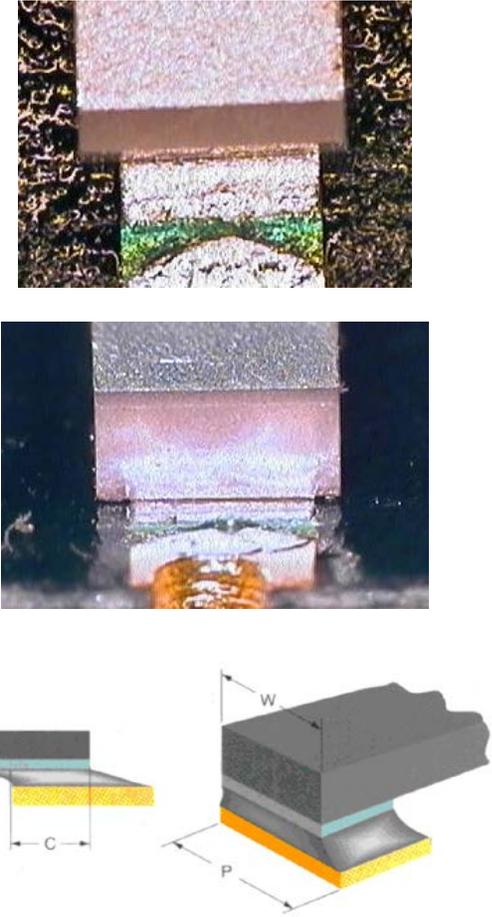


Inspection Method Sheet

Part Number: Generic
Drawing Number: Generic
Page 1 of 10
Doc. #: TT-PC-0378 Rev. 13

Part Name: PCB Filters
Operation: In Process / Final
Written By: Myra Cope
Date: 10-15-08

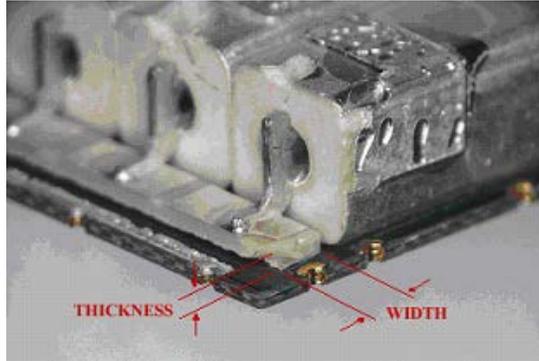
Applicable customer specifications take precedence over this procedure (reference customer drawing).

Description / Dimensions	Picture / Detail	Sample Size / Method / Standard
<p>1) Applicable Documents:</p> <ul style="list-style-type: none">Applicable customer specifications take precedence over this procedure.	<ul style="list-style-type: none">Please Reference customer's drawingIPC-A610 and IPC-A-600 Class 2 shall be used to quality conditions not covered in this procedure or customer requirements.	
<p>2) Shifted Substrate: (side to side)</p> <p>The substrate should be centered on the pad side to side. As shown, no more than 50% of the pad is exposed on one end.</p> <p>Shifted Substrate: (front to rear)</p> <p>Solder joint width (c) shall be a least 50% of the width of the substrate (w) or the pad (p) whichever is less.</p>		<ul style="list-style-type: none">Method: Visual inspection at 4x, Verify at 10x magnification (per IPC-A-610 Section 8.2.1.3)Sample Size: 100%

3) Substrate Chips:

Inspect for Chips on Substrate. No chips greater than 50% of the substrate thickness or 50 % of the width of the substrate. No hanging chips allowed.

