PREFACE

.
PCB rework is the process of making changes to or fixing flaws on a printed circuit board after it has already been produced or assembled. This may require anything from changing components to cleaning up excess solder to fixing broken traces or pads. Circuit board rework is a crucial step in electronics production because it allows manufacturers to address any issues with their final products, before they reach the end user.

In Electronics, defects leading to rework may include:

- Poor solder joints because of faulty assembly or thermal cycling.
- Solder bridges—unwanted drops of solder that connect points that should be isolated from each other.
- Faulty components.
- Engineering parts changes, upgrades, etc.
- Components broken due to natural wear, physical stress or excessive current.
- Components damaged due to liquid ingress, leading to corrosion, weak solder joints or physical damage.

In this book, Surface mount component rework and through hole component rework have been described in detail with illustration.

This E-book is dedicated to my wife ,Ajanta.

Acknowledgement

Author is thankful to all the agencies and clients, with whom he has consulted for preparing this manuscript.

While preparing a manuscript, the author has consulted various Standards, Articles and Company brochure, some of whose name are depicted in bibliography.

Last, but not the least, my sincere thanks goes to our Digital publication executive, Ms. Payal Pansuriya, who has done noticeable effort in making the digital version of this E-book.

Thane **Dr. R. Sengupta**
29-08-2023

CONTENT

Chapter: 1 Introduction and basic steps of Assembled PCB Repair & Rework

1.1 Introduction of PCBA repair and rework process

Rework (or re-work) is the term for the refinishing operation or repair of an electronic printed circuit board (PCB) assembly, usually involving desoldering and re-soldering of surface-mounted electronic components (SMD).

PCB rework is the process of making changes to or fixing flaws on a printed circuit board after it has already been produced or assembled. This may require anything from changing components to cleaning up excess solder to fixing broken traces or pads. Circuit board rework is a crucial step in electronics production because it allows manufacturers to address any issues with their final products, before they reach the end user.

In Electronics, defects leading to rework may include:

- Poor solder joints because of faulty assembly or thermal cycling.
- Solder bridges—unwanted drops of solder that connect points that should be isolated from each other.
- Faulty components.
- Engineering parts changes, upgrades, etc.
- Components broken due to natural wear, physical stress or excessive current.
- Components damaged due to liquid ingress, leading to corrosion, weak solder joints or physical damage.

1.2 Types of PCB Rework

Surface mount technology (SMT) rework and through hole technology (THT) rework are the two most common forms of the printed circuit board (PCB) rework. Surface mount technology, or SMT, refers to small, flat components that are soldered directly onto the surface of a printed circuit board.

Through-hole rework (THT rework) is the process of changing or repairing printed circuit boards (PCBs) that have through-hole components (components whose leads are inserted through holes in the PCB and soldered on the reverse side).

1.3 Why Printed Circuit Boards Fail?

Aside from physical damage, component failure is the most common reason why PCBs need repair and rework. Component failure may be the result of low-quality dislodging, or ageing of components. A bad PCB layout, however, is the most common cause. Power failure, overheated parts, and burned components may result from a poor design.

1.4 Benefits of Proper PCB Rework

It is crucial to do PCB rework correctly to guarantee the longevity and performance of electronic devices. When PCB rework is done correctly, it can have many positive results.

- **Increased product dependability**: In order to make products more reliable and less likely to break, businesses often repair PCBs that have flaws or faults. As a result, you may see an uptick in positive feedback from customers and a decrease in the price of warranty repairs.
- **Increased productivity**: If done correctly, PCB rework can improve production efficiency and reduce waste. Early detection and correction of mistakes and flaws, can help businesses save money and time that would otherwise be spent on repairs and rework.
- **Increased competitiveness**: Companies may stand out from the crowd and get an edge in the marketplace by creating superior, defect-free products.
- **Improved long-term viability**: The manufacturing process can be made more environmentally friendly by careful PCB rework. Companies can lessen their negative effect on the environment by resolving faults and flaws as soon as possible.

1.5 Issues in PCB Repair & Rework

1) Storage and Handling of Electronic Assemblies

In maintaining the consistency of a PCB, handling, packaging, and storage are all essential factors. Lack of sufficient storage, packaging, and handling can lead to contamination, physical damage, deterioration of solderability, and moisture absorption. The products used for transport and storage should be tested for packaging to ensure that they have no adverse effects on the board they intend to protect.

2) Recognize Your Product

Does your PCB contain lead or lead-free solder? Does halogen-free resin make use of your PCB? Does your PCB show corrosion, and if so, where are the common points of corrosion and the correction procedure? It is essential to know every aspect of your product to use the correct methods and reduce repair time. To ensure a smooth PCB rework and repair process, the materials from which it is manufactured, common problems with your particular board, and where the board was initially created are all critical pieces of information to know. In the rework and repair of PCB’s, there are several modules with the IPC certification as well as correlated training given on the subject that can help facilitate the process.

3) Divide & Conquer

Dividing the labour needed on each board is a wise choice, depending on the scope of rework and repair required, as it can save a company time and money. It has long been recognized that assembly lines’ production is a highly efficient way of assembling equipment. PCBs are no different from that. The separation of the steps needed to complete each board improves manufacturing productivity and decreases assembly error

1.6 Board Level Repair

Rework centres can perform a vast array of PCB repair services. Repair is a term, when referred to in the context of the IPC standards, is the repair of any physical damage done to the board, whether it be flex, rigid flex both for populated as well as unpopulated boards.

Our PCB repair services is performed by our top-notch group of PCB artisans who have been performing PCB repair services for many years. Their delicate hand can add new test pads that were forgotten in the board layout, repair mask that was damaged when the BGA was removed from the board or when a barrel was ripped out of a board while removing a component. The repairs are aesthetically pleasing and meet the form, fit and function of the original boards.

In the case of PCB ECO services production boards have a design or manufacturing problem that is manifested over many boards and there is no time to “rework the board in order to fix the mistake. In this case traces may be cut; new conductors or pads may be added such that the board can be assembled more rapidly and or more economically than a board can be designed.

1.7 PCB Circuit Board Repair

There are cases when you need a fast and/or extremely precise repair completed on an un-populated PCB.
Typical changes that BEST can perform on your board and are performed routinely include:

- Trace and conductor cuts
- Elimination of pads
- Addition of traces and pads
- Mechanical outline changes to a boar include the cutting of mounting holes
- Precise solder mask removal or additions
- Addition or removal of identification mark

1.8 Device Rework

We can not only perform rework at the board level, but also at the device level. Re-balling of area array devices whether they be plastic-, metal- or ceramic-bodied components can be competed in lot sizes from one piece to tens of thousands. Robotic hot solder dip services allow for a new alloy to be deposited on to the leads of parts. If you have a solderability problem on older components or need a new alloy on the component lead or even need to meet the newer requirements of the J-STD-001 in terms of the

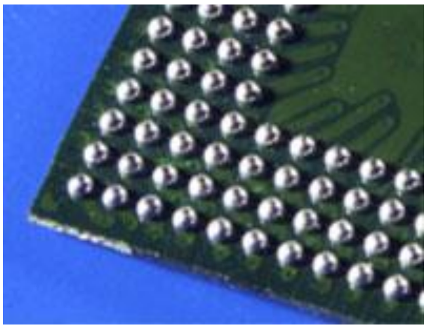
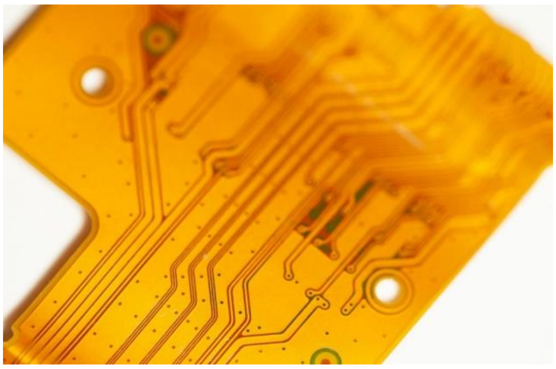


Fig. 1.1 BGA Reballing

1.9 Flexi & Rigid-Flexi Board Repair

Flex PCBs and rigid-flex PCBs offer many useful advantages over rigid boards, such as in devices like wearables and mil-aero. The flex section is built from polyimide-based materials and adhesives, as well as copper foils, which build up a layer stack that can support reflow soldering. Assembly on flex materials generally follows similar procedures as standard rigid boards with two important exceptions: the need for fixation and pre-baking.



Flex PCB materials can absorb moisture that interferes with high-volume assembly, specifically during reflow soldering. There are also the chances of deformation during the reflow cycle, where the board is slowly heated up to a required assembly temperature.

Fig. 1.2: Flex Board

Some simple pre-assembly steps can help ensure maximum quality of the assembled flex PCBA, and the same steps could be necessary during rework.

Chapter 2: Handling PCBs during repairing

2.1 Introduction:

You need to take serious care when you are handling PCBs. This is to ensure that more damage doesn't come during the repair. Below are some of the issues you need to worry about.

2.2 Discharge and unplug your electronics

In order to avoid your electronics from experiencing short circuits and possibly causing electrocution, ensure that you unplug the faulty device and discharge it completely. Power supplies and capacitors can hold charges. This is why you have to pay much attention to ensure that they are totally discharged. Also, if there are supplemental batteries, make sure you disengage them too.

2.3 Put on safety glasses

Safety glasses are very important because the solder could splatter and leads could fly during trimming. Therefore, make sure you put on safety glasses. This offers some protection for your eyes during PCB repairs.

2.4 Prevent any static discharge during PCB Repair

A typical technician working on a PCB can generate thousands of volts. SMOS logic devices can be damaged with 250-3000 volts, EPROM devices down to 100 volts, and microprocessor chips as low as 10 volts. Damage can shut down functions or create intermittent problems, and it can either be catastrophic (immediate) or latent (as in later "field failure"). *(Source: Phil Storrs PC Hardware Book)*

Electrons are constantly being exchanged as objects interact with each other. Problems occur when the materials involved don't allow the electrons to flow easily and reach equilibrium. Called insulators, they tend to collect electrons, which create negatively charged hot-spots on their surfaces.

When these charged hot-spots come in close proximity (not necessarily even direct contact) with another object, a sudden exchange or "discharge" can occur, where electrons are suddenly exchanged to create equilibrium. That event is called "electrostatic discharge" or "ESD".

Electrostatic discharge happens all the time, especially in dry climates and in the winter season, when there isn’t moisture in the air to help bleed off excess electrons (water is conductive, so allows the electrons to flow more readily). Normally it is nothing more than an irritation, but with sensitive electronics, it can lead to PCB failures.

