Association Connecting Electronics Industries

IPC-7711B/7721B Change 1
November 1, 2011

Rework, Modification and Repair of Electronic Assemblies

Developed by the Repairability Subcommittee (7-34) of the Product Assurance Committee (7-30) of IPC

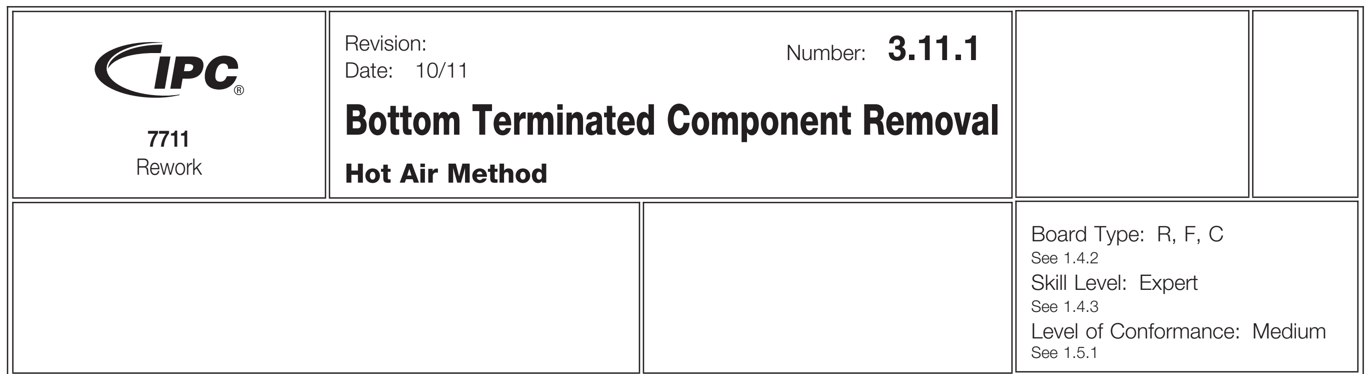
These procedures are new additions to the IPC-7711B/7721B. The pages herein are authorized for download, reproduction, and printing for insertion into the IPC-7711B/7721B.

These procedures are not intended for previous versions of IPC-7711/7721.

Users of this publication are encouraged to participate in the development of future revisions.

Contact:
IPC
3000 Lakeside Drive, Suite 309S
Bannockburn, Illinois
60015-1249
Tel 847 615.7100
Fax 847 615.7105

This Page Intentionally Left Blank



GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

OUTLINE

This procedure identifies the procedural steps which need to be accomplished to affect bottom termination component (QFN, LGA, BTC) removal. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (systems manufacturers will customarily provide generalized operating procedures which must be further refined to achieve optimum results).

NOTE

The following preconditions shall be accomplished prior to performing the procedures:

1. Develop a time/temperature profile (TTP) for the specific bottom terminated component (BTC) and PCB.
2. Moisture sensitive components (as classified by IPC/JEDEC J-STD-020 or equivalent documented procedure) must be handled in a manner consistent with J-STD-033 or an equivalent documented procedure. If cleaning solvents such as saponifier or DI water enter the vent holes, it can cause internal capacitors in the part to corrode and this corrosion can eventually impact proper part operation. Vented parts include, but may not be limited to Flip Chip FPGAs in the BGA configuration.
3. Bake the PCB to remove moisture which may, if not removed, precipitate measling or delamination.

REFERENCE

1.0 Foreword
2.1 Handling Electronic Assemblies
2.2 Cleaning
2.5 Baking and Preheating

EQUIPMENT REQUIRED

Hot gas reflow system
Gas focusing nozzle
Preheater

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

OPTIONAL EQUIPMENT

N/A

MATERIALS

Flux
Cleaner

PROCEDURE

NOTE: Some systems do not include integrated preheating capability and it may be necessary to preheat the PCB and BTC separately.

1. Place the PCB in the system work piece holder.
2. Inject flux underneath the BTC (optional).
3. Set hot gas reflow system to achieve the TTP defined by procedural analysis.
4. Perform alignment of gas nozzle to component locations.
5. Bring gas focusing nozzle in to reflow position.
6. Perform TTP re-flow cycle defined by procedural analyses.
7. Confirm solder melt of ALL joints and lift component from PCB.
8. Release component onto a heat resistant surface.
9. Allow PCB to cool.
10. Clean and inspect PCB as appropriate to customer requirements.

INSPECTION GUIDELINES

Visual examination of the following:

Solder connections
Board damage
Conductor damage
Components damage
Conformal coating or Solder resist damage

7711
Rework

Revision:
Date: 10/11

Number: **5.7.6**

BGA Reballing Procedure

Polyimide Solder Ball Stencil Carrier

Board Type: R, C
See 1.4.2
Skill Level: Advanced
See 1.4.3
Level of Conformance: High
See 1.5.1

GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

EQUIPMENT REQUIRED

Solder removal system
Convective reflow station

OPTIONAL EQUIPMENT

Reflow oven
Bake-out (vacuum, convection) oven

MATERIALS

Flux
Cleaner
Tissues/wipes
Polyimide solder ball stencil carrier to suit component

NOTE

Moisture sensitive components (as classified by IPC/JEDEC J-STD-020 or equivalent document procedure) must be handled in manner consistent with J-STD-033 or an equivalent documented procedure.

CAUTION

Verify component can withstand multiple reflow cycles. Multiple reflow cycles may degrade the component and the PWB.