*** Chips must not under cut the solder pad.**

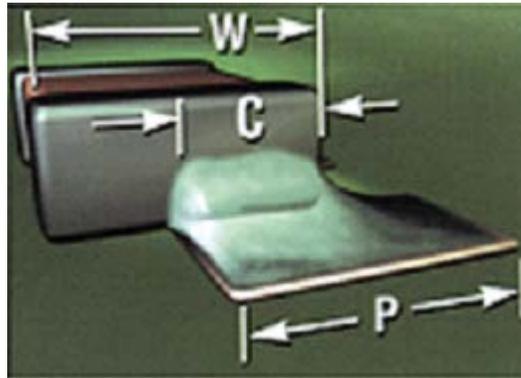


• **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A-610 section 9.4)

• **Sample Size:** 100%

4) Discrete inductors and capacitors:

Must be placed so that a proper overhang, joint width and solder fillet are present. Joint with (C) must be at least 50 % of the component width (W) or the width of the pad (P) whichever is smaller.



• **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A-610 section 8.2.2.3)

• **Sample Size:** 100%

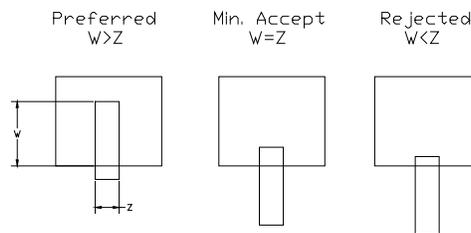
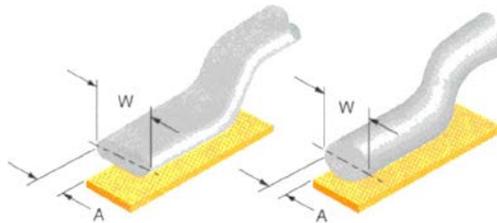
5) Round or Flattened leads:

Side overhang (A) shall not be greater than 50% of the lead width (W).

Minimum side joint length (W) is equal to or greater than the leadwidth or diameter.

(This also relates to tab overhang on substrate pads)

A joint must be visible on at least one side of the lead/bottom of tab.



• **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A-610 Section 8.2.6.1)

• **Sample Size:** 100%

6) Solder Joint Quality:

Acceptable:

Solder joints shall exhibit complete solder melting and good wetting between parts being soldered.

Reject: Solder is dull and over heated. Evidence of oxidized solder balls on top of the tab. Improper wetting to the tab is shown.

Reject: solder does not form a smooth edge with the components being joined. Un-melted solder balls are evident.

Solder Fillets:

Lead Based:

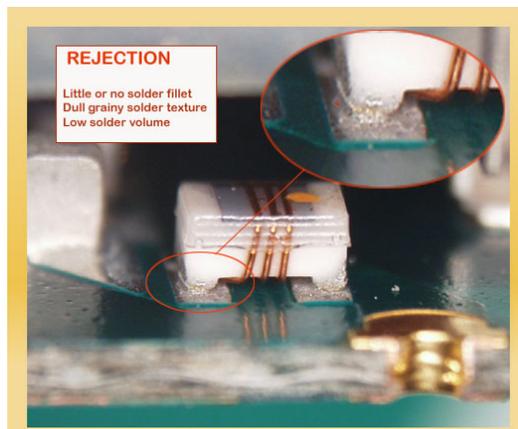
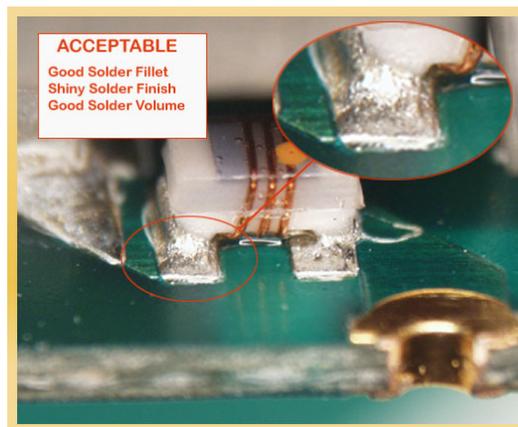
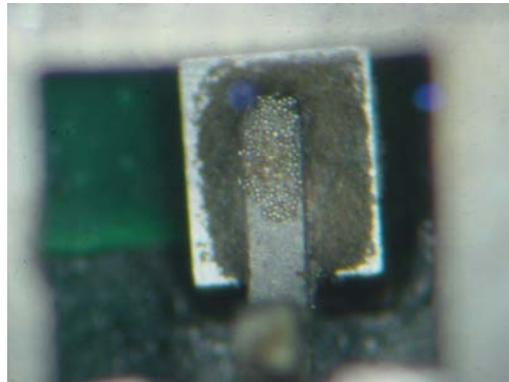
Verify solder joints have adequate solder fillets, there is good solder flow and joint cannot contain unmelted solder balls and/or have cracks. Solder joints should not be gray in color.