Wearing a grounded wrist strap and working on a grounded ESD-safe mat is the best way to prevent ESD. A wrist strap has a conductive surface that touches the skin, so can bleed off excess electrons. An ESD-safe mat works by a similar principal, bleeding off any charge from its surface. If working on a PC, you can clip the wrist strap directly to an unpainted area of the case.

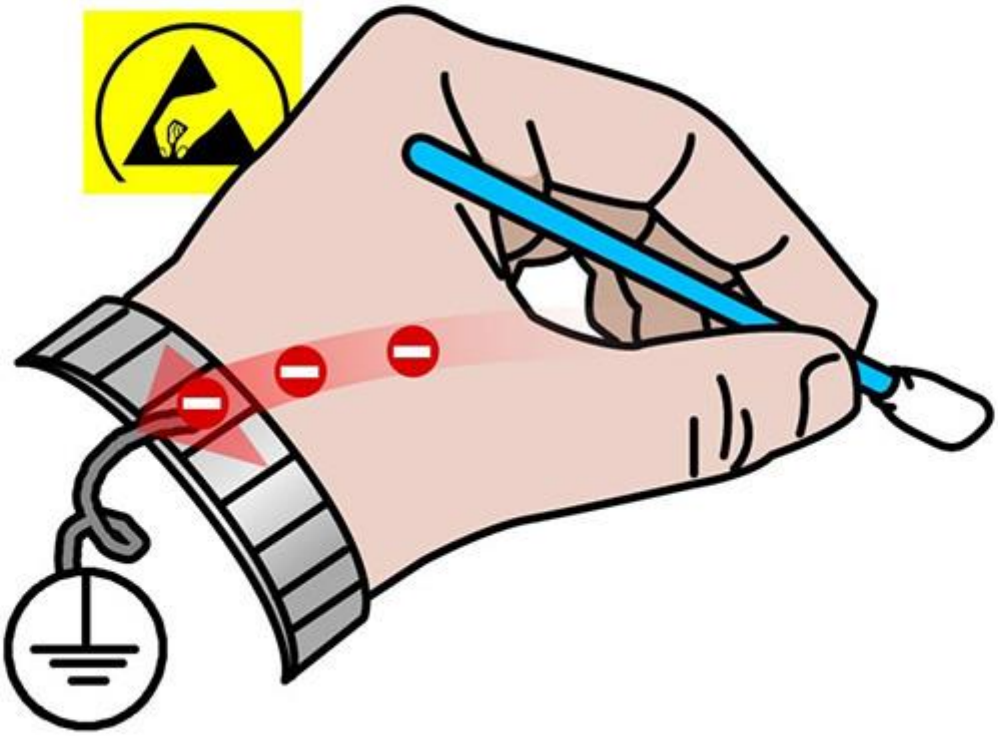


Fig.2.1 Grounded wrist strap bleeds off excess electrons to prevent damaging static discharge

2.5 Wash your hand and handle just the edges of the Printed Circuit Board to Reduce possible contamination

Contamination from the hands can lead to solderability issues. It can also result in board failure as a result of dendritic growth or corrosion. Silicon from your usual hand lotion or you picked it up mistakenly from somewhere could form a barrier on the contact areas, which could prevent this solder from creating a strong metallurgic bond.

Furthermore, this solder might stick temporarily, but at exposure to physical or thermal stress, it may pop off. Contamination, such as salt on fries you consumed with lunch is another cause of issues that come in during the repair. Salt is ionic and when it combines with moisture present in the air could cause corrosion. These ionic particles could also join and form dendrites leading to leakage and short circuiting.

Also, wash your hands making use of hand lotions that are electronic safe only. This is a way of preventing this board failure type. Furthermore, holding only the edges of the board without contacts could help in preventing the deposition of contaminants on the critical

Chapter 3.Bare PCB Modification & Repair

3.1 REPAIR

This can be defined as a minor engineering change, made in an assembly to effectively result in an acceptable finished product. Engineering changes made, are to be documented and considered as standard repairing procedure. A fully assembled PCB is a product of the most sophisticated technology often worth a lot of money. Occasionally failures occur in use which require repair or modifications. A PCB may fail either due to electrical or mechanical reasons.

3.2 Precautions to be Taken to Avoid Mechanical and Electrical' Damage during Repairing

Careless handling of PCBs and hand tools can lead to further damage during the repair process. The following precautions should be taken:

a) Remove the board from the equipment during the repair process.
b) Great care should be taken to avoid mechanical damage to the board when removing the component which has been stuck to the board. Cleaning solvents must be used carefully to avoid damage to the components on the board.

c) Mechanical damage to the copper loll is most likely to occur when stress Is applied to the component leads in a direction that would force the copper away from the board. Avoid applying such a force.

d) During the soldering process, when the joint is at the soldering temperature, the strength of the bond between the track or land and the board is less. So, the soldering process must be correctly carried out and the joint should be allowed to cool and regain the strength of the bond.

e) Excessive heating of the board should be avoided and care must be taken to avoid unnecessary damage to any protective coating.

f) In multilayer boards, care must be taken during repairs to avoid damage to
any of the PTH, as these interconnect the internal conductors.

g) While replacing the components, it must be checked to see whether special handling techniques are required. If so, those instructions must be followed strictly. The components which are sensitive to static electricity, should be handled carefully so that they are not damaged.

h) The components can easily be damaged, if leads are bent very close to the body.

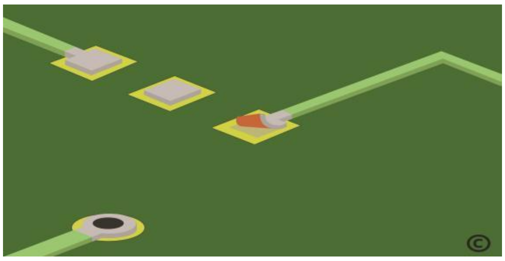


Fig. 3.1

1.The board area requiring repair is first cleaned in and around the pad/trace to be repaired. See **Fig.**3.1

Fig. 3.2

2. This epoxy (**Fig.**3.2), is designed to withstand the chemicals, heat and can adhere to materials common in PCB assembly. It has been used since 1998 and can be used in a variety of applications including but not limited to pad and trace repair, solder mask repair, baseboard material repair as well as numerous others. The mix ratio is already taken care of for you in the pre-mixed bags and therefore is easy to remember and easy to mix. It is a 3:1 ratio mix.

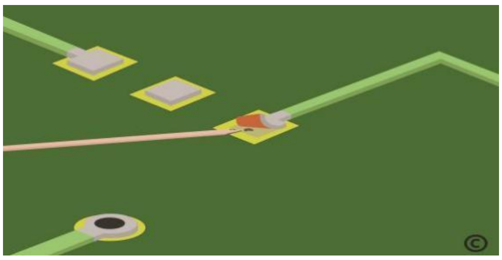


Fig. 3.3

3. The board area requiring repair is first cleaned in and around the pad/trace to be repaired.

4. Carefully apply a small amount of epoxy **(Fig.3.3)** under the entire length of the lifted circuit. The tip of an orange wood stick may be used to apply the epoxy.

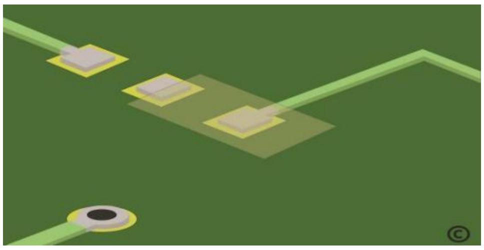


Fig. 3.4

5. Press the lifted pad/trace down into the epoxy and into contact with the board.

6. Apply additional epoxy to the surface of the lifted circuit and to all sides as needed for additional strength. Tape the repaired pad/trace into position while curing **(Fig.3.4)**

7. Cure the BEST epoxy per the instructions.
8. Re-apply conformal coating to match prior coating if it required.
9. Perform a visual examination per IPC A-610 acceptability criteria or customer requirements.
10. Perform a tape test per IPC-TM-650.10. Perform continuity and other electrical tests as applicable.

3.3 Gold Finger Repair

Gold fingers are prone to damage as they are on the periphery of the circuit board and they are also prone to solder splash during the wave soldering process. When edge contacts are contaminated with solder, get scratched, torn or damaged, technicians can bring them back into specification. These contacts can either be replaced or repaired. The repair methods will restore your edge contacts to a high level of performance and reliability.

When the gold fingers get scratched, contaminated with solder or if the plating is worn, they can be re-plated. The contaminated contacts are stripped down to the bare base metal and replaced to meet the thickness specified. Our electroplating process is reliable and will restore damaged contacts to "like new" condition with full functional conductivity and durability.

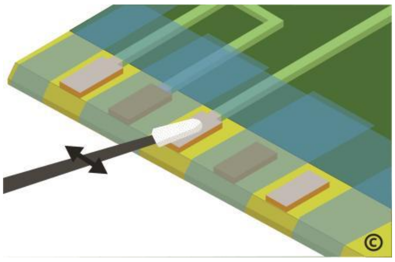


Fig. 3.5

Generically for a replating process the first step is to clean and inspect the area to replated. At this point BEST will determine whether or not a replating process will be appropriate based on the extent of the damage or splash on the gold-plated surface **(Fig. 3.5)**

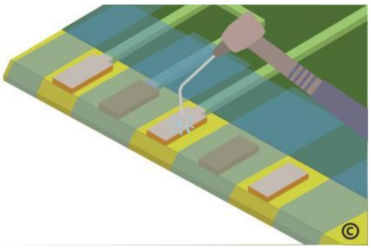


Fig. 3.6

Areas in and around those with solder splash are taped off and the solder is wicked off and then a solution helps to strip off the remnant solder from the surfaces. If a replacement pad is required then a new gold-plated pad is installed instead of spending time re-plating the existing pad (**Fig. 3.6)**

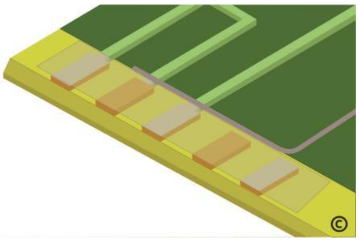


Fig. 3.7

Areas in and around those with solder splash are taped off and the solder is wicked off and then a solution helps to strip off the remnant solder from the surfaces. If a replacement pad is required then a new gold-plated pad is installed instead of spending time re-plating the existing pad (**Fig.3.7)**