PROCEDURE

1. Remove excess solder in accordance with procedures 4.1.2, 4.1.3, or 4.2.1 (see Figure 1).
2. Clean and inspect BGA for coplanarity.
3. Apply flux to lands on BGA (see Figure 2).
4. Place BGA on top of polyimide solder ball stencil carrier (solder balls up and white tape down) (see Figure 3).
5. Place BGA and carrier onto the reflow station or into a reflow oven and reflow using established profile.
6. Allow BGA to cool for 30 - 60 seconds and remove polyimide solder ball stencil carrier from BGA (see Figure 4).
7. Clean and inspect the BGA.
8. Bake BGA as necessary.

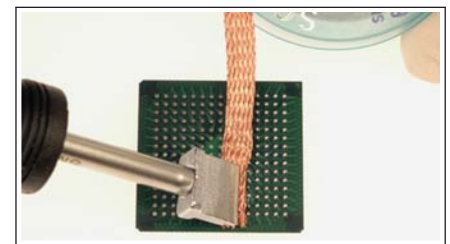


Figure 1

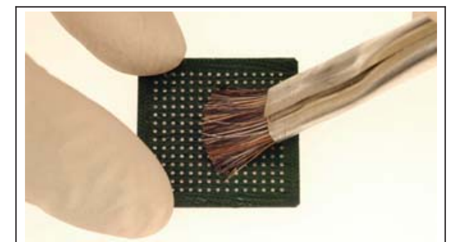


Figure 2

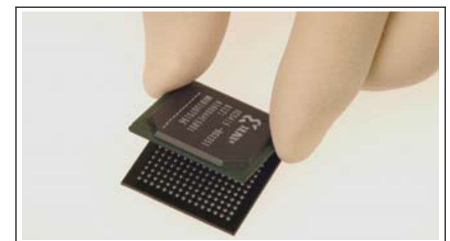


Figure 3

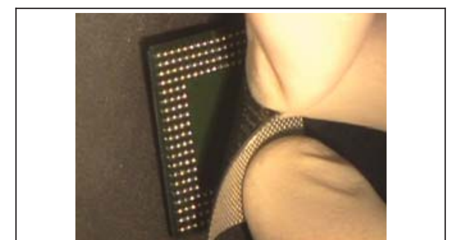
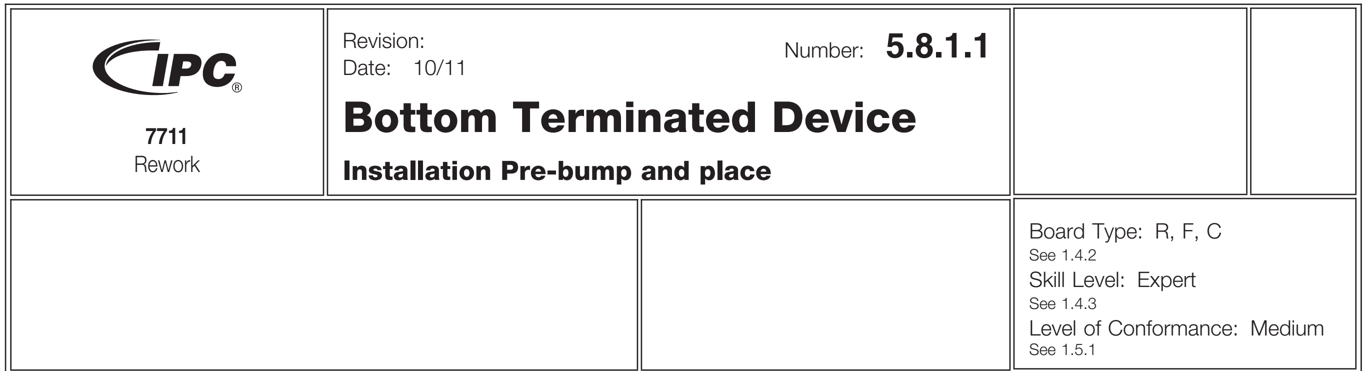


Figure 4

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

NOTES


IPC®

7711
Rework

Revision:
Date: 10/11

Number: **5.8.1.1**

Bottom Terminated Device

Installation Pre-bump and place

Board Type: R, F, C
See 1.4.2
Skill Level: Expert
See 1.4.3
Level of Conformance: Medium
See 1.5.1

GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

OUTLINE

The procedure outlined below is generic in nature and identifies the procedural steps which need to be accomplished to affect bottom terminated component (BTC) installation. This process requires the need for a split vision system for alignment of the BTC. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (system manufactures will customarily provide generalized operating procedures which must be further refined to achieve optimum results).

NOTE

The following preconditions shall be accomplished prior to performing the procedures:

1. Develop a time/temperature profile (TTP) for the specific BTC and BTC / PCB combination.
2 Moisture sensitive components (as classified by IPC/JEDEC J-STD-020 or equivalent documented procedure) must be handled in a manner consistent with J-STD-033 or an equivalent documented procedure.
3. Bake the PCB to remove moisture which may, if not removed, precipitate measling or delamination.

REFERENCE

1.0 Foreword
2.1 Handling Electronic Assemblies
2.2 Cleaning
2.5 Baking and Preheating

EQUIPMENT REQUIRED

Hot air, hot gas reflow system
Gas focusing nozzle (sized to package dimensions)
Gas supply (if other than ambient atmosphere)
Preheat method (oven, hotplate, high intensity lamp)
Handheld miniature Squeegee