Tin/Silver base no clean:

Verify solder joints have adequate solder fillets, there is good solder flow. Surface may appear slightly grainy. Solder joint not contain unmelted solder balls and/or have cracks. Solder joints should not be gray in color.

Blowholes, pinholes, and voids:

Refer to IPC-A-610 Section 5.2.2



- **Method:** Visual inspection at 4x, Verify at 10x magnification
- **Sample Size:** 100%

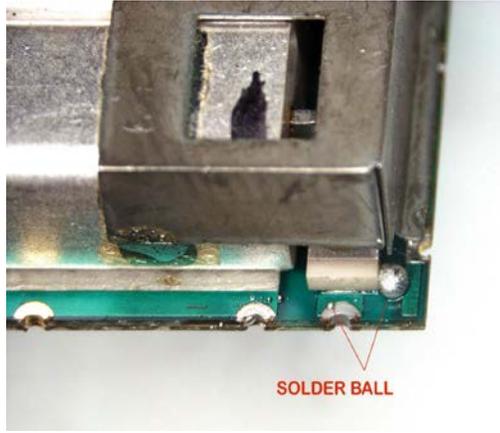
7) Solder Balls and Excess Solder:

Solder balls are acceptable if:

1. Attached – Entrapped or encapsulated by flux or trapped underneath a resonator or attached to a metal surface.

2. Unattached- Loose/unattached solder balls that do not meet acceptable conditions must be removed.

Excess solder that bridges between non-common conductive surfaces must be removed or reworked.



- **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A-610 section 5.2.6.1)
- **Sample Size:** 100%

8) Component Damage:

Verify components are not burned, broken, or otherwise damaged. (Burned component shown)



- **Method:** Visual inspection at 4x, Verify at 10x magnification
- **Sample Size:** 100%

9) Case Solder Joints:

- Inspect solder joints on cases. Top of case must be securely soldered to the resonators.
- Gaps along the case window are acceptable as long as the top of the case is secure.

Note: See router for special instructions.

Inspect for voids along the cases and PCB joint:

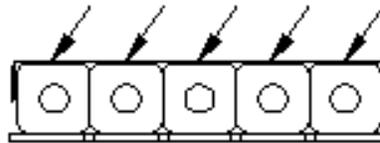
Acceptable conditions:

- No single area has a void greater than the following in length

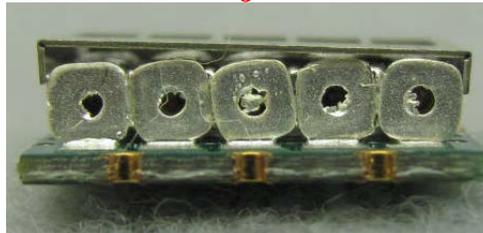
Resonator Size/ Max Void Size

- 1.3mm / 0.55mm void
- 4mm / 1.0mm void
- 6.8mm / 1.5mm void
- 12mm / 2.0mm void

A solder joint that exhibits less than complete solder joint fill, but shows visible attachment between PCB and cases.



Reject



- **Method:** Visual inspection at 4x, Verify at 10x magnification
- Spot Size Gauge
- **Sample Size:** 100%

Definition:

- A void is an area between the case and PCB land pattern with no visible solder connecting the PCB and the case. An area of the fillet that has less solder fill but shows visible attachment between the PCB and the case is not a void.

