Association Connecting Electronics Industries

IPC-7711B/7721B Change 2
March 2014

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 the IPC-7711A/7711A.

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

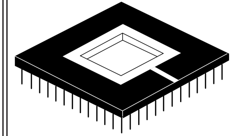
7711
Rework

Revision: C
Date: 03/14

Number: 3.2.1

PGA and Connector Removal

Solder Fountain Method



Board Type: R, F, W, 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.

EQUIPMENT REQUIRED

Solder fountain
Chimney or nozzle to match part
Removal tool
Pallet to hold board over fountain
Preheat oven

OPTIONAL EQUIPMENT

Vacuum pick-up tool

MATERIALS

Flux-cored solder
Cleaner
Heat resistant, antistatic gloves
Protective face gear
Heat resistant tape

CAUTION: Copper dissolution is a concern when using a solder fountain and can be impacted by dwell time, temperature and alloy in use.

PROCEDURE

This process is for experienced operators only. Caution must be exercised due to working with hot, molten solder.

1. Remove conformal coating (if any) and clean work area of any contamination, oxides or residues.
2. Set solder fountain pot control to the required temperature for removing that particular component from that particular board. Wait until solder pot reaches the set temperature.
3. Attach the correct nozzle or chimney to the solder pot. (See Figure 1.)
4. Set the timer for the amount of time the fountain is to be running for that particular part.
5. The area around the rework site may be masked with a high temperature resistant tape, or similar material, to protect the adjacent area during rework. (See Figure 2.)

Figure 1 Attach Nozzle

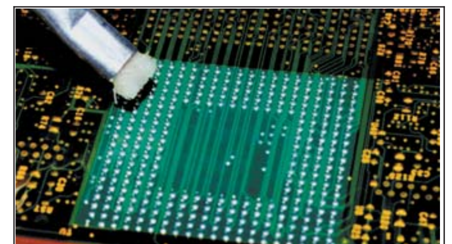


Figure 2 Flux

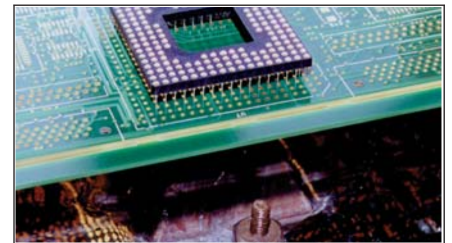


Figure 3 Place Over Solder Fountain

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. Preheat the board to the desired temperature, depending on the component restrictions and the board T_g material.
7. Flux the bottom side site where the part will be removed. (See Figure 2.)
8. Place the board on the pallet over the solder fountain and trip the timer. (See Figure 3.)
9. At the end of the timer cycle, use vacuum pickup tool, tweezers, or removal tool to remove the part from the board.
10. Clean the flux residue, if required, and inspect.
11. Clean as applicable and inspect to established workmanship requirements.

NOTES

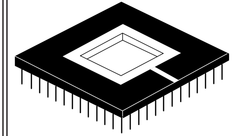
7711
Rework

Revision: C
Date: 03/14

Number: **5.2.1**

PGA and Connector Installation

Solder Fountain Method with PTH Prefilled



Board Type: R, F, W, 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.

EQUIPMENT REQUIRED

Solder fountain
Chimney or nozzle to match part
Removal tool
Pallet to hold board over fountain
Preheat oven

MATERIALS

Flux
Cleaner
Heat resistant, antistatic gloves
Protective face gear
Heat resistant tape

CAUTION: Copper dissolution is a concern when using a solder fountain and can be impacted by dwell time, temperature and alloy in use.

PROCEDURE

This procedure variation is for components or connectors with sturdy leads that do not readily bend.

This process is for experienced operators only. Caution must be exercised due to working with hot, molten solder.

1. Attach the correct nozzle or chimney to the solder pot. This operation must be done with proper care per solder fountain supplier's instructions. (See Figure 1.)
2. Set solder fountain pot control to the required temperature for soldering that particular component into that particular board. Wait until solder pot reaches the set temperature.
3. Set the timer (if applicable) for the amount of time the fountain is to be running for that particular part.
4. The area around the rework site may be masked with a high temperature resistant tape, or similar material, to protect the adjacent area during rework. (See Figure 2.)
5. Preheat the new component and the board to the desired temperature, taking into consideration component thermal restrictions and glass transition temperature T_g of the board material.