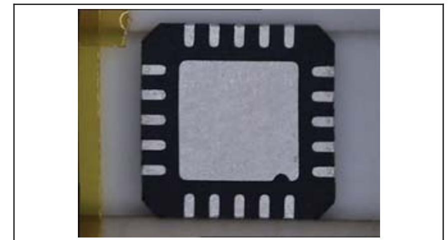


Figure 1

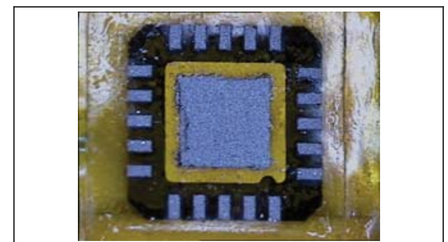


Figure 2

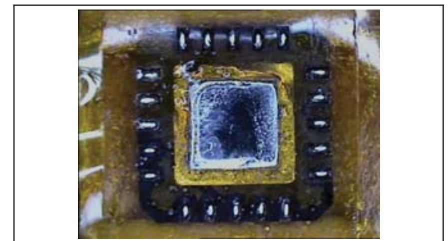


Figure 3

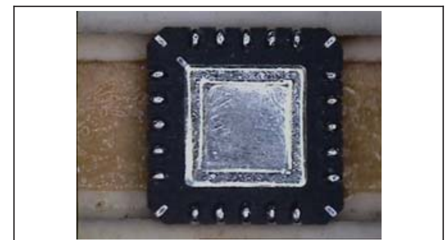


Figure 4

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

OPTIONAL EQUIPMENT

Bake-out oven (vacuum, convection)
X-RAY inspection system
Forced (ambient) air cooling system
Inert gas supply, if used
Microscope/vision system
Endoscope

MATERIALS

Solder paste
Cleaner
Stencils for Part Bumping
Kapton tape-ESD

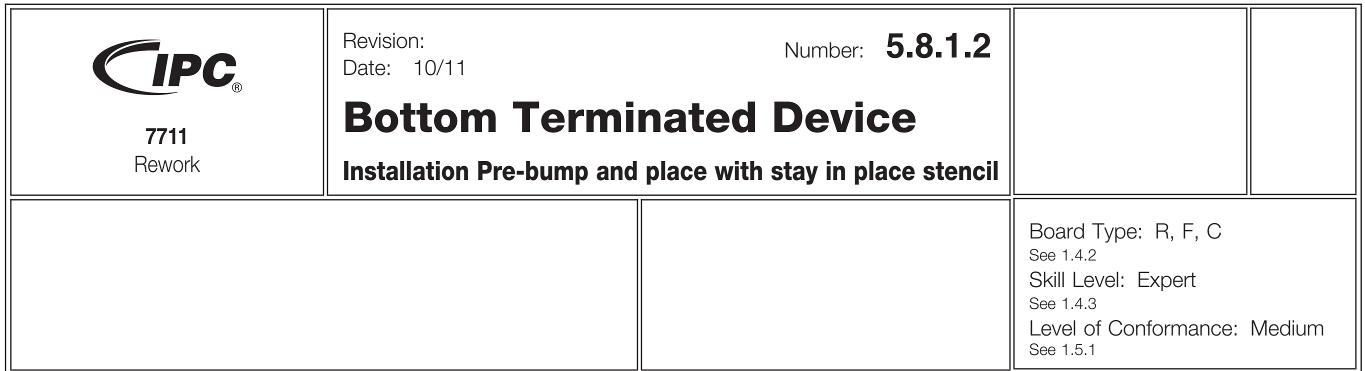
PROCEDURE

1. Remove solder from surface of part lands and clean (see Figure 1) (see procedures 4.1.1 or 4.1.3).
2. Place the part stencil, aligning it with part lands and applying pressure to the stencil.
3. Squeegee solder paste with a manual squeegee into the apertures. Clean off surface of stencil (see Figure 2).
4. Set hot gas reflow system to achieve the TTP of the BTC defined by procedural analysis.
5. Perform alignment of gas nozzle to component size.
6. Bring gas focusing nozzle in to reflow position.
7. Perform TTP reflow cycle defined by procedural analyses (see Figure 3).
8. Remove stencil from device and clean per customer requirements (see Figure 4).
9. Inspect to make sure the ''bumps'' are uniform and consistent and are on all of the pads.
10. Coat the lands on the board with flux.
11. Set hot gas reflow system to achieve the TTP defined by procedural analysis.
12. Perform alignment of gas nozzle to component location.
13. Bring gas focusing nozzle in to reflow position.
14. Perform BTC/PCB combination TTP reflow cycle defined by procedural analysis.
15. Perform accelerated cooling cycle if appropriate or allow to cool.
16. Clean PCB as appropriate to customer requirements.
17. Perform X-Ray, endoscopic and other inspection of PCB as appropriate.

INSPECTION GUIDLINES

Visual examination of the following:

Solder connections
Board damage
Conductor damage
Components damage
Conformal coating or Solder resist damage



Bottom Terminated Device

Installation Pre-bump and place with stay in place stencil

Board Type: R, F, C
See 1.4.2
Skill Level: Expert
See 1.4.3
Level of Conformance: Medium
See 1.5.1

GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

OUTLINE

The procedure outlined below is generic in nature and identifies the procedural steps which need to be accomplished to affect bottom termination component (BTC) installation. This process has been developed to eliminate the need for a split vision system for alignment of the BTC. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (system manufactures will customarily provide generalized operating procedures which must be further refined to achieve optimum results).

NOTE

The following preconditions shall be accomplished prior to performing the procedures:

1. Develop a time/temperature profile (TTP) for the specific BTC and BTC / PCB combination.
2. Moisture sensitive components (as classified by IPC/JEDEC J-STD-020 or equivalent documented procedure) must be handled in a manner consistent with J-STD-033 or an equivalent documented procedure
3. Bake the PCB to remove moisture which may, if not removed, precipitate measling or delamination.