9) Case Solder Joints (Cont'd):

Unacceptable:

- Any single void greater than specified.
- Any more than 4 voids of acceptable size.
- The case flange has less than 50% overlap of PCB land pattern.

For cases with teeth:

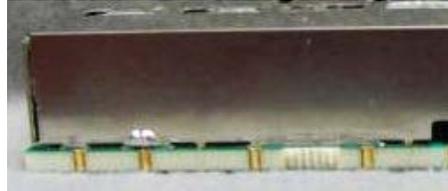
Open Area must be < 50% between "Teeth"



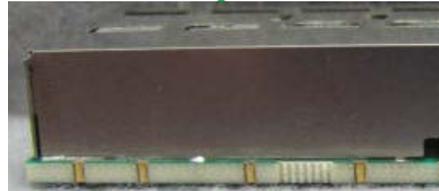
V is less than or equal $\frac{1}{2}$ W
(No less than 50% of the land exposed)

For Cases with Teeth

Rejected



Acceptable



- **Method:** Visual inspection at 4x, Verify at 10x magnification
- Spot Size Gauge
- **Sample Size:** 100%

Definition:

A void is an area between the case and PCB land pattern with no visible solder connecting the PCB and the case. An area of the fillet that has less solder fill but shows visible attachment between the PCB and the case is not a void.

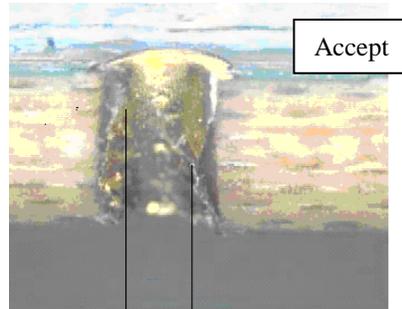
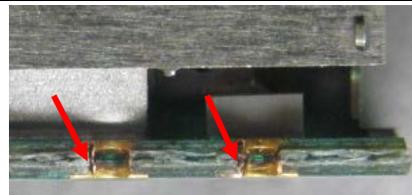
10) Inspection Castellations:

Solder Resist:

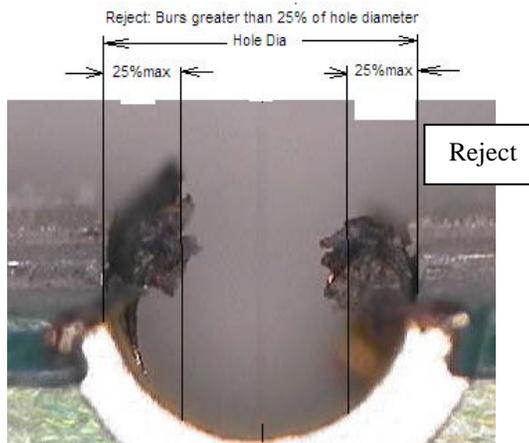
No Solder resist is allowed to cover any portion the half holes around the perimeter of the filter.

Metalization:

- Edge Castellations: Conductor must be continuous from top to bottom, and width must not be reduced more than 50% at any point.
- Edge Castellations: Conductor burrs may not be longer than 25% of castellation width (hole diameter).