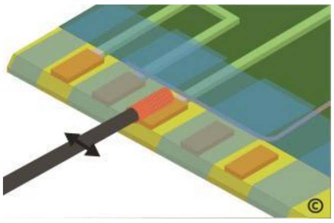


Fig. 3.8

A bus wire is then attached in close proximity to the re-plating area. Using a small power supply with swabs connected to the electrodes the plating process is undergone using a nickel (if required) then gold plating solution. After plating the area is inspected and tested (**Fig. 3.8)**

3.4 SOLDER MASK REPAIR PROCEDURE

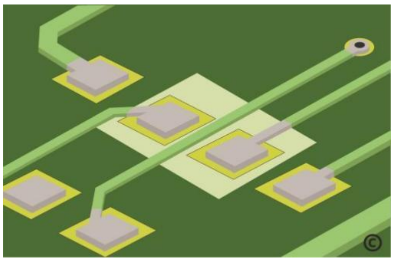


Fig. 3.9

1. Make sure that the area to have the solder mask repaired is cleaned and inspected (**Fig. 3.9)**

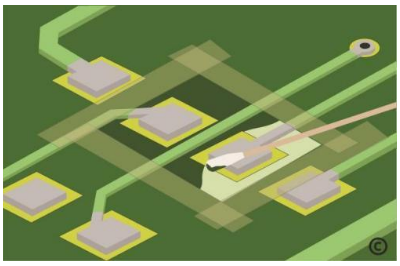


Fig. 3.10

2. If required, apply high temperature tape in order to outline the area where the replacement solder mask will be applied **(Fig. 3.10)**

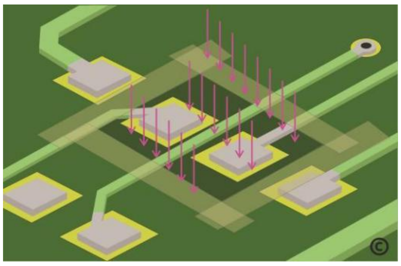


Fig. 3.11

3. using the replacement UV-curable solder mask material apply this to the areas requiring repair. If desired, add colour agent to the mixed epoxy to match the circuit board colour **(Fig. 3.11)**

4. Cure the replacement solder mask by setting the PCB underneath the UV curing lamp as per the manufacturer's recommendations. Watch out for heat sensitive components.

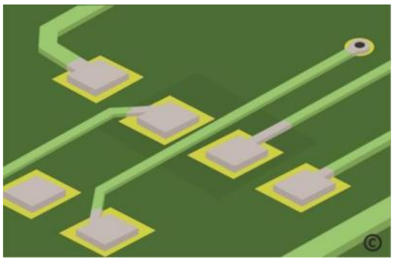


Fig. 3.12

5. Perform a visual inspection for colour match and adhesion per the IPC TM-650 specifications **(Fig.3.12)**
6. Inspect per the latest IPC-A-610 requirements.

3.5 REPAIR OF A CIRCUIT TRACK

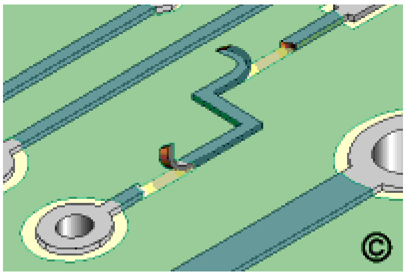


Fig. 3.13

1. Evaluate damage and measure conductor width (Fig. 3.13).

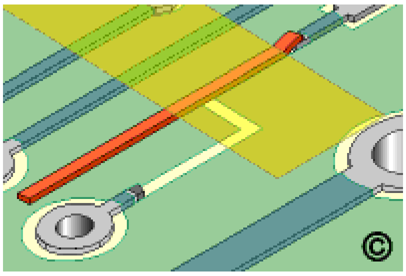


Fig. 3.14

2. Remove damage, place new track and solder (**Fig. 3.14)**

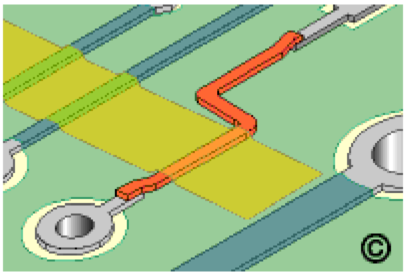


Fig. 3.15

3. Form circuit track to match conductor path (**Fig. 3.15)**

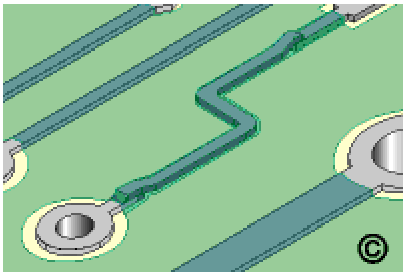


Fig. 3.16

4. Make final bends and solder
5. Bond in place and overcoat with epoxy (**Fig. 3.16)**

Chapter-4: Desoldering Thru' Hole Components

4.1 INTRODUCTION

This chapter describes the various disordering techniques of through hole and surface mount devices. The precautions to be taken during the disordering process have been described along with the method for the replacement of surface mount devices. The tools and the equipment essential for flawless disordering of components have also been described in this chapter

4.2 REWORK

This is the process of redoing the job to the original specification without any basic change. With every rework, the solder metal alloy zone widens and the basic strength of the joint decreases. Due to repeated soldering and disordering operations, conductor may get peeled off. A maximum number of three reworks is permitted on any one solder joint.

4.3 REPAIR

This can be defined as a minor engineering change, made in an assembly to effectively result in an acceptable finished product. Engineering changes made, are to be documented and considered as standard repairing procedure. A fully assembled PCB is a product of the most sophisticated technology often worth a lot of money. Occasionally failures occur in use which requires repair or modifications. A PCB may fail either due to electrical or mechanical reasons.

4.4 Precautions to be Taken to Avoid Mechanical and Electrical' Damage during Repairing

Careless handling of PCBs and hand tools can lead to further damage during the repair process. The following precautions should be taken:

- Remove the board from the equipment during the repair process.
- Great care should be taken to avoid mechanical damage to the board when removing the component which has been stuck to the board. Cleaning solvents must be used carefully to avoid damage to the components on the board.
- Mechanical damage to the copper loll is most likely to occur when stress is applied to the component leads in a direction that would force the copper away from the board. Avoid applying such a force.
- During the soldering process, when the joint is at the soldering temperature, the strength of the bond between the track or land and the board is less. So, the soldering process must be correctly carried out and the joint should be allowed to cool and regain the strength of the bond.
- Excessive heating of the board should be avoided and care must be

Taken to avoid unnecessary damage to any protective coating.

- In multilayer boards, care must be taken during repairs to avoid damage to any of the PTH, as these interconnect the internal conductors.
- While replacing the components, it must be checked to see whether special handling techniques are required. If so, those instructions must be followed strictly. The components which are sensitive to static electricity, should be handled carefully so that they are not damaged.

- The components can easily be damaged, if leads are bent very close to the body.
-

4.5 Disordering of Damaged Components

The process which is used for removing solder from solder joints is called disordering. Extreme care should be taken to avoid damage to the circuitry, base laminate or other components which could be caused by excessive heat application, improper use of tools, and use of incorrect tools or rough handling of the boards, while performing disordering.

4.6 Different disordering methods

The different methods for disordering are given in detail below:

(a) Disordering/Removal of axial lead components by cutting the component body (see Figure 4.1): Stepwise procedure

- Cut the leads using a small side cutter.
- Remove the component body.
- Straighten the remaining part of the leads gently, do not apply too

Much force as the pad may get damaged.

- Clean the bit of the soldering iron every time to ease heat transfer.
- Keep the board almost vertical.
- Apply a temperature-controlled soldering iron to the solder joint to melt the solder, press the lead from the component side of the board with the help of a rod.

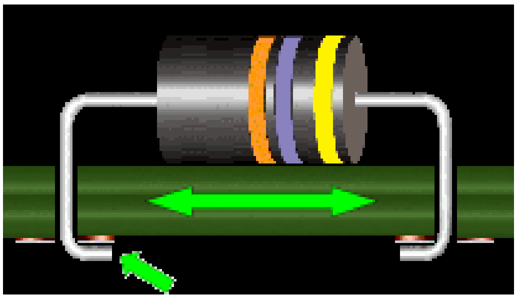


Figure. 4.1 Removal of axial lead components

by lead cutting

- Remove the lead from the hole with the help of tweezers.
- Remove the remaining solder with the help of a temperature Controlled solder removal tool.
- Clean the surface with isopropyl alcohol.
- Check for any pad/track or conductor lifted from the PCB.
- Replace anew component. Care should be taken to ensure that excessive heat is not applied to a replacement component that may be heat sensitive.
- Repair original coating, if removed in above process. If the original coating is unobtainable, coat the affected area with polyurethane varnish.

(b) Disordering/Removal of axial lead components

This method should only be used when it is essential to remove the component without damaging it.

Figure: 4.2 vacuum type de-soldering station

- Clean the solder joint with isopropyl alcohol.
- Clean the bit every time and use a small amount of solder to ease heat transfer.
- Hold the board in a vice or by any other mechanical means that will give free access to either side of the faulty component.
- Use a temperature-controlled soldering iron at 250°C to melt the solder around the lead. Straighten the lead with a dental probe or scalpel while the solder is molten.
- Repeat the previous step for all other leads. Remove the solder from the component leads and holes using a temperature controlled desoldering tool (Figure 7.2).
- Check that all the leads are free.
- Remove the component from the board.
- Clean the surface with isopropyl alcohol.
- Insert the replacement component and resolder, ensuring that the

terminal is heated only for the minimum time required to create a successful joint.

- Remove the flux residues with isopropyl alcohol.
- Repair the original coating if removed in the above process. If the original coating is unobtainable, coat the affected area with polyurethane varnish.

4.7 Disordering/Removal of component where the leads are inaccessible from the component side (e.g., a transistor)

- This method (Figure 4.3) is applicable to all components whose leads are inaccessible (applicable to plated through hole board also). The disordering procedure is the same as described in (b) above.

Figure. 4.3: Desoldering of components having Inaccessible leads

(d) Disordering/Removal of through hole mounted Dual-in-line ICs using the vacuum disordering tool

i) Clan the solder joints with isopropyl alcohol.

ii) Clean the disordering bit with a damp sponge and tin it to ease heat transfer.

i) Using the disordering tool, melt the solder at the joint. Move The disordering tool around the lead in all directions to remove the solder from PTH.

i) Repeat the procedure for all the remaining leads until all the leads are free from the board.

ii) Using a scalpel or any other suitable tool carefully straightens any bent leads.
i) Remove the device carefully from the board by hand or by using any approved removal tool.
ii) Examine the board.
iii) Insert the replacement package and resolder. Care should be taken that the terminal is heated only for the minimum time required to affect a successful joint.