Figure 1 Attach Nozzle

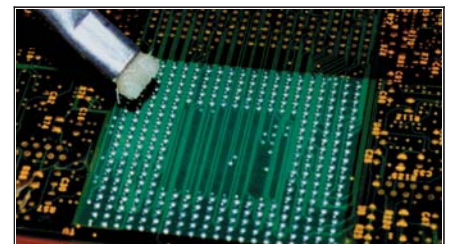


Figure 2 Flux

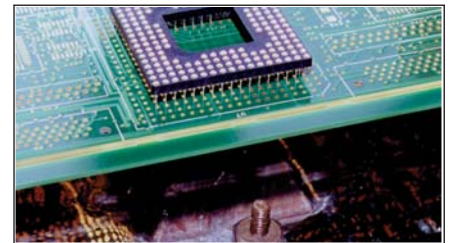


Figure 3 Place Over Solder Fountain

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. Flux the board on the top and bottom side at the site of the new component. The component leads may also be fluxed, depending on the board and component leads. Place the component on the board in its correct site. (See Figure 2.)
7. Place the board on the pallet, over the solder fountain with the component sitting in location and trip the solder fountain timer. (See Figure 3.)
8. As the solder in the holes reflows, the component may have to be reoriented to drop into the holes.
9. At the end of the timer cycle, wait at least 5 seconds for the solder to solidify, then remove the board.
10. Clean as applicable and inspect to established workmanship requirements.

NOTES

7711
Rework

Revision:
Date: 03/14

Number: **5.7.2.1**

BGA/CSP Installation

Stay-in-Place Stencil

Board Type: R, F, C
See 1.4.2
Skill Level: Advanced
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 BGA/CSP installation. This process has been developed for easy placement of BGA with no high end rework system required. This procedure also eliminates the need to repair damaged mask underneath the BGA/CSP prior to installation. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (system manufacturers will customarily provide generalized operating procedures which must be further refined to achieve optimal results).

EQUIPMENT REQUIRED

Reflow Source (Reflow Oven, Controlled Hot Air, or Other Source)
Squeegee
Tweezers

OPTIONAL EQUIPMENT

Bake-out Oven
X-ray Inspection System
Convective Reflow System

MATERIALS

Stay-in-place Stencil
Solder Paste
Cleaner

NOTE

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.

REFERENCE

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

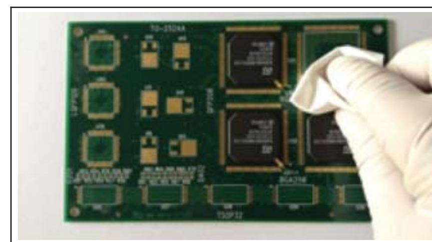


Figure 1 Clean Work Area

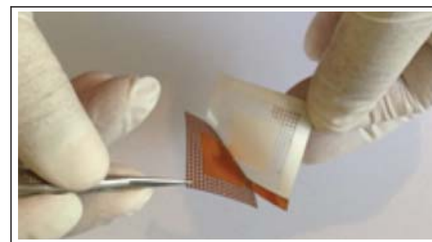


Figure 2 Remove the Liner

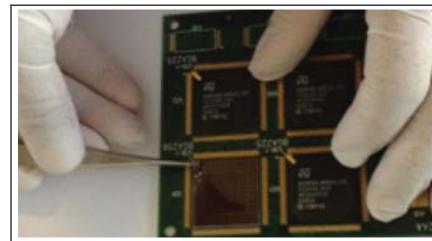


Figure 3 Align Stencil Over Land Pattern

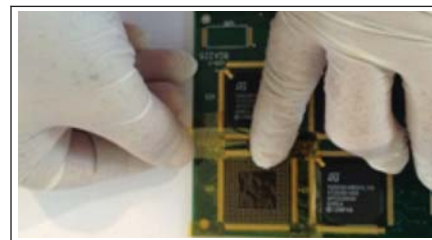


Figure 4 Tape Around Periphery of Stencil

Figure 5 Squeegee Solder Paste

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.