50% of width remaining



- **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A-600 section 2.5.4)
- **Sample Size:** 100%

11) Excess Solder:

Complanarity:

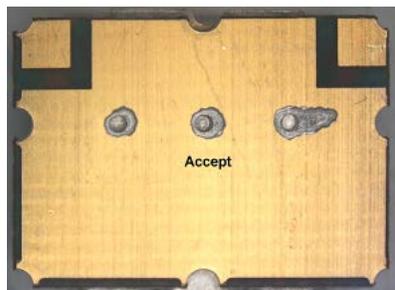
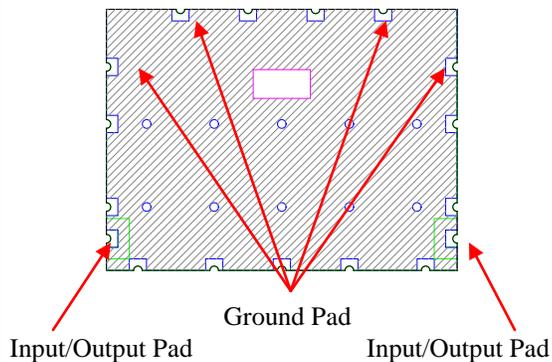
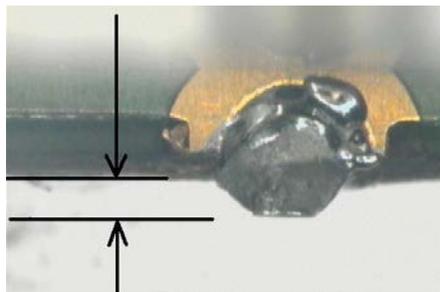
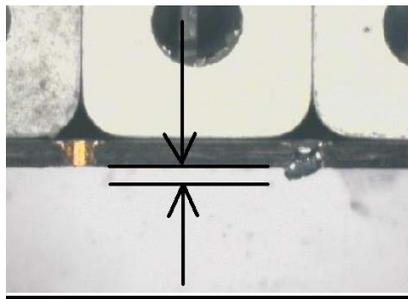
Solder may not cause a non-complanarity in excess of 0.005”

Back (or bottom) of the PCB:

- Solder on the bottom of PCB is acceptable as long as coplanarity is not violated and there are no bridges between I/O pad and ground.

Solder in vias:

- Solder is allowed in half hole vias only up to 50% filled. Totally filled vias are not acceptable.
- Input/Output vias shall be free of solder buildup. A thin coating of solder on the gold is permissible.
- Solder may flow through vias onto the back of the board as long as coplanarity is not violated.
- Solder may not protrude past the board edge causing the part outline to be violated.



- Method:** Visual inspection at 4x, Verify at 10x magnification

Use one of the 2 methods below:

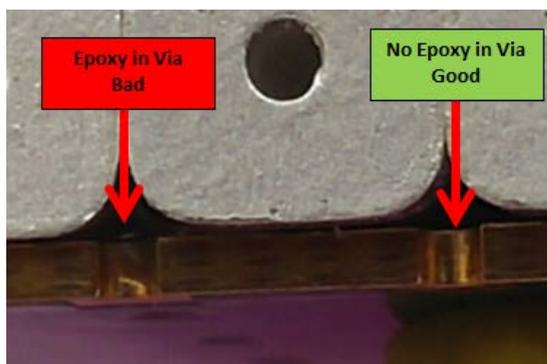
- Visual – using a Spot Size Gauge Guide
- Granite Block and a 0.005” shim

- Sample Size:** 100%

12) Inspect for Epoxy in the Vias:

No Epoxy allowed in the Vias

(Reference TT-PC-0676, Assembly Process – Epoxy for RoHS Filters, for epoxy placement)

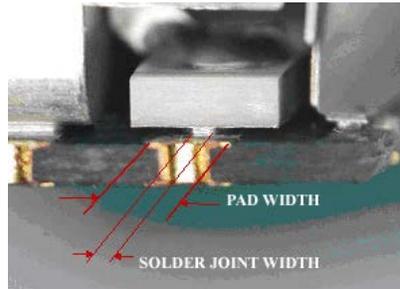


- Method:** Visual inspection at 4x, Verify at 10x magnification

- Sample Size:** 100%

13) Lifted Substrate:

- The substrate should not be lifted from the mounting pad such that it causes an improper solder joint.
- Solder joint to be at least 50% of the pad width or substrate width, whichever is smaller