(e) Disordering/Removal of through mounted dual—In—line ICs using the disordering block

Fig.4.4 Desoldering Block for IC Removal

When removing a dual—in—line package by this method, it is essential to melt the solder retaining both rows of terminals simultaneously, and remove the package the very moment the solder melts. This is best done with the special tool (Fig.4.4) ,as described below:

i) Temperature controlled multiple bit soldering irons.
ii) Spring loaded removal tool.

If any one of the leads is found bent, it should be straightened before applying the following procedure.

i) Fit the spring-loaded removal tool over the dual-in-line package.
ii) Clean the multiple bits with a damp sponge and tin it.
iii) Apply the multiple bits simultaneously to both rows of terminals, the instant the solder retaining every terminal is sufficiently melted, the package is automatically withdrawn from the board by the removal tool.
iv) Remove the excess solder from the hole using a temperature

Controlled solder removal iron.

Check the track pads and conductors of the board.

Insert the replacement component and resolder. Care should be

Taken that the terminal is heated only for the minimum time required to affect a successful joint.

vii) Remove the flux residue with isopropyl alcohol.
viii) Repair the original coating, if removed. If the original coating Is unobtainable, coat the affected area with polyurethane varnish

ix) Inspect to ensure that repair has been carried out neatly.

(f) Disordering/Removal of through hole mounted dual—in--line ICs where special tools are not available

i) Cut gently through each of the leads on both sides of the component, Flush with the board, using side cutters.
ii) Remove the component body.
iii) Using the temperature-controlled soldering iron, melt the solder around the lead and suck -it out using either a hand-held disordering pump (Figure 4.5) or disordering wick braid (Figure 4.6).

Fig. 4.5:

Manual de-soldering pump

Fig. 4.6:

Desoldering wick/braid

Remove the lead from the hole using tweezers or snipe—nosed pliers.

iv) Repeat steps (iii) and (iv) until all the leads are removed from the Board.

vi) Remove the excess solder from the hole using a temperature-controlled solder removal tool.
vii) Inspect the board.
viii) Insert the replacement component and resolder. Care should be taken that each terminal is heated only for the minimum time

required to affect a successful joint.

ix) Remove flux residues with isopropyl alcohol.

x) Repair the original coating if damaged in the above process. If the original coating is unobtainable, coat the affected area polyurethane varnish.

xi) Inspect to ensure that the repair has been carried out properly.

CHAPTER- 5: SMD SOLDERING & DESOLDERING

5.1 REWORK OF SMDs

This is the process of redoing the job to the original specification, without any basic change. This chapter mainly describe various Desoldering Methods of SMDs.

Please note that with every rework, the solder metal alloy zone widens and the basic strength of the joint decreases. Due to repeated soldering and desoldering operations, conductor may get peeled off. A maximum number of three reworks is permitted on any one solder joint.

5.2 DESOLDERING OF SMDs

INTRODUCTION

When it comes to repairing electronics, desoldering is an important skill to learn. Whether you want to replace defective or incorrectly placed parts, fix bad solder joints, troubleshoot an electric circuit or salvage electronic components, desoldering can help you do so.

Desoldering is the process of removing solder and components from a circuit board. It involves getting rid of one or more solder joints to free up the component before removing it. There are various methods through which this can be done. It is the opposite process of soldering, but can be done with mostly the same tools.

Desoldering generally consists of multiple steps. The first one is to heat up the solder of one or more solder connections. This is usually done with a soldering iron or a hot air source. Sometimes you remove the solder. After this, manual

Take the component there are several reasons why you would want to desolder. For example, to repair solder joints or replace broken components on a circuit board. Another reason would be to troubleshoot an electrical circuit that is not

Working correctly. You can also desolder components to **salvage them** for use in other projects.

5.3 SMD DESOLDERING TIPS

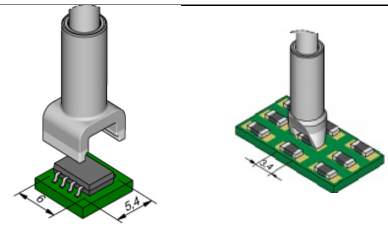
- **Always limit exposure time of components to heat as much as possible.** High heat for a prolonged period of time damages components and boards. Don't blindly turn the temperature setting of your soldering iron all the way up to make desoldering easier.

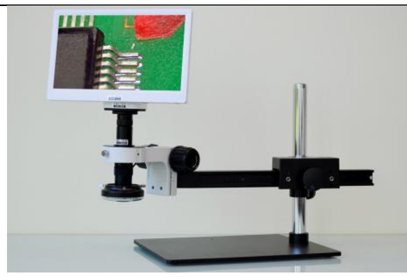
- **If you are desoldering something that has been soldered with lead-free solder, add some leaded solder to the solder joint.** Leaded solder has a lower melting point than lead-free solder. Mixing it in lowers the overall melting temperature and makes desoldering easier. You can also use Chip Quick for this (you can find more info on how to use Chip Quick further down the page).
- **If possible, use a temperature adjustable soldering iron for desoldering.** A temperature adjustable soldering iron allows you to set the right temperature to prevent overheating components and solder pads.
- When using a simple, non-adjustable iron, aim for one that uses 15 to 30 watts.

Table: 5.1 SMD De-Soldering KIT

S. No	Tools & Materials	Description	Picture
1.	Desoldering Pump	Manual desoldering pump is used to remove molten solder from the PCB pad, after the removal of Component,	
2.	Desoldering Braid	A **Desoldering Braid or solder wick** is a pre-fluxed finely braided copper wire that is used to remove excess solder during the hand soldering, rework and repair process.	

3.	Soldering Iron	Soldering iron and properly sized soldering iron tips are used to soldering/resoldering of components	
4.	PCB Pad/Track cleaner	General purpose cleaner for removing contamination.	
5.	Solder lint free, antistatic Wipes	These anti-static wipes are suitable for use in (electronics) laboratories or even in the cleanroom (ISO class 4). They can be used both dry and wet (with e.g., IPA).	
6.	Tweezer	Tweezer is generally used for Surface Mount Component Placement and Removal from PCB pad.	

7.	Solder fume Extractor	Soldering fumes (also known as soldering smoke) are hazardous fumes that are generated from the process of melting down the flux past its boiling point. When the flux is melted down, its state goes from solid - to liquid - to gas (the “Solder Fume”).	
8.	SMD Hot Tweezers	Hot tweezers designed exclusively for the easy removal of SMD chips and flat packages of up to 25mm The iron holder clip allows accurate positioning of the tip.	
9.	Chip Quick	Chip Quick®, Inc. is the manufacturer of the new patented Chip Quick® SMD removal kit. This innovative method of removing SMD's (surface mounted devices) at a safe low temperature has revolutionized the printed circuit board rework industry.	
10.	"Slot tips"	Surface Mount Tip Cartridges are designed by M/S Oki, for its Metcalf products, to remove SMD components. The slots are sized to match the lead width of the component.	
11.	Hot Air SMD Rework Station	They provide a controllable flow of hot air that is used to rework circuits and boards. They provide better temperature precision than	

		the soldering irons and are faster and easier to use.	
12.	Inspection Microscope	Precision microscope with stand and lighting for work and inspection	

5.4 COMMON METHODS OF DESOLDERING OF SMDs

5.5 De soldering of SMD with a Soldering Iron

Introduction:

There are many different ways to go about removing an SMD chip from a board. To go about this safely you could use a desoldering gun, soldering iron and solder wick or other methods. One of the safest ways to do so is to use a simple soldering iron. How do we do this? First you need a soldering iron, flux, and low melt solder. This process is much less complicated than what it is going to sound like so bear with me.

Tools used:

1. Soldering Iron
2. Sn/Pb Solder
3. RMA Flux
4. ESD tweezers

Decide on what you want to use for flux and for low melt solder. For example, a small project requires picking a kit that has both included. These kits are called Fast Chip and can come either with leaded or lead-free solder depending on what you need. You want these for small uses because they come with a limited amount of flux and solder. Larger packages of removal alloy and flux are available if you are doing this on a regular basis.

Second, the soldering iron. Now if you have a soldering iron that you are comfortable with you can use that. Being new you do not need an advanced soldering iron but you do want one that will last you for this project. Also, it can help for all the future ones. That being said, find a simple iron that will allow you to perform these tasks without issue. In addition, it would be a benefit to pick up a pair of **ESD tweezers**.

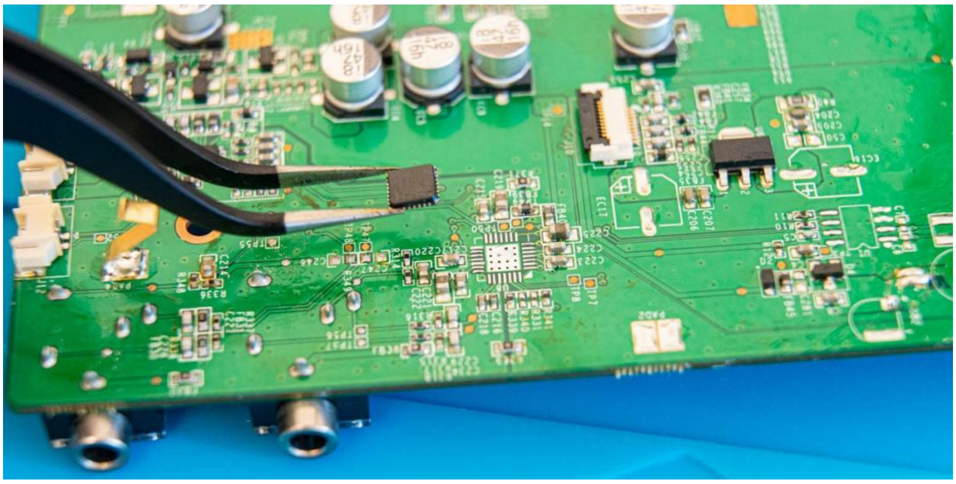


Figure. 5.1: Lifting SMD using ESD tweezers

5.6 SMD De-Soldering steps

1. Heat up the soldering iron.
2. Apply some tacky flux to both sides of the SMD.
3. Using the soldering iron and low melt solder, apply the solder to the joints on the SMD legs in large amounts.
4. Then move the soldering iron across all the leads of the SMD unit all the solder is molten.
5. Using a pair of tweezers and the soldering iron, remove the SMD chip

5.7 Desoldering with Solder Wick

Solder wick, desoldering wick, or just "wick" are all names for a copper braid that is used to absorb solder. It is generally coated with flux so that when heated solder is melted, drawn up, and retained using a combination of wetting and capillary action. Solder wick allows you to remove solder in isolated areas without thermally stressing the overall board or nearby components. Solder wick can only remove exposed solder, so components covering contact areas, like ball grid arrays (BGAs), have to be removed before the residual solder can be removed.