Number: **5.7.2.1**	Subject: **BGA/CSP Installation**
Revision: Date: 03/14	

PROCEDURE

1. Clean work area of any solder, contamination, oxides, residues or fluxes (see Figure 1).
2. Select appropriate stay-in-place Stencil that matches land patterns of component being installed.
3. Remove the release liner from bottom of stencil (see Figure 2).
4. Align the stencil over the land patterns, as per reference designator, by starting in one corner and working in X and Y directions until alignment is achieved (see Figure 3).
5. Lightly press on liner after alignment to activate the adhesive.
6. Tape off the area around the periphery of the stencil (see Figure 4).
7. Apply a small line of solder paste, the length of one side of the stencil, over the top row of apertures.
8. Squeegee solder paste across the top of the stencil making sure all of the apertures of the stencil are filled (see Figure 5).
9. Remove the tape applied to the periphery.
10. Wipe off excess solder paste from the top of the stencil with a lint free cloth (see Figure 6).
11. Ensure all apertures are filled with solder paste.
12. Install component, as per reference designator, and check alignment (see Figure 7).
13. Reflow solder paste with a reflow station or reflow oven using an established time, temperature profile.
14. Clean as applicable
15. Inspect to established workmanship requirements.

Figure 6 Wipe off excess solder paste

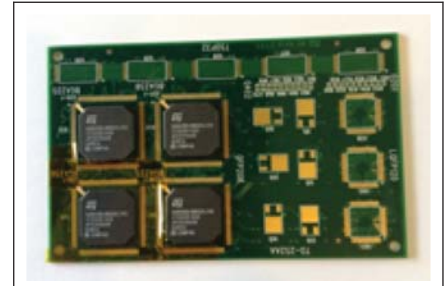


Figure 7 Install component

NOTES

7721

Modification
and Repair

Revision: **A**

Date: **03/14**

Number: **4.7.4**

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

Board Type: R, F
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.)

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

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

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.

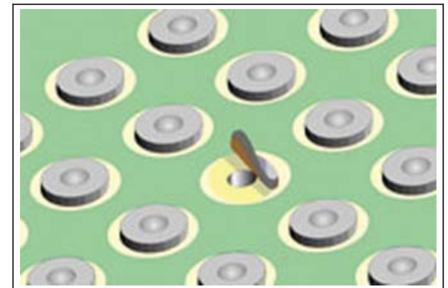


Figure 1 Damaged BGA Land.

Figure 2 Remove the defective land and prepare connecting via.

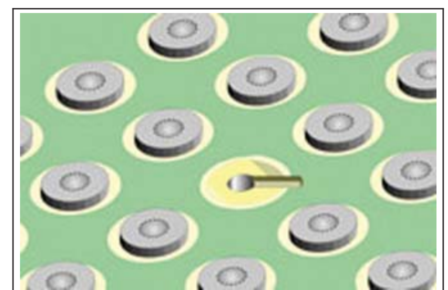


Figure 3 Channel or canal recessed into board to permit circuit routing to be level with board surface.

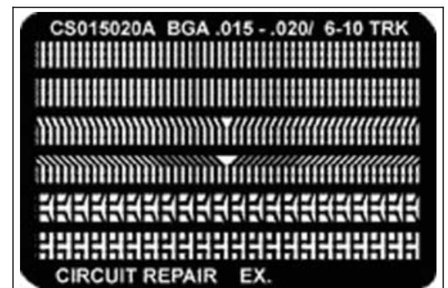


Figure 4 Select a replacement land that matches the missing land.

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.

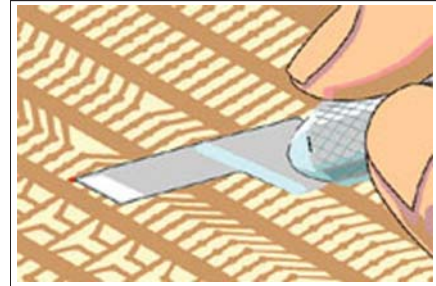


Figure 5 Scrape off the adhesive bonding film from the solder joint connection area on the back of new land.