- **Method:** Visual inspection at 4x, Verify at 10x magnification
- **Sample Size:** 100%

14) Pad Delamination

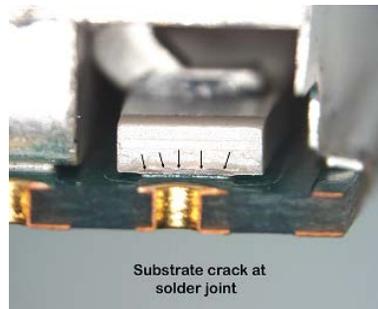
- There should be no lifting, tearing or exposed copper on PCB solder lands (not including half holes).



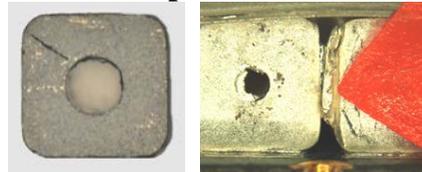
- **Method:** Visual inspection at 4x, Verify at 10x magnification
- **Sample Size:** 100%

15) Cracks and Fractures:

- Inspect for cracks on both substrate and resonator body. None are allowed on the substrate.
- Inspection for cracks to be performed at 4x magnification. Suspect parts may be verified at 10x magnification.



Unacceptable Cracks



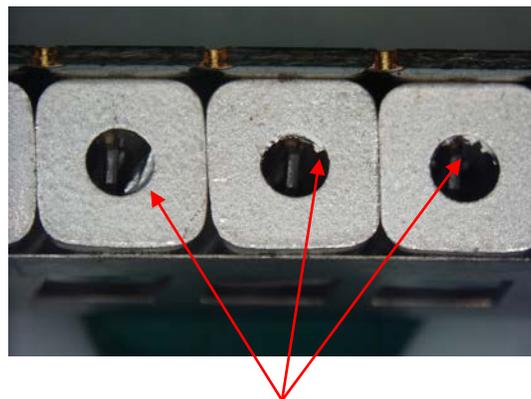
- **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A-610 section 9.4)
- **Sample Size:** 100%

16) Flashing Metallization:

- There will be no excess silver on the inner diameter that can detach.

Flashing that will not detach easily is acceptable and should not be removed.

- Flashing that protrudes out beyond the face of the resonator shall be removed.
- Acceptable conditions.



NOTE: Do NOT use any sharp objects to remove flashing (blade, knife....)

- **Method:** Visual inspection at 4x, Verify at 10x magnification
- **Sample Size:** 100%

17) Flux Residue:

- Verify that the filters have been processed through the cleaning operation

- Residues resulting from a no-clean flux are acceptable as long as they do not adversely affect coplanarity nor prevent verification of other visual requirements is acceptable.

- **Method:** Visual inspection at 4x, Verify at 10x magnification (per IPC-A610 section 10.4.4)

- **Sample Size:** 100%

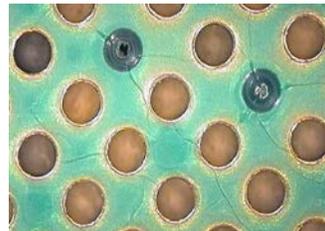
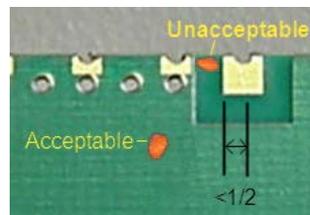
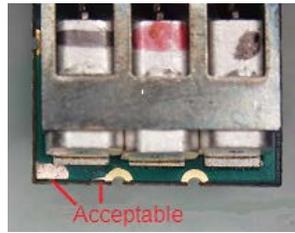
18) Check Soldermask:

- Chips in soldermask are allowed on the top of the board if there is not a bridge between adjacent circuit pads. Missing soldermask can extend between ground vias.

- Chips on the back are allowed if the size is not greater than 50% of the largest solder pad.

Chips that bridge ground and input/output pads are not allowed.