5.8 What is solder wick?

Solder wick, de-soldering braid, or just “wick” are all names for a copper braid that is used to absorb solder. It is generally coated with flux so that when heated the solder is melted, drawn up, and retained using a combination of wetting and capillary action. Solder wick allows you to remove solder in isolated areas without thermally stressing the overall board or nearby components. Solder wick can only remove exposed solder, so components covering contact areas, like ball grid arrays (BGAs), have to be removed before the residual solder can be removed.

Solder wick comes in a wide variety of widths to efficiently remove solder from different types of contact areas. Solder wick generally comes coated with flux that can be matched to your original soldering process: rosin, no-clean, or un-fluxed to add your own flux as you use it.

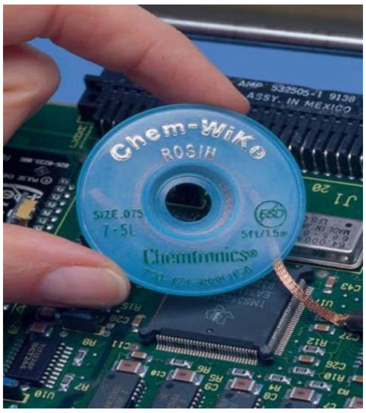


Figure. 5.2: *Solder* Wick

Figure. 5.3: Close up of copper solder wick

5.9 How to use solder wick?

1. Place the braid over unwanted solder, preferably on the greatest solder build up so that it maximizes the contact of the braid to the surface area of the solder.

2. Next place your iron tip over the wick at 45 degrees and allow heat to transfer to the pad. Molten solder will absorb into the braid.
3. Move the solder tip and braid as needed to remove all of the solder at one time. Be careful not to drag the braid over the pads as it could scratch. Once the braid is full of solder, you must trim the spent portion and move to fresh braid in order to pull more solder. Remove the iron and braid simultaneously to avoid soldering the wire to the board.

5.10 Common sizes of solder wick:

Wick that is too thin won't remove enough solder and Wick that is too wide takes longer to heat and may interfere with other components on the circuit board.

Choose a desoldering wick width that closely matches the size of the contact area. This will ensure that you get proper heat conduction and that you don’t desolder unwanted areas. Widths of desoldering wire are designated by the numbers 1 through 6 or by colour codes, and these are standard across the industry.

- #1/white braid is the smallest (under 1mm wide) and is mainly used for SMD’s and microcircuits.
- Most will find #2/yellow, #3/green, and #4/blue to be the most common desolder wire.
- #5/brown is ideal for removing large blobs of solder and #6/red is best for desoldering BGA pads or terminals.

Figure. 5.4 ESD-safe bobbin for 5' and 10' solder wick

5.11 Common types of flux coating on solder wick:

- Rosin – Rosin-fluxed desoldering wick has the fastest wicking action but does leave behind residues that need to be thoroughly cleaned.
- No-Clean – No-clean fluxed desoldering wick is ideal when cleaning isn’t practical or possible. After desoldering, the only thing that remains is a clear, non-ionic residue.
- Un-fluxed – Un- fluxed wick will not remove solder unless flux is added. Different types of fluxes are available in pen packaging, which is ideal for fluxing braid.
- Solder wick can also be packaged in a variety of lengths. 5’ and 10’ lengths are handy for use at a work station. Static dissipative spools, also called “bobbins”, are available to prevent damaging components sensitive to ESD.

- Other specialized braid is also available, like Lead Free Solder-Wick used to minimize thermal shock for high-heat lead-free applications.

5.12 De-soldering of SMDs with Surface Mount Tip Cartridges

Removal of SMD Chip Components

OK International currently offers nearly 40 Surface Mount Tip Cartridges for its Metcalf products, to remove small, discrete components. Oki calls these tips their "slot tips," and in most cases, the slot is sized to match the lead width of the component.

Choose the correct tip temperature for the removal task. When choosing the temperature, it is always best to choose the lowest temperature possible to accomplish the job.

5.13 Steps for removal of Chip Components:

1. Bring the tip down directly over the component so the inside edges make direct contact with the leads of the part.

2. Allow the tip to reflow the solder completely on both sides of the component. This requires a minimal degree of skill and judgment.

When you are certain it is completely reflowed, slightly twist the tip in a circular motion and lift simultaneously. This action is necessary to break the surface tension bond of the solder that has been holding the component to the lands.

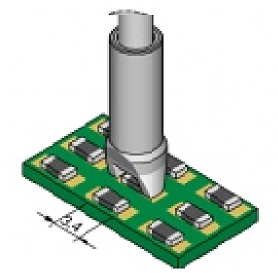


Figure. 5.5: Removal of Chip Components

Removal of SMDs having multi-leaded, two-sided components (SOICS, SOJS, and SOPS packages)

Bring the tip down directly over edges make direct contact with the leads of the part.

Allow the tip to reflow the solder completely on both sides of the component. This requires a minimal degree of skill and judgment. Once reflow process is completed take out the components.

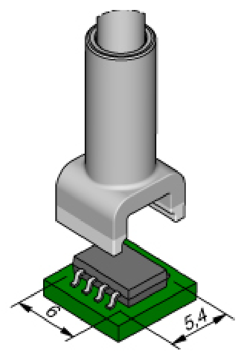


Figure. 5.6: Removal of Multi-leaded, two sided SMDs Components

5.14 Steps for removal of multi-leaded, two sided SMDs:

A) Identify the component. If you know the component identification, you can find the proper SMTC tip in Oki's Metcalf catalo.

B) Choose the proper tip temperature. Again, Oki makes these removal tips in several temperature series, but we suggest using a 600 Series for most applications. Exceptions would be heavy ground plane boards or components that are much larger and denser.

C) After plugging in the tip and allowing it to heat, fully tin the inside edges of the tip.

D) Bring the tip straight down on top of the leads of the component. Make sure that you have contact with all of the leads.

E) Leave the tip on the leads without rocking or twisting the tip on the component. An approximate time would be one second per two leads. Patience is the key to not lifting a pad.

F) Once you are certain all leads are reflowed, use a slight twisting and wiping motion to remove the component off the board. The surface tension of the tip's tinned edges will hold the component in the tip.

G) Wipe the component on the wet sponge to dislodge it from the tip.

5.15 Removal of Standard Pitch Gull-Wing SMDs

Steps:

1. Align the component to the pads on the board.
2. Align the leads with the pads, making sure the number one lead is lined up with the number one pad. Directly centre the leads onto the lands.
3. Holding the component steady with one hand, flux the opposing corner leads of the part.
4. Place enough solder onto the face of the tip so that it covers about one third of the tip. Keep the solder down towards the "toe" of the tip.
5. Bring the tip down and "paint" the leads where you applied the flux. The object here is to tack the part down, not make good solder joints.
6. Wipe the excess solder from the tip.
7. Make sure that the face of the hoof is shiny and wettable. You will not get good results if the tip is oxidized. If solder balls up on the face, you need to tin the tip. Once the tip is tinned and shiny, the solder should flow on the tip.
8. Apply enough solder to cover approximately one-half of the

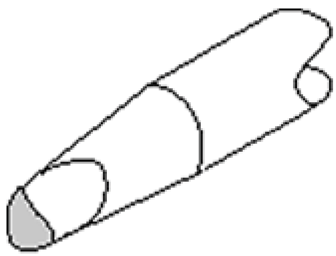


Figure. 5.7: Chisel Tip

tip. The amount of solder will vary with the number of leads you have. For fewer leads, apply less solder.

9. Flux one entire row of leads at a time. Work with the row of leads going from left to right in front of you. Work from left to right if you are right-handed, and vice-versa if you are left-handed.
10. Hold the tip so the toe of the hoof runs parallel along the row of leads.
11. Bring the tip in on a flat landing onto the first lead. Immediately begin running the tip down the row of leads drawing a straight line.
12. When you reach the end, wipe the tip down the lead towards you.
13. Repeat these steps for each row of leads.

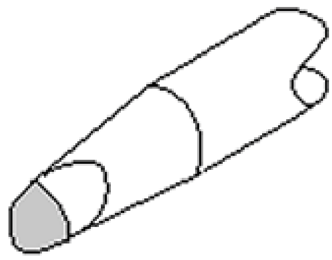


Figure. 5.8: ½ of the Tip

5.15.1 Additional information:

The angle of the tip should be parallel, but can be up to 30 degrees for good results. If the toe of the hoof is the front, then the right side should make contact right at the bend of the leg of the lead. It should be held in that position for the entire process. If you are left-handed, it will be just the opposite. The speed is more crucial than the angle. The speed should be steady and slow enough to deposit the solder from the tip onto each of the leads as you draw it across. An approximate time is one second per lead. As you get more comfortable with the procedure, the speed becomes less crucial.

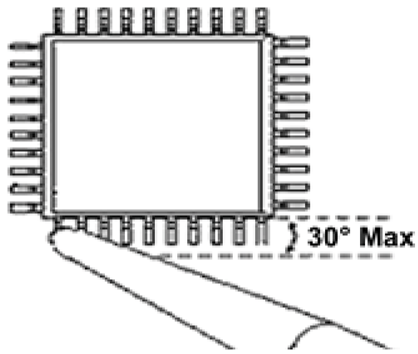


Figure. 5.9: Angle of the tip should be max. 30 degrees

5.16 Removal of Four Sided SMDs

1. Removing larger component packages will generally require more heat and the use of flux for the best, fastest results. Large four-sided components will require Dual heater tips, which use two hand pieces and two power supplies. This is to deliver 80 watts of

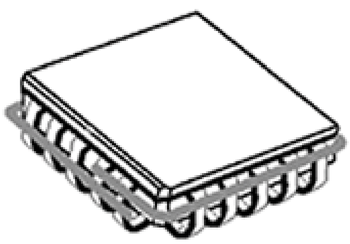


Figure.5.10: Wire Wrap

power. When using a dual heater tip,
Use a dual handle support (STSS-DHS) over
The hand pieces to keep the shaft of the tip
Cartridge from bending.

a) Allow the tip to heat fully. This is important, because if it is not up to full temperature, your removal time will be increased.
b) Tin all inside edges of the tip. Just like your blade or hoof tip, the removal tips must remain shiny and wettable.
c) Flux all sides of the component.
d) Take some wire solder (.025 or .031 works best) and wrap it around the leads of the component. On J-leaded components this step is easy. The wire will wrap easily around the leads and stay in place.
e) Bring the tip straight down on top of the art, making full contact on all four sides of the leads.