REFERENCE

1.0 Foreword
2.1 Handling Electronic Assemblies
2.2 Cleaning
2.5 Baking and Preheating

EQUIPMENT REQUIRED

Hot air or hot gas reflow system (Example: tabletop oven)
Gas focusing nozzle (sized to package dimensions)
Gas supply (if other than ambient atmosphere)
Preheat method (oven, hotplate, high intensity lamp)
Handheld miniature squeegee

OPTIONAL EQUIPMENT

Bake-out oven (vacuum, convection)
X-RAY inspection system
Forced (ambient) air cooling system
Inert gas supply, if used
Microscope/vision system
Endoscope

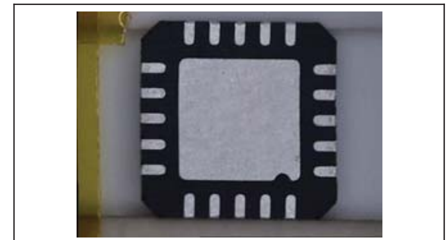


Figure 1

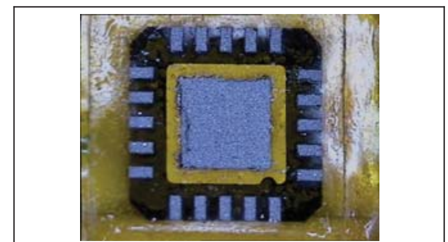


Figure 2

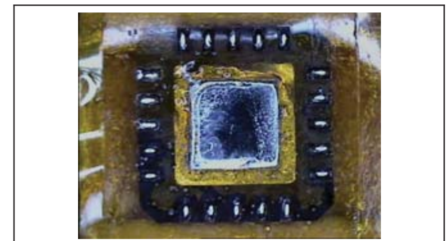


Figure 3

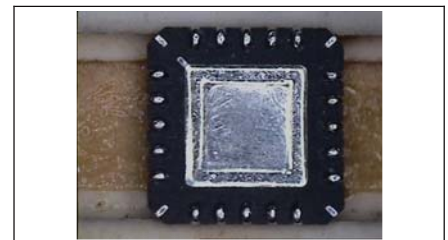


Figure 4

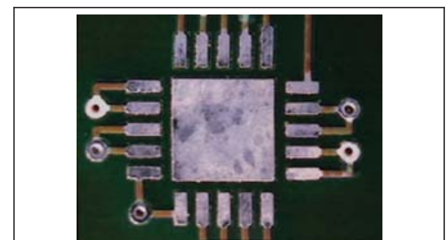


Figure 5

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

MATERIALS

Solder paste
Cleaner

PROCEDURE

1. Remove solder from surface of part lands and clean (see Figure 1) (see 4.1.1 or 4.1.2).
2. Place the part stencil aligning it with part lands and applying pressure to the stencil.
3. Squeegee solder paste with a manual squeegee into the apertures. Clean off surface of stencil (see Figure 2).
4. Set hot gas reflow system to achieve the TTP of the BTC defined by procedural analysis.
5. Perform alignment of gas nozzle to component.
6. Bring gas focusing nozzle in to reflow position.
7. Perform TTP reflow cycle defined by procedural analysis (see Figure 3).
8. Remove stencil from device and clean per customer requirements.
9. Inspect to make sure the ''bumps'' are uniform and consistent and are on all of the lands (see Figure 4).
10. Clean up lands on board by wicking or scavenging and cleaning (see Figure 5).
11. Place the board stay in place polyimide stencil onto the board by aligning and applying pressure to the stencil.
12. Squeegee solder paste with a manual squeegee into the apertures. Clean off surface of stencil (see Figure 6).

    CAUTION: The surface of the stencil must be free of particulate matter.
13. Place the bumps of the BTC into the solder pasted apertures of the stencil (see Figure 7).
14. Set hot gas reflow system to achieve the TTP of the BTC/PCB combination defined by procedural analysis.
15. Alignment the hot air source to component locations.
16. Perform TTP reflow cycle defined by procedural analyses.
17. Perform accelerated cooling cycle if appropriate or allow to cool
18. Clean PCB as appropriate to customer requirements.
19. Perform X-Ray, endoscopic and other inspection of PCB as appropriate (see Figure 8).

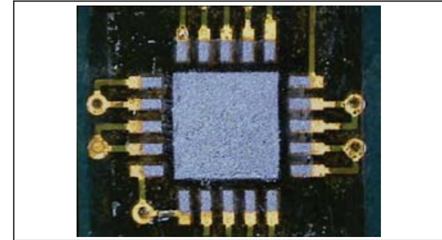


Figure 6

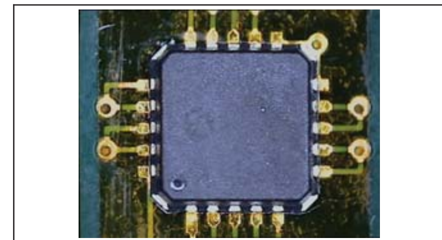


Figure 7

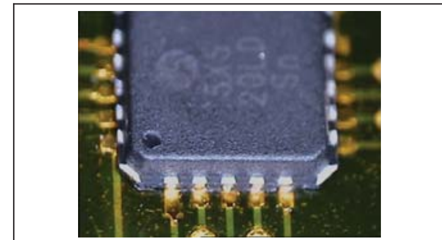
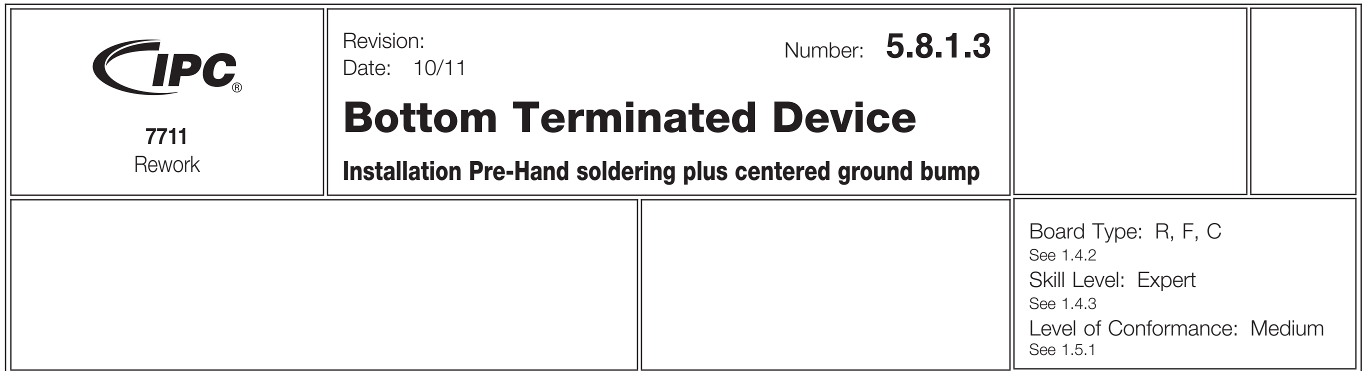