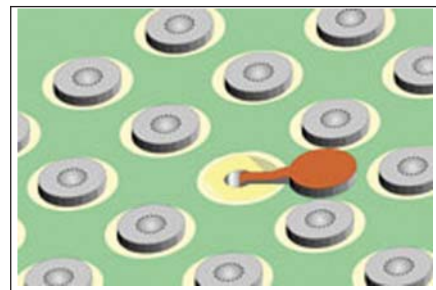


Figure 6 Connecting circuit is fit into via.

Figure 7 Bond the new land using a Bonding System.

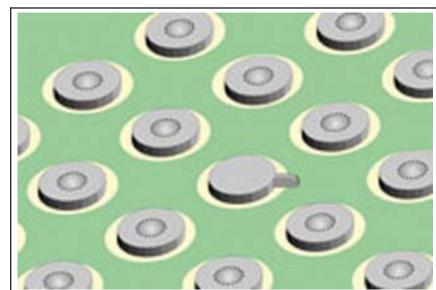


Figure 8 Completed repair.

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

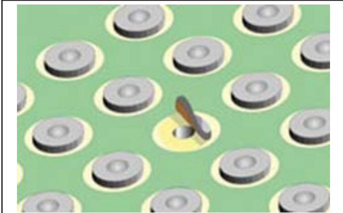
Modification and Repair

Revision:

Date: **03/14**

Number: **4.7.4.1**

Surface Mount Pad with Integral Via Repair Film Adhesive Method - No Conductor Bend



Board Type: R,F
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 pads with new dry film adhesive backed pads. The new pads are bonded to the circuit board surface using a bonding press or bonding iron. Electrical connection is made to a via which is originally manufactured integral to the pad.

NOTE

This method uses replacement pads. The new pads 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.

REFERENCES

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

TOOLS & MATERIALS

Pad Repair Kit
Bonding Iron
Bonding Tips
Bonding System
Circuit Frames, Pads
Cleaner
Epoxy
Flux, Liquid
Knife
Microscope
Oven
Scraper
Solder
Soldering Iron
Tape, Heat Resistant Polyimide
Tweezers
Wipes

PROCEDURE

1. Clean the area.
2. Carefully remove the defective pad 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 1).

Figure 1 Remove the defective land and prepare connecting via.

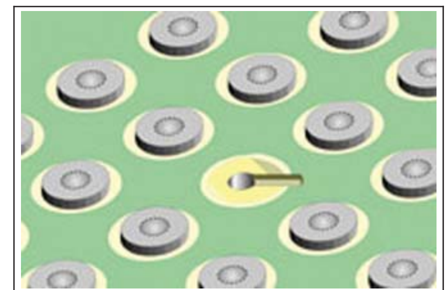


Figure 2 Channel or canal recessed into board to permit circuit routing to be level with board surface.

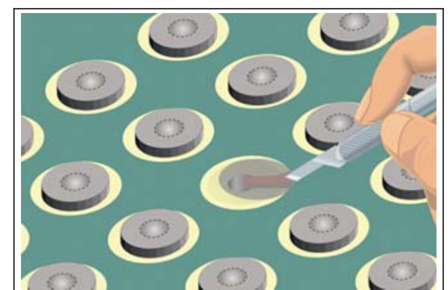


Figure 3 Trim epoxy from the tail.

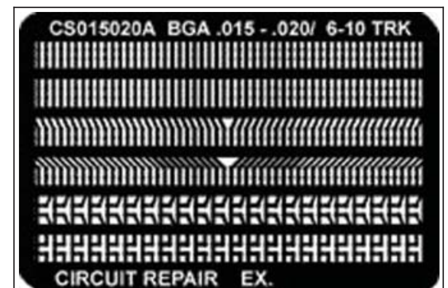


Figure 4 Select a replacement land that matched the missing land.

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.