   Hold the tip in place for about 10-15 seconds without moving back and forth or rocking. Once you practice this step a little, you will get a feel for the reflow and will actually find that it becomes a little faster.

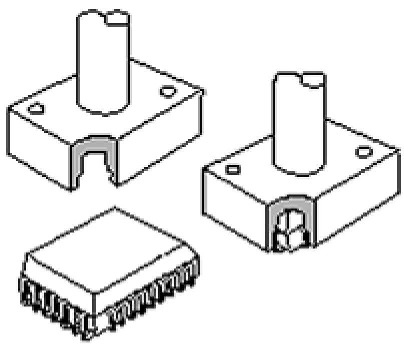


Figure. 5.11: Removing a PLCC

When you are certain you have full reflow, twist the handle between your thumb and forefinger just slightly. If you do have complete reflow you will feel the component move. If you do not, the cartridge will twist in the handle. This is a built-in mechanism to prevent you from removing a component that is not ready, thereby decreasing the chances for lifting pads.

If you feel the component move, wipe it to one side just slightly and lift simultaneously. The component should be held in the tinned tip.

Remove the component immediately by wiping it on the sponge or tapping it on the stand gently.

5.17 De-soldering with Solder Suckers

Solder suckers use a spring-loaded vacuum to draw up melted solder. The solder needs to be melted either with a soldering iron or hot air, like from a heat gun or rework station. Only small amounts of solder are drawn up at a time, or it often takes multiple attempts to remove a component. Maintaining soldering temperature or remelting a solder joint over-and-over increases the thermal stress on components, other solder joints, and the overall circuit board. Solder suckers can only remove exposed solder, so components covering contact areas, like ball grid arrays (BGAs), have to be removed before the residual solder can be removed.

5.18 Using Solder Sucker to De-solder Components

Step 1: Add Flux to Joint

Figure.5.12: Adding Flux

Flip the PCB to the non-component side. Now, add flux to the joint you intend to desolder. You may use anything ranging from a brush (for semisolid flux) to a syringe (liquid flux) for this purpose, but we find flux dispensing pens to be the most convenient.

Step 2: Get Your Soldering Iron Ready

Choose the right soldering iron tip for the job. Compared to tip temperature, tip shape and size affect the thermal efficacy of a soldering iron to a greater degree. Choose the right soldering iron tip for the job. Compared to tip temperature, tip shape and size affect the thermal efficacy of a soldering iron to a greater degree.

Figure.5.13: Soldering Iron

Step 3: Get the Right Temperature

Fig.5.14: Soldering Iron Tip Temperature

Bring the soldering iron to the correct temperature. This entirely depends on the type of solder used in the joint. If PCB has used lead solder, the require tip temperature may vary ranging from 520 °F (270 °C) to 570 °F (300 °C). However,if you are desoldering a joint on a PCB from a commercial device solderred with lead-free solder,it will require the soldering iron tip to be maintained anywhere between 570 °F (300 °C) to 660 °F (350 °C).

Step 4: Tin Your Iron

Tin the soldering Iron tip, with Solder wire (either Sn63/Pb37).

Fig.5.15: Tinning Solder Iron Tip

Step 5: Melt the Solder

Fig.5.16: Melting Solder

Bring the soldering iron tip to the joint you wish to de-solder from the opposite side of the de-soldering pump. The solder should melt within two seconds.

With your hand on the trigger, immediately touch the tip of the solder sucker/de-soldering pump to one side of the joint.

Step 6: Suck Up the Solder

The moment the solder melts, remove the soldering iron tip, tilt the solder sucker such that the tip encloses the joint, and press the release trigger. These three actions should be executed in one fluid motion. Ideally, each joint should be desoldered within three to four seconds from the time you touch the hot soldering iron tip to it.

Step 7: De-solder All Leads

Repeat this process for all component leads. Be sure to re-tin the soldering iron tip after every couple of joints. The component should come off the PCB with ease at this juncture

. 5.19 De-soldering with Hot SMD tweezers

Figure.5.17: Hot SMD Tweezers

Hot SMD Tweezers can be best visualized as a tool, made of two soldering irons. The idea is to clamp onto the leads on both sides of the component to melt all the solder at once. This limits the overall thermal stress compared to hot air. Once the component is out of the way, the solder will need to be removed another way.

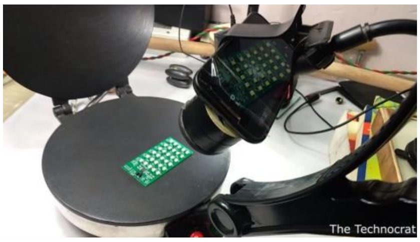
5.20 De-soldering with Hot air solder stations

Figure.5.18: Hot air Solder Station

These stations blow hot air over the repair area to melt the solder, and the component is removed either with tweezers or a vacuum lifter. Once the component is out of the way, the solder will need to be removed another way. Common uses for hot air rework stations include using the solder feature to add extra components to a printed circuit board.

For polarized components, you can use the de-soldering tool on the station to remove the part. After reversing the part, you can then resolder it back onto the board.

5.21 De-soldering with Hot plate



Placing the PCB on a hot plate melt (or reflows) all the solder joints to allow several components to be removed at the same time. This method may unnecessarily stress other components and the board as a whole.

Figure.5.19: Hot Plate De-soldering

5.22 Cleaning PCBs after soldering

5.22.1 Cleaning Flux residues with Swab and IPA:

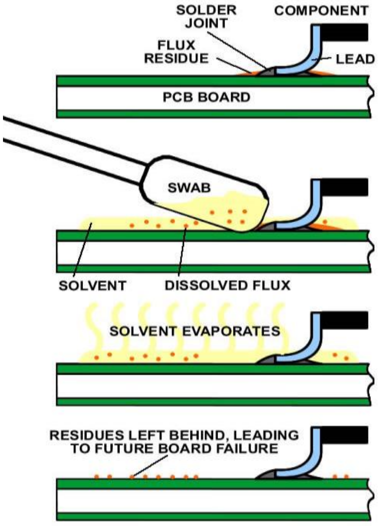


Figure.5.20: Cleaning Flux residues with Swab and IPA.

The most common way to clean flux residues from a repair area is to saturate a cotton or foam swab with isopropyl alcohol or another cleaning solvent and rub it around the repair area. While this may be adequate for no-clean flux, where the goal is a visually clean PCB, this may not be clean enough when more heavily activated fluxes are involved, like RA or aqueous. The dirty little secret is that flux residues will not evaporate along with the solvent. You may dissolve the flux, and some of the residues will soak into the swab, but most of the residues will settle back onto the board surface. Many times, these white residues are more difficult to remove than the original flux.

5.22.2 Brush Cleaning

Brush Clean system is also used as flux remover. The cleaning solvent sprays through the brush, so agitation can be increased by scrubbing while spraying. To absorb the flux residues, a lint-free wiper can be placed over the repair area, and then spraying and scrubbing can occur over the material. Then remove the wipe and brush attachment, and spray over the board for the final rinse.

CHAPTER-6 BGA Desoldering & Resoldering

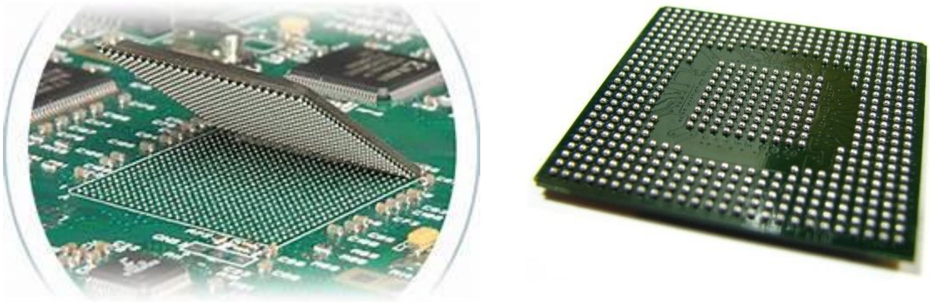


Fig.6.1: BGA packages

6.1 BGA Rework Processes

A **ball grid array** (**BGA**) is a type of surface-mount packaging (a chip carrier) used for integrated circuits. BGA packages(see Fig.6.1), are used to permanently mount devices such as microprocessors. A BGA can provide more interconnection pins than can be put on a dual in-line or flat package.

This chapter describes the general BGA rework process covering the development and criteria for establishing profiles for BGA removal and replacement.

Critical elements of the BGA rework operation, including paste pattern printing and rework profile development, are described in great detail in this BGA rework process outline. There are several technologies which can be used as the heating source, in removal of a BGA, including but not limited to hot air and IR sources. The BGA rework process described herein is generic in nature to the type of heating system used.

6.2 Material required

Solder paste or paste flux

- BGA Rework System including computer-controlled heating source on the part area, PCB preheater capable of heating underneath entire board area, calibrated vision system, automatic vacuum pick up system and data logging functionality
- X-RAY System
- Endoscope or other optical inspection system
- Squeegee
- StencilQuik(TM), Stik Peel(TM) or metal stencil
- Kapton (TM) Tape, Heatshields (TM) or Heatshield Gel
- Hand held soldering iron
- Reflow oven
- Solder wick or solder extraction tool
- Cleaning brush, ESD-safe
- Cleaning Solution or water
- Stereo microscope

The BGA rework process can be methodically worked out by following these process guidelines even with differences in printed circuit boards as well as devices populated on those boards. Years of experience, varied equipment, the right tools and fixturing and a robust BGA rework process, all help to insure high expected yields. While several skills can be trained including but not limited to BGA rework profiling, site preparation and process troubleshooting, having the “real world” experience of hundreds of different boards and part configurations make this a skill which is supported by experiences. By having the right equipment, the BGA rework process will proceed with higher yields. For example, some of the packages and board combinations are better-suited for hot air rework systems while other combinations are better-suited for infrared rework heat sources. A hot air non-contact excavation source is better-suited for sensitive boards susceptible to lifted pads or damaged solder mask while robust printed circuit boards can have the solder wicking approach for site preparation be more appropriate and cost-effective for the BGA rework process. We invite you to go through each of the steps of the BGA rework process as listed at the top of this page.