Figure 8

INSPECTION GUIDELINES

Visual examination of the following:

Solder connections
Board damage
Conductor damage
Components damage
Conformal coating or Solder resist damage



IPC ®

7711
Rework

Revision:
Date: 10/11

Number: **5.8.1.3**

Bottom Terminated Device

Installation Pre-Hand soldering plus centered ground bump

Board Type: R, F, C
See 1.4.2
Skill Level: Expert
See 1.4.3
Level of Conformance: Medium
See 1.5.1

GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

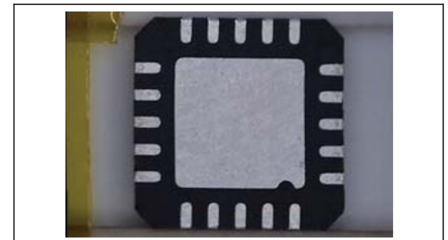


Figure 1

Figure 2

OUTLINE

The procedure outlined below is generic in nature and identifies the procedural steps which need to be accomplished to affect bottom termination component (BTC) installation for device with land patterns that extend beyond the footprint. This process has been developed to eliminate the need for a split vision system hot air system. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (system manufactures will customarily provide generalized operating procedures which must be further refined to achieve optimum results).

NOTE

The following preconditions shall be accomplished prior to performing the procedures:

1. Develop a time/temperature profile (TTP) for the specific BTC and BTC / PCB combination
2. Moisture sensitive components (as classified by IPC/JEDEC J-STD-020 or equivalent documented procedure) must be handled in a manner consistent with J-STD-033 or an equivalent documented procedure
3. Bake the PCB to remove moisture which may, if not removed, precipitate measling or delamination.

REFERENCE

1.0 Foreword
2.1 Handling Electronic Assemblies
2.2 Cleaning
2.5 Baking and Preheating

EQUIPMENT REQUIRED

Localized hot air gun or hand held or hot gas reflow system
Gas supply (if other than ambient atmosphere)
Preheat method (oven, hotplate, high intensity lamp)

OPTIONAL EQUIPMENT

Bake-out oven (vacuum, convection)
X-RAY inspection system
Microscope/vision system
Endoscope

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

MATERIALS

Soldering iron
Solder-resin cored wire solder
Cleaner
Kapton tape-ESD

PROCEDURE

CAUTION: Care must be taken to ensure full reflow of the paste in the thermal pad.

1. Remove solder from surface of part lands and clean (see Figure 1) (see 4.1.1 or 4.1.2).
2. Bump center ground/heat slug area of the PCB with solder allowing the solder to flow such that the device is bumped to a total height consistent with this devices' overall standoff distance prior to the removal of the device.
3. Set hot gas reflow system to achieve the TTP of the BTC/PCB combination defined by procedural analysis.
4. Place the component and align to the PWA.
5. Perform alignment of gas nozzle to component.
6. Bring gas focusing nozzle in to reflow position.
7. Perform TTP reflow cycle defined by procedural analyses.
8. Measured to make sure standoff height is consistent with the device prior to removal.
9. Clean per customer requirements.
10. Inspect center ground/heat dissipation pattern to make sure that make sure the ''bumps'' are uniform and consistent.
11. Using a small chisel tip solder the peripheral IO leads to the pads on the PCB (see Figure 2).
12. Clean and inspect PCB as appropriate to customer requirements.
13. Perform X-Ray, endoscopic and other inspection of PCB as appropriate.

INSPECTION GUIDELINES

Visual examination of the following:

Solder connections
Board damage
Conductor damage
Components damage
Conformal coating or Solder resist damage

7721

Modification and Repair

Revision:

Date: **10/11**

Number: **4.7.4**

Surface Mount, BGA Land with Integral Via Repair Film Adhesive Method

Board Type: R, F, C
See 1.4.2
Skill Level: Expert
See 1.4.3
Level of Conformance: Medium
See 1.5.1

GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

OUTLINE

This method is used to replace damaged BGA lands with new dry film adhesive backed lands. The new lands are bonded to the circuit board surface using a specially designed bonding press or bonding iron. Electrical connection is made to a via which is originally manufactured integral to the land. (See Figure 1.)

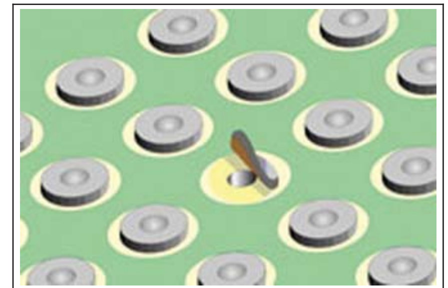


Figure 1 **Damaged BGA Land.**

NOTE

This method uses replacement BGA lands. The new lands are fabricated from copper foil. They are available in a variety of sizes and shapes and are generally supplied solder plated. If a special size or shape is needed they can be custom fabricated.