- Cracks in the soldermask are acceptable as long as the soldermask is secure to the board and shows no evidence of lift. This can be verified with tape.



- **Method:** Visual inspection at 4x, Verify at 10x magnification

- **Sample Size:** 100%

19) PCB Delamination:

Definition:

Delamination is the separation of laminate layers within the PCB. It shows visually as a light spot in the laminate or in extreme cases as a blister where the laminate thickness is dramatically increased.

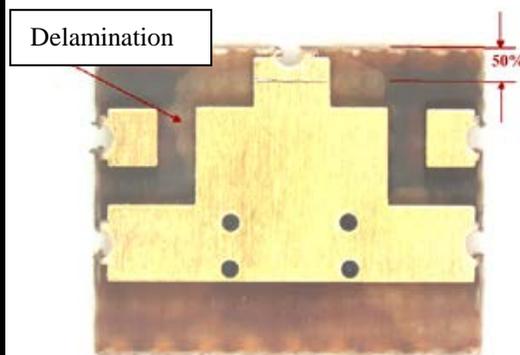
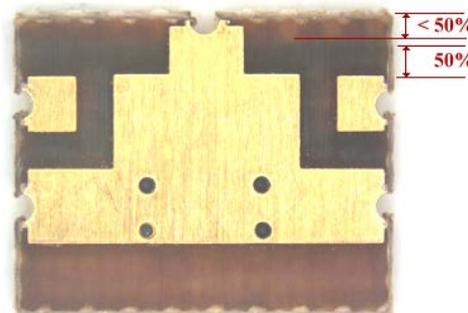
Minimum acceptable:

Delamination of the PCB along the scored edge is acceptable provided the delamination does not extend more than 50% of the distance from the board edge to the metal circuit trace.

Reject:

- Delamination between circuit traces
- Edge delamination more than 50% of the distance from the edge of the PCB to circuit trace.

Blistering (raised areas) in the laminate.



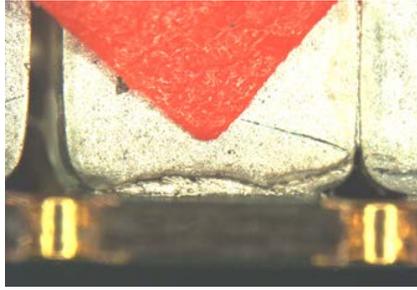
- **Method:** Visual inspection at 4x, Verify at 10x magnification

- **Sample Size:** 100%

20) OD Flashing:

- Flashing that will not detach is acceptable on the outer diameter of resonators.

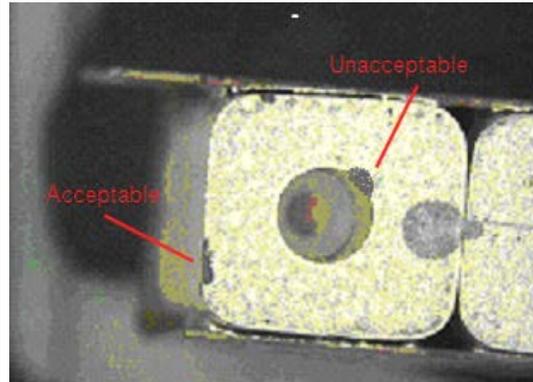
NOTE: Do NOT use any sharp objects to remove flashing (blade, knife....)



- Method:** Visual inspection at 4x, Verify at 10x magnification
 - Sample Size:** 100%

21) Chips:

- Chips in the ceramic and metalization are acceptable on the edge or face of resonator unless metal is peeling from edge of chip.
- Exposed ceramic chips on the ID are unacceptable
- No hanging chips allowed.
- Perform Tape test to confirm peeling/ adhesion



- Method:** Visual inspection at 4x, Verify at 10x magnification

Tape Test per IPC-TM-650 (using a 600 3M tape)

- Sample Size:** 100%

22) Blister:

- Small blisters on end of resonators are acceptable. Large blisters are to be rejected.