6.3 Rework of Large Scale BGA Components

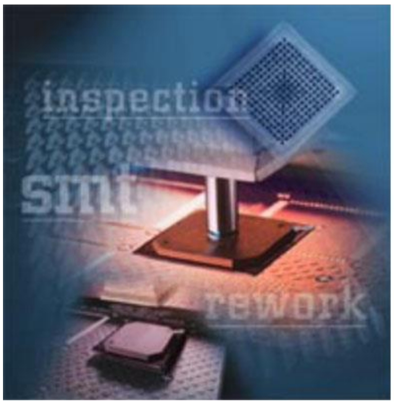


Fig.6.2: Large Scale BGA Component Rework

You've just been given a board with a shorted BGA site. If management decides the board is worth salvaging, there's only one thing to do; remove the BGA component, clean the site, paste the site, and replace the BGA. However, please note that BGA rework at any level can be finicky and challenging.
Yes, it is complicated enough, but the challenge is magnified in this case as this BGA component has 2600 balls at a 1.0 mm pitch.
You're no quitter, so you plow ahead. Removal of the component is delicate but routine for an experienced pro. You get that done with a little cautious foresight and plenty of monitoring thermocouples under and around the site.

Figure 6.3: BGA site vacuum system.

- Next, you find that clearing the site of excess solder is a little more delicate, and you decide to use your BGA rework machine's installed vacuum solder removal system. (See Figure 6.3).

  So far, so good, but next comes the solder paste application at this site. Even after you've chosen solder paste with the best possible slump characteristics, you begin to confront another challenge.

  Proper solder paste application, an acquired skill, even for low pin counts, is crucial for successful BGA rework, and the more pins there are, the more chances there are for uneven paste deposition.

6.4 BGA Solder ball deballing and reballing processes:

Following the de-balling process, re-balling of individual or multiple BGA devices is carried out consisting of the following process steps: BGA components placed in custom fixtures Flux applied to BGA pads Precision aperture stencil positioned above BGA device Fill apertures with solder spheres of new alloy.

a) Component Removal

Figure 6.4: BGA Component Removal-Thermal technique

You need to pre-heat the PCB and then apply heat to the top of the BGA component that you need to remove. Apply hit till the solder melts, and then you can remove or lift by using a vacuum cup.

b) Deballing

You need to apply water-soluble paste flux by using a syringe. Smear the paste across the balls of the device using a gloved finger. With the help of a good blade tip, you can remove the solder balls. Ensure that the surface is flat and does not contain any bumps.

c) Clean the Deballed Part

Clean the PCB surface using isopropyl alcohol and clean off the flux residue from the bottom of the deballed part.

d) Apply Paste Flux

Closely inspect the device to ensure no scratches and lifted pads on the bottom of the device. Apply water-soluble paste flux to this bottom part. Evenly spread the flux with a soft brush till it forms uniform thickness.

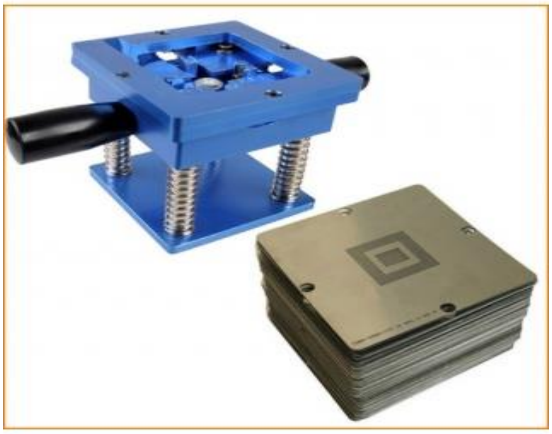


Figure 6.5: Reballing Stencil and Machine

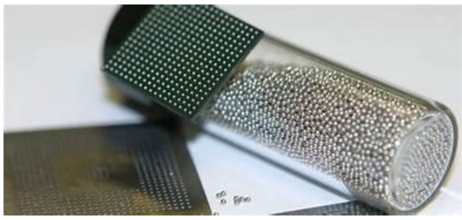


Figure 6.6: Preform Solder

e) **Attach the Preform Ball**

Place the preform ball on the PCB, using Stencil. Ensure that the preform lines align to the patterns on the device.

With a metal stencil, you get one pass; if you try for a second or third, you risk pumping excess paste between the stencil apertures. This can cause paste bridging and, ultimately, a solder short, which is what you were trying to correct in the first place.

Figure 6.7: Edge balls shorting when BGA component "potato-chips" during rework. If, in the effort to prevent shorts, you under-paste several pads on this very large site, it's not inconceivable that you could induce opens under the component. Something as simple as whether or not there is enough room to tape the stencil to the board securely can become an issue. There is no substitute for skill and experience in performing a proper solder paste deposit. Additionally, you place high-temperature spacers under the four corners of the component to prevent shorting due to uneven column collapse or "potato-chipping" at the corners. (See Figure 6.7).

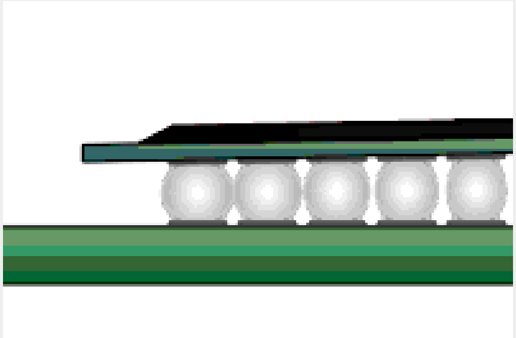


Figure 6.7: Edge balls shorting when BGA component

f) Place Device on Top of the Preform

You need to attach the BGA component to the ball array perform. Ensure that the pattern orientation is correct. Then place the device in a reflow oven. The temperature settings should be proper.

g) Inspect

While the device is still slightly warm, you need to remove the preform from it. Closely inspect the device to ensure that all the balls are correctly transferred to the device and clean it with water and a soft brush. Inspect (preferably with X-Ray) the reballed device to determine if it follows the set standards.

6.5 Conclusion:

Well, you can't hold your breath for an entire BGA reflow cycle, but neither can you breathe easy until the component is properly reflowed and passes an x-ray inspection. But your hard work, preparation, and care pay off in a successfully reworked site.

CHAPTER-7: Rework and Repair on Flex Circuits

7.1 Rework and Repair on Flex Circuits

Flex circuits are used in a variety of applications for the interconnection of conductors which need to be bent or exercised continuously. This interconnection technology has historically been used as a wire interconnection. There are a variety of versions of this type of circuit. One of them is the dual access flex circuit which is a single-sided flex circuit that is manufactured so that the conductive material can be accessed from both sides of the flex. A double-sided flex circuit is a circuit having two conductive layers, one on each side of the base layer within the circuit. Trace patterns, specific to your needs, can be created on both sides of the substrate film. They can be interconnected where desired with copper plated through-holes. A multilayer flex circuit combines several single-sided or double-sided circuits with complex interconnections, shielding and/or surface mounted technologies in a multilayer design. Rigid flex circuits combine the best of both rigid printed circuit boards and flexible circuits integrated. Circuits are typically interconnected between the rigid and flex circuits through plated through-holes.

There are a variety of benefits to the flex circuits. One of the major benefits of a flex assembly is the nearly error-free implementation of wiring in lieu of labour-intensive hand wiring. They are also able to configured, unlike their rigid counterparts, as complex 3-dimensional configurations as they can be contorted in to a variety of shapes. As the name implies, the materials used in flex circuits can be bent back and forth numerous times meaning they can be used in in highly repetitive applications such as on print heads. When weight is an issue, flex circuits are a good alternative to rigid boards and wires as both the dielectric material and conductor runs are very thin.

Over the last several years the flex industry has seen a growth in demand. It is now a 10 billion WWD industry with growth rates in the 7–10% year range.

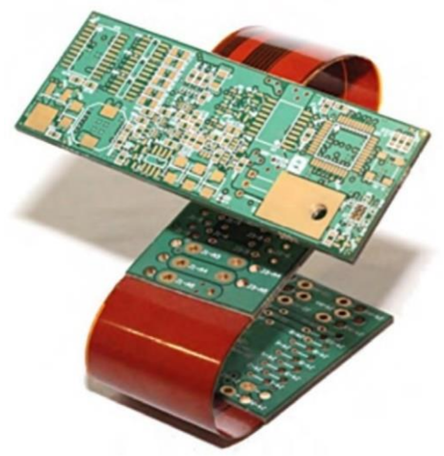


Figure 7.1: Rigid-flex circuit board assembly.

With this surge in usage of flex, the standards for rework (replacing devices while still meeting the initial specification and functionality) and repair (repairing the physical damage on a flex circuit) of these type of electronic interconnection circuits has not kept pace.

There are some rework challenges that come from the very nature of flex circuits. First of all, it is difficult to hold the flex circuit flat. The flexible nature of the Kapton or other base flex material, which makes it so attractive in the application, makes it challenging from a rework perspective. In order to retain the flatness of the assembly, it needs to be taped to be held down. In some cases, a vacuum fixture, a relatively pricey endeavour, is fabricated for flex circuit rework. When placing fine pitch components, the vacuum structure of such fixtures has a significant influence. If the vacuum is directly under some of the leads of a fine pitch component, there is a likelihood any vacuum will “pull” the flex into the hole, preventing the component from contacting the flex circuit lead, thereby resulting in an electrical “open.” For rework paste printing, co-planarity is a challenge when the stencil and surface to be printed are not coplanar. Therefore, paste print deposition using a syringe is often used instead. Sometimes, conductive epoxies are used in interconnecting devices to the flex material. While the curing temperature of these joining materials is much lower than the reflow temperature of more standard solder, it can make a mess. Even when the rework process is engineered properly, many times the limitation on rework is that the marginal cost of the assembly is far less than the burdened rework cost, making the scrap pile a more attractive economic alternative.