REFERENCE

1.0 Foreword
2.1 Handling Electronic Assemblies
2.2 Cleaning
2.5 Baking and Preheating
2.6 Epoxy Mixing and Handling

Figure 2 **Remove the defective land and prepare connecting via.**

TOOLS & MATERIALS

BGA Land Repair Kit
Bonding Iron
Bonding Tips
Bonding System
Circuit Frames, BGA Lands
Cleaner
Epoxy
Flux, Liquid
Knife
Microscope
Oven
Scraper
Solder
Soldering Iron
Tape, Kapton
Tweezers
Wipes

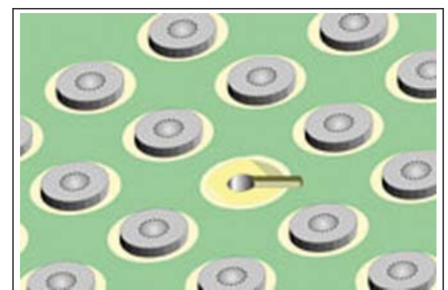


Figure 3 **Channel or canal recessed into board to permit circuit routing to be level with board surface.**

PROCEDURE

1. Clean the area.
2. Carefully remove the defective land to avoid damaging the integral via.
3. Use a knife and scrape off any epoxy residue, contamination or burned material from the board surface.
4. Scrape off any residue from the exposed via. (See Figure 2.)
5. Clean the area.

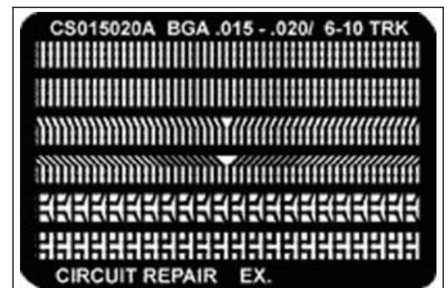


Figure 4 **Select a replacement land that matches the missing land.**

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

6. Apply a small amount of liquid flux to the via connection on the board surface. Using the appropriate size solder braid remove solder from the via. Clean the area.
7. In general, the area for the new land on the board surface must be smooth and flat. If internal fibers of the board are exposed, or if there are deep scratches in the surface, they should be repaired. Refer to procedure 3.5.1. In this procedure a channel or canal must be recessed into the board from the via location to the outside edge of the affected land. This permits the via to land connecting circuit adequate space for routing without interfering with the pressing of the new BGA land. (See Figure 3.)
8. Select a replacement BGA land that most closely matches the surface mount land to be replaced. (See Figure 4.) If a special size or shape is needed they can be custom fabricated.

   NOTE

   New BGA lands are fabricated from copper foil. The foil is plated on the top side with solder, and an adhesive bonding film is applied to the bottom side.
9. Before trimming out the new land, carefully scrape off the adhesive bonding film from the solder joint connection area on the back of the new land. In order to ensure that the final pressed land lays flat, it may be necessary to remove the film from the whole length of the connecting circuit up to the land itself. (See Figure 5.)

   CAUTION

   When handling the replacement land, avoid touching the adhesive backing with your fingers or other materials that may contaminate the surface and reduce the bond strength.
10. Cut out and trim the new land. Cut out from the plated side.
11. Turning the land upside down, place the land on the board so that the connecting circuit can be fit into the via. Ensure the connecting circuit is running over the previously cut channel or canal. (See Figure 6.) The connecting circuit of the new BGA land will be inserted into the integral via hole of the original BGA land.
12. Insert the connecting circuit into the via and carefully flux the connection area.
13. Solder the connecting circuit into the via.
14. Lay the connecting circuit into the bottom of the trough. If required, apply a securing film of high strength thermosetting epoxy over the connecting circuit to secure in place and provide a flat surface on which to press the new BGA land.
15. Fold over the new BGA land 180° (**CAUTION:** ensure no damage to the replacement conductor) and place the new land into position on the circuit board surface using high temperature tape to help in alignment.
16. Select a bonding tip with a shape to match the shape of the new land. See bonding tip chart in the replacement parts section of the manual provided with the repair system or repair kit. (See Figure 7.)

    NOTE

    The tip used for bonding should be as small as possible but should completely cover the entire surface of the new land.

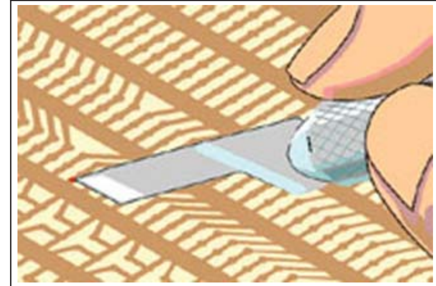


Figure 5 Scrape off the adhesive bonding film from the solder joint connection area on the back of new land.

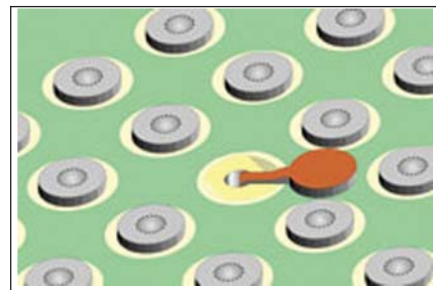


Figure 6 Connecting circuit is fit into via.

Figure 7 Bond the new land using a Bonding System.

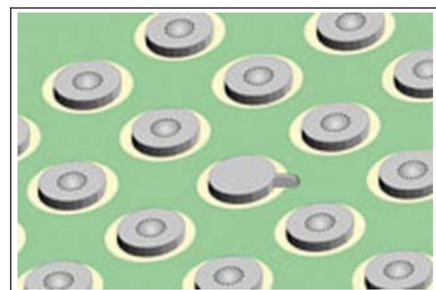


Figure 8 Completed repair.

17. Position the circuit board so that it is flat and stable. Gently place the hot bonding tip onto the Kapton tape covering the new land. Apply pressure as recommended in the manual of the repair system or repair kit for 5 seconds to tack the new land in place. Carefully peel off the tape.

    CAUTION

    Excessive bonding pressure may cause measling in the circuit board surface or may cause the new land to slide out of position.

18. Gently place the bonding tip directly onto the new land. Apply pressure as recommended in the manual of the repair system or repair kit for an additional 30 seconds to fully bond the land. After the bonding cycle remove the tape used for alignment. The new land is fully cured. Carefully clean the area and inspect the new land for proper alignment. (See Figure 8.)
19. Additional epoxy can be applied around the perimeter of the new land to provide additional bond strength. Mix epoxy. Cure the epoxy per Procedure 2.6 Epoxy Mixing and Handling. Use the maximum recommended heat cycle to ensure the highest strength bond. BGA lands are routinely subjected to one or more reflow cycles.

    CAUTION

    Some components may be sensitive to high temperature.

20. Apply surface coating to match prior coating as required.

EVALUATION

1. Visual examination of solder area, conductor and laminate..
2. Measurement of new land width and spacing.
3. Electrical continuity measurement.

NOTES

7721

Modification and Repair

Revision:

Date: **10/11**

Number: **4.7.5**

Surface Mount, BGA Land with Integral Via Repair Circuit Extension Film Adhesive Method

Board Type: R, F, C
See 1.4.2
Skill Level: Expert
See 1.4.3
Level of Conformance: High
See 1.5.1

GENERAL REQUIREMENTS

Clauses 1.7 (Basic Considerations), 1.8 (Workstations, Tools, Materials and Processes) and 1.9 (Lead Free) provide important information and guidance about the use of this procedure, including but not limited to tin-lead and lead-free alloys. This procedure is also applicable to lead free products.

OUTLINE

This method is used to replace damaged BGA lands with new dry film adhesive backed lands. The new lands are bonded to the circuit board surface using a specially designed bonding press or bonding iron. Electrical connection is made to a via circuit which is originally manufactured integral to the lands.

NOTE

This method uses replacement BGA lands. The new lands are fabricated from copper foil. They are available in a variety of sizes and shapes and are generally supplied solder plated. If a special size or shape is needed they can be custom fabricated.

REFERENCE

1.0 Foreword
2.1 Handling Electronic Assemblies
2.2 Cleaning
2.5 Baking and Preheating
2.6 Epoxy Mixing and Handling

TOOLS & MATERIALS

Ball Mills
Bonding Iron
Bonding Tips
Bonding System
Circuit Frames, BGA Lands
Cleaner
Epoxy
Flux, Liquid Knife
Micro-Drill System
Micro Pad Repair Kit
Microscope
Oven
Scraper
Solder
Soldering Iron
Tape, Kapton
Tweezers
Wipes

PROCEDURE

NOTE

Prior to proceeding determine whether this procedure or 4.7.4 Surface Mount, BGA Land with Integral Via Repair, is the appropriate method. This method is best used when a connecting circuit or plane clearly runs from the integral via and is close to the board's affected surface to permit exposure of the circuit for the repair connection.

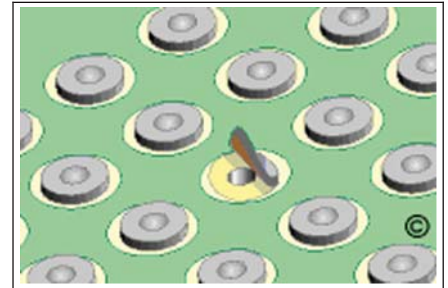


Figure 1 Remove the damaged land with integral via.

Figure 2 Micro-Drill System and ball mills.

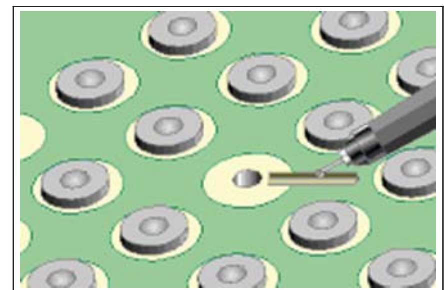


Figure 3 Using a Micro-Drill expose the copper circuit by recessing a Channel or canal.

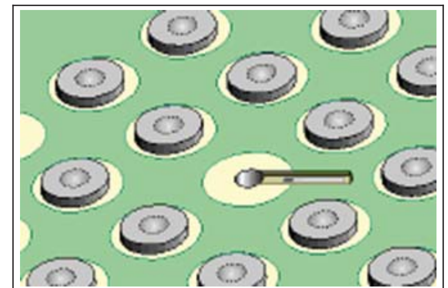


Figure 4 Tin the circuit with solder.

Material in this manual, IPC-7711/7721 Rework, Modification and Repair of Electronic Assemblies, was voluntarily established by Technical Committees of IPC. This material is advisory only and its use or adaptation is entirely voluntary. IPC disclaims all liability of any kind as to the use, application, or adaptation of this material. Users are also wholly responsible for protecting themselves against all claims or liabilities for patent infringement. Equipment referenced is for the convenience of the user and does not imply endorsement by IPC.