5. Clean the area.
6. Clean the inside of the via using the appropriate tool. Being careful not to damage the walls of the via. Some of the copper inside the via must be exposed for the solder attachment.
7. In general, the area for the new pad 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. In this procedure a channel or canal must be recessed into the board from the via location to the outside edge of the affected pad using a precision drill system or equivalent. This permits the via to pad connecting circuit adequate space for routing without interfering with the pressing of the new pad (see Figure 2). The canal should be wide enough to accommodate the replacement conductor and allow room for working/soldering of the connection.
9. Insert a piece of the conductor/land material that is capable of fitting inside the via and solder.
10. Clean the residue with alcohol and let dry.
11. Apply epoxy over the top of the whole repair. Epoxy and repair must be flat over the top of the repaired area.
12. Let epoxy cure per manufacturer's instructions. This could be from an oven or letting it set for 24 hours.
13. Trim a small amount of the epoxy off the end of the land (tail) that is away from the via (see Figure 3). Tin it. This will be used for the attachment of the pad.
14. Select a replacement pad that most closely matches the surface mount pad to be replaced (see Figure 4). If a special size or shape is needed they can be custom fabricated.

    NOTE

    New pads 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. The new conductor material may need to be trimmed to fit the size of the damaged or missing circuitry.
15. Before trimming out the new pad, carefully scrape off the dry film epoxy from the area that will be soldered to the embedded conductor.

    CAUTION

    When handling the replacement pad, avoid touching the adhesive backing with your fingers or other materials that may contaminate the surface and reduce the bond strength.
16. Cut out and trim the new pad. Cut out from the plated side.

    CAUTION

    Ensure that the epoxy and repair area are smooth and flat when cured (see Figure 5).

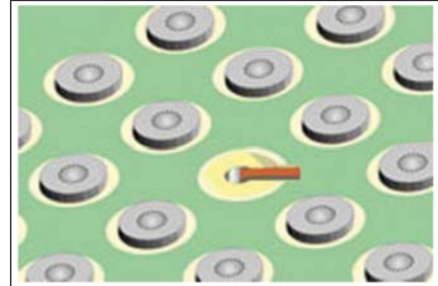


Figure 5 Conductor soldered and epoxied into the via.

Figure 6 Bond the new land using a bonding system.

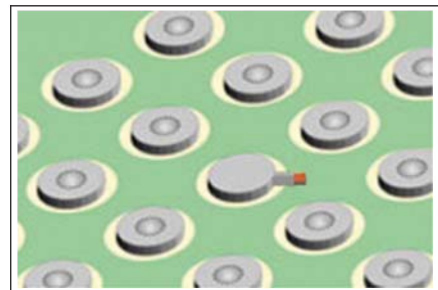


Figure 7 Completed repair

17. Select a bonding tip with a shape to match the shape of the new pad. See bonding tip chart in the replacement parts section of the manual provided with the repair system or repair kit (see Figure 6).

    NOTE

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

18. Use heat resistant polyimide tape to assist in aligning the replacement pad to the pad site.
19. Position the circuit board so that it is flat and stable. Gently place the hot bonding tip onto the heat resistant polyimide tape covering the new pad. Apply pressure as recommended in the manual of the repair system or repair kit for 5 seconds (or per the manufacturer's instructions) to tack the new pad in place. Carefully peel off the tape.

    CAUTION

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

20. Gently place the bonding tip directly onto the new pad. Apply pressure as recommended in the manual of the repair system or repair kit for an additional 30 seconds to fully bond the pad. The new pad is fully cured. Carefully clean the area and inspect the new pad for proper alignment.

    CAUTION

    Some components may be sensitive to high temperature.

21. Solder the tail of the replacement pad to the embedded conductor.
22. Carefully clean the area and inspect the new pad for proper alignment (see Figure 7).
23. Additional epoxy can be applied around the perimeter of the new pad 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. Pads are routinely subjected to one or more reflow cycles.
24. Apply surface coating to match prior coating as required.

EVALUATION

1. Visual examination
2. Measurement of new pad width and spacing.
3. Electrical continuity measurement.
4. Proper alignment of contact.
5. No film adhesive on contact or that interferes with solder requirements.
6. No electrical clearance violation.
7. No damage to contact surface plating.

NOTES