteria

The IPC documents come in handy to set the level of acceptance criteria for each class of products.

Significant documents for board performance and quality:

1. IPC 2220 series for circuit board design and fabrication
2. IPC 6010 series for board performance quality
3. IPC-A-600 for board acceptability requirements
4. J-STD-001 for soldering requirements

SIERRA CIRCUITS

Sierra Circuits, Inc.
1108 West Evelyn Avenue
Sunnyvale, CA 94086
Phone: +1 (408) 735-7137
www.protoexpress.com