1. Clean the area.
2. Carefully remove the defective land to avoid damaging the integral via (see Figure 1).
3. Use a knife and scrape off any epoxy residue, contamination or burned material from the board surface.
4. Using a Micro-Drill system (see Figure 2) and the appropriate ball mill down to the connecting circuit, exposing the copper surface of the etch (see Figure 3). Be careful not to cut through the circuit.
5. Clean the area
6. Apply a small amount of liquid flux to the circuit connection now exposed beneath the surface of the board. Tin the circuit with solder (see Figure 4). Clean the area.
7. In general, the area for the new land on the board surface must be smooth and flat. If internal fibers of the board are exposed, or if there are deep scratches in the surface, they should be repaired. Refer to procedure 3.5.1.
8. Select a replacement BGA land that most closely matches the surface mount land to be replaced. If a special size or shape is needed they can be custom fabricated (see Figure 5). Ensure the selected lands attached circuit matches the width and thickness of the circuit to be replaced. Trim the width approximately as needed. The circuit should overlap the existing circuit a minimum of 2 times the circuit width.

   NOTE

   New BGA lands are fabricated from copper foil. The foil is plated on the top side with solder, and an adhesive bonding film is applied to the bottom side.
9. Before trimming out the new land, carefully scrape off the adhesive bonding film from the solder joint connection area on the back of the replacement circuit portion of the new land. In order to ensure that the final pressed land lays flat (and the new circuit is properly connected to the exposed board circuit), it may be necessary to remove the film from the whole length of the connecting circuit up to the land itself (see Figure 6).

   CAUTION

   When handling the replacement land, avoid touching the adhesive backing with your fingers or other materials that may contaminate the surface and reduce the bond strength.
10. Cut out and trim the new land. Cut out from the plated side.
11. Place the land on the board so that the connecting circuit can be fit into the trough containing the board's tinned circuit (see Figure 7).
12. Select a bonding tip with a shape to match the shape of the new land. See bonding tip chart in the replacement parts section of the manual provided with the repair system or repair kit (see Figure 8).

    NOTE

    The tip used for bonding should be as small as possible but should completely cover the entire surface of the new land.
13. Position the circuit board so that it is flat and stable. Gently place the bonding tip directly onto the new land. Apply pressure as recommended in the manual

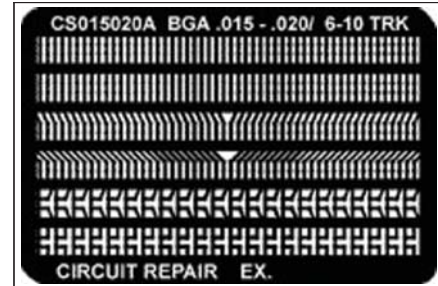


Figure 5 Select a replacement land that matches the missing land.

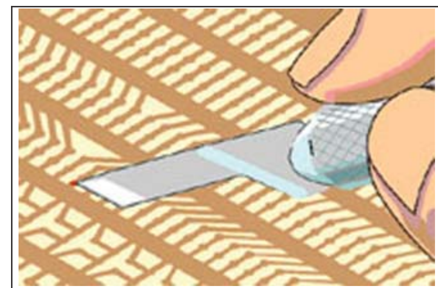


Figure 6 Scrape off the adhesive bonding film from the solder joint connection area on the back of new land.

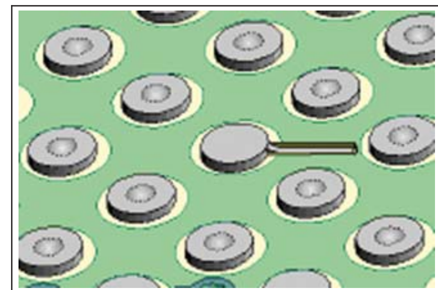


Figure 7 Connecting circuit connected to the exposed and tinned circuit.

Figure 8 Bond the new land using a Bonding System.

of the repair system or repair kit for 30 seconds to fully bond the land. The new land is fully cured. Carefully clean the area and inspect the new land for proper alignment.

CAUTION

Excessive bonding pressure may cause measling in the circuit board surface or may cause the new land to slide out of position.

14. Apply flux to the overlapping connecting circuit in the bottom of the trough. Solder in place. Clean the area

NOTE

The circuit board may be preheated prior to filling the area with epoxy. A preheated circuit board will allow the epoxy to easily flow and level out. Epoxy applied to an unheated circuit board may settle below the circuit board surface as the epoxy cures.

15. Mix epoxy. If desired, add color agent to the mixed epoxy to match the circuit board color.
16. Coat the top and sides of the replaced circuit with epoxy. The epoxy bonds the new circuit to the base board material and insulates the circuit. Continue adding epoxy up to the top surface of the circuit board.

NOTE

A very slight overfill of epoxy may be desired to allow for shrinkage when the epoxy cures. Additionally, in order to allow the circuits appearance to match the board, it may be necessary to mask the circuit prior to filling the excavated area.

17. Cure the epoxy per Procedure 2.6 Epoxy Mixing and Handling

CAUTION

Some components may be sensitive to high temperature.

18. Additional epoxy can be applied around the perimeter of the new land to provide additional bond strength. Mix epoxy. Cure the epoxy per Procedure 2.6 Epoxy Mixing and Handling. Use the maximum recommended heat cycle to ensure the highest strength bond. BGA lands are routinely subjected to one or more reflow cycles.
19. Apply surface coating to match prior coating as required (see Figure 9).

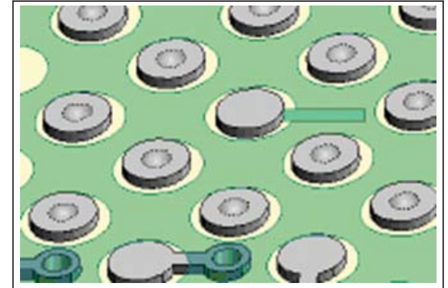


Figure 9 Completed repair.

EVALUATION

1. Visual examination of solder area, conductor and laminate.
2. Measurement of new land width and spacing.
3. Electrical continuity measurement.

NOTES