Tape Test to confirm adhesion



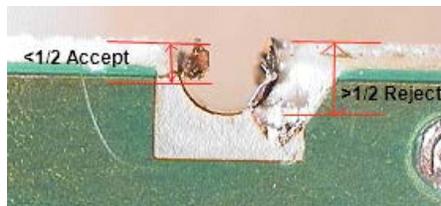
- Method:** Visual inspection at 4x, Verify at 10x magnification

Tape Test per IPC-TM-650 (using a 600 3M tape)

- Sample Size:** 100%

23) Board Damage:

- Damage is unacceptable if the amount of material removed is greater than $\frac{1}{2}$ the depth of the half hole.
- Nicks are acceptable as long as coplanarity is not affected and copper is not exposed. If copper is exposed possible rework may be performed with a solder mask pen.
- Damage is unacceptable if the laminate is peeling and/or the board is split.
- Damage is unacceptable if the board is split or coplanarity is affected.



- Method:** Visual inspection at 4x, Verify at 10x magnification

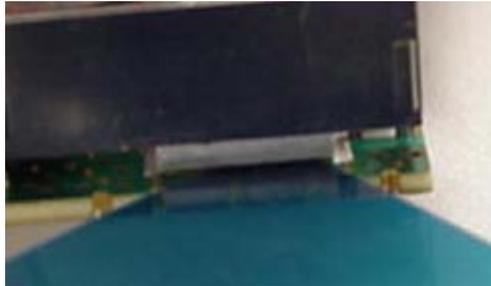
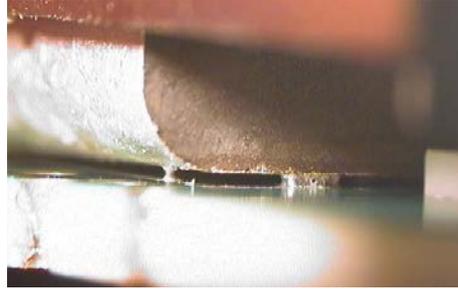
- Sample Size:** 100%

24) Lifted Resonators:

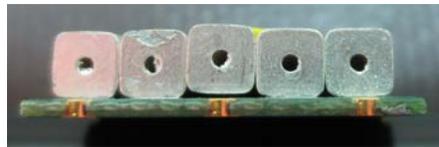
Side Resonators:

- Unacceptable if a 0.005" shim can be inserted easily under half the width of the resonator.

Side Resonators



End Resonators



End Resonators:

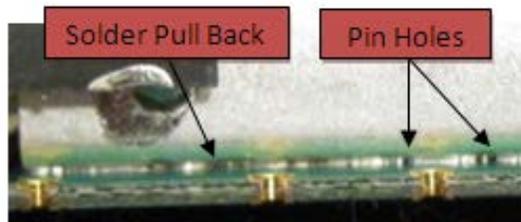
- Unacceptable if a 0.005" shim can be inserted easily under half the length of the resonator.

Both:

- Lifted resonators can possibly be reworked as long as the resonator is secure and the case fit is not affected.

Acceptable solder connection between the resonator and the board:

- Pin Hole in the Solder Joints
- 50% solder joint between the resonator body and the pad

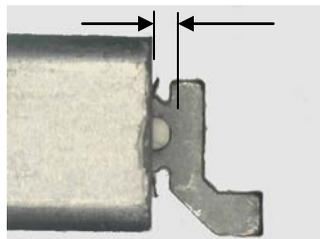


- **Method:** Visual inspection at 4x, Verify at 10x magnification and/or use a 0.005" shim

- **Sample Size:** 100%

25) Tab Protrusion:

- Tab must be seated properly and not protruding from opposite end. The dimension cannot be greater than 0.005".



- **Method:** Visual inspection at 4x, Verify at 10x magnification and/or use a 0.005" shim

- **Sample Size:** 100%