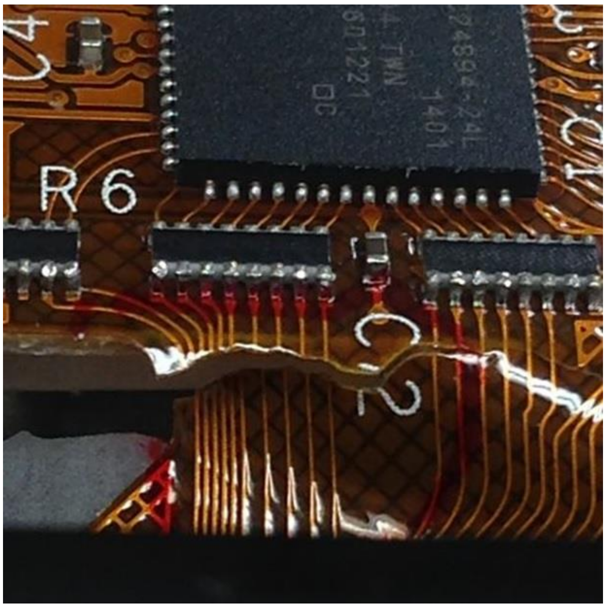


Figure 7.2: Rigid flex ripped circuit.

There are some advantages to reworking flex circuits from a process standpoint. The lower thermal mass compared to a rigid PCB shortens the duration time to reach liquidus when soldering to a flex board. These speeds up the rework process for replacement. In addition, many times this lowers the air temperature required from the hot air system thereby resulting in less potential component damage. The high temperature withstands properties of flex materials such as Kapton, Peek and high-temperature polyimide all give the flex rework process a larger process window.

In terms of industry standards for PCB repair, the IPC 7711/21 Repair and Modification of Printed Boards and Electronic Assemblies covers the rework and repair processes for flex circuits. Each of the processes in the standard are listed in terms of their applicability to flex rework or repair with an “F” in the upper right-hand part of the process documents under the “Board Type” section heading. There is even a flex-specific standard of conductor repair. Conductor repairs on flex is covered in procedure 7.1.1.

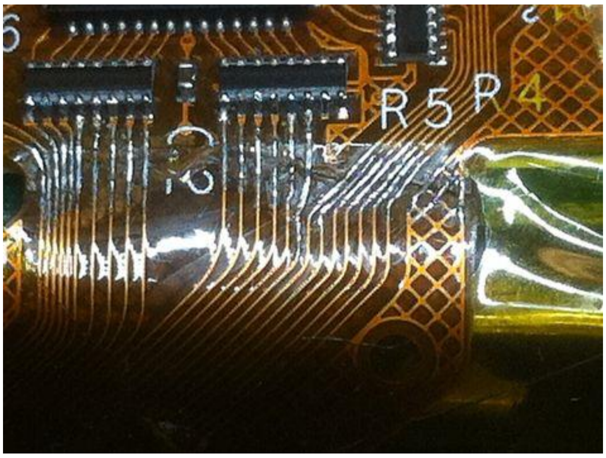


Figure 7.3: Repaired base Kapton material along with conductors.

By way of an example of a conductor repair, Figure 2 illustrates a torn flex circuit as part of a rigid-flex board. The standard process found in IPC 7721 3.5.1 was used to repair the material. The conductor runs had a copper foil jumper installed to replace the damaged conductors and then they were soldered together for further rigidity. The outcome of this repair can be found in Figure 3.

Rework and repair of flex circuit assemblies is evolving and remains a challenge as the industry continues to adopt best practices from the rigid assembly world.

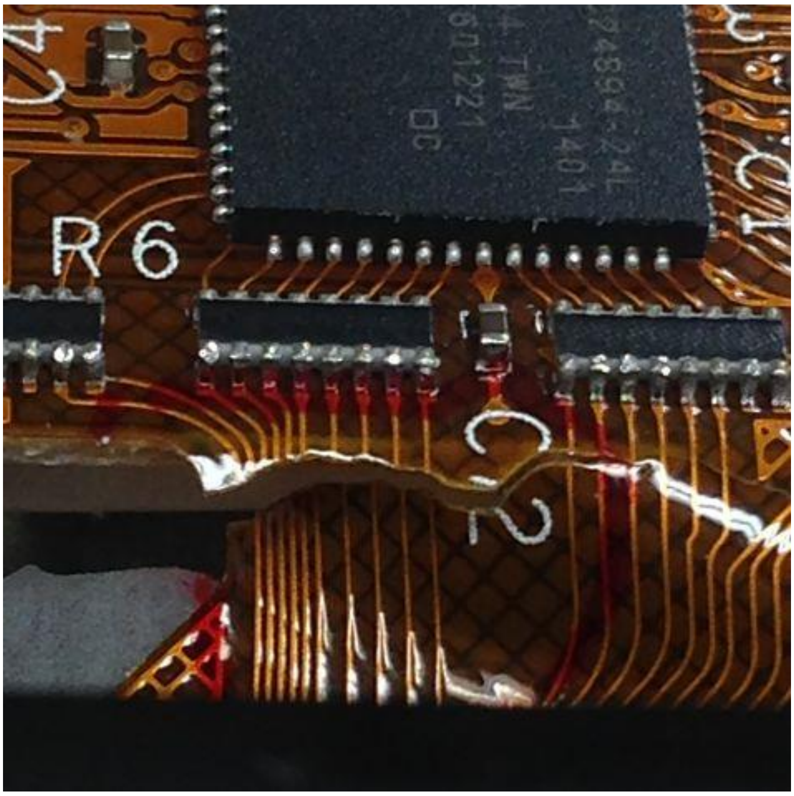


Figure 7.5- Ripped Flex Circuit Material

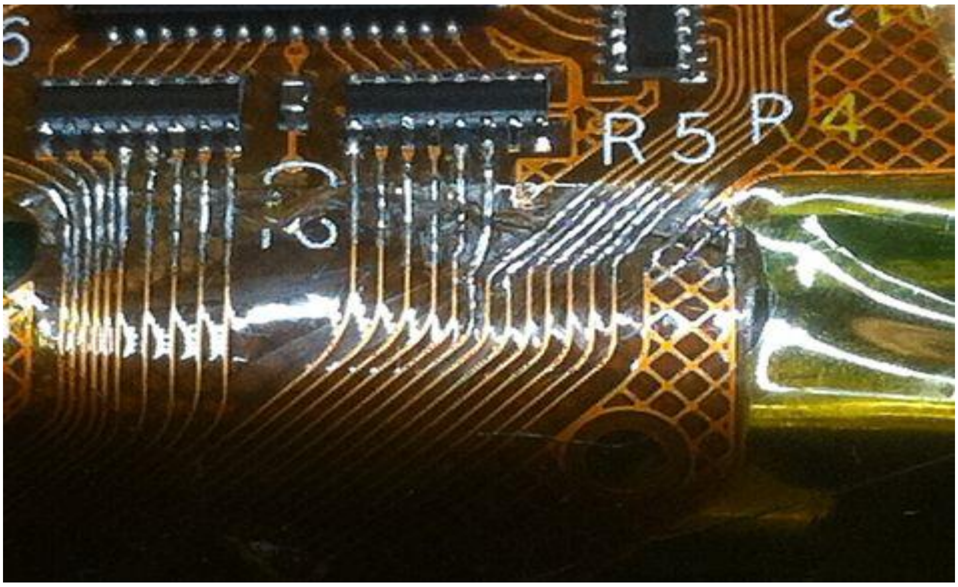


Figure 7.6- Flex Circuit Material Post Repair

Annex-A: ESD Control during Repair & Rework

A1.ESD -An Introduction

ESD generated from the human body can be of the order of several thousand volts. This high voltage pulse enters the electronic device that is touched, resulting in a malfunction or destruction of the IC circuits inside it. In order to prevent the destruction of a product or system due to the intrusion of ESD into an electronic device that was touched, it is necessary to install countermeasure components that suppress or remove ESD.

An electrostatic discharge can damage electronic components. This damage isn't always immediately obvious, but can eventually cause products to fail. Static electricity can also attract contaminants in clean environments and cause products to stick together.

The cost of repairing and replacing products damaged by static can be significant. Way back in 1984, researchers estimated losses in the electronics industry directly attributed to ESD amounted to $18 billion.

ESD first requires a build-up of an electrostatic charge. This occurs when two different materials rub together. One of the materials becomes positively charged; the other becomes negatively charged. The positively-charged material now has an electrostatic charge. When that charge comes into contact with the right material, it is transferred and we have an ESD event. When the charge is released onto an electronic device such as an expansion card, the intense heat from the charge can melt or vaporize the tiny parts in the card causing the device to fail. Sometimes an ESD event can damage a device, but it continues to function. This is a called a latent defect, which is hard to detect and significantly shortens the life of the device.

A2. What is Static Electricity

Figure A1

Electrostatic discharge (ESD) is a swift discharge of electric current between two objects with different charges and different numbers of electrons. This exchange of electrons creates a large electromagnetic field buildup, resulting in ESD. Certain electronic devices are vulnerable to low-voltage ESD. For example, a hard drive is susceptible to just 10 volts.

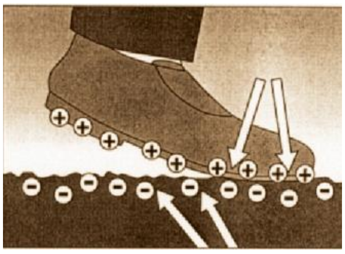


Figure A2.

ESD has several causes, but static electricity and electrostatic induction are the most common. Static electricity is often produced through tribo charging, while electrostatic induction results from the rearrangement of electrical charges as an object. Generally, tribo charging results when the surface of an object gains negative electrons as another object loses electrons and becomes positively charged. When opposite charged objects come into contact with each other, electrons transfer energy and then separate, creating a type of contact electrification of electrical charges.

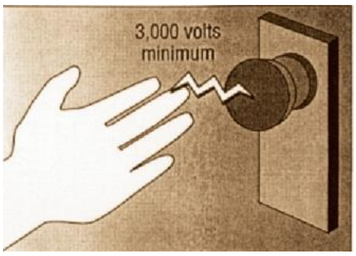


Figure. A3

Static electricity is an imbalance of electrical charges within or on the surface of a material. And, what is electrostatic discharge? Electrostatic discharge (ESD) definition is a rapid, spontaneous transfer of electrostatic charge induced by a high electrostatic field.

Figure. A4

ESD causes two types of electrical equipment damage, as follows:

- Catastrophic: Creates permanent damage
- Upset failure: Nearly undetectable. Damages components, but there may be a degree of continued equipment performance.

Figure. A5

It is believed that all elements existing on Earth in their natural state will always generate static electricity. It can occur not only for solids elements, but also liquids and gases. We have all experienced an electrostatic discharge (or shock) which results in a "shock sensation", a slight noise of a spark and sometimes the spark can even be visible.

Elements that interact with each other can be the same or different (solid, liquid or gases), and it will still generate static electricity.

For example, **lightning is static electricity generated by the friction of frozen particles of water vapor in clouds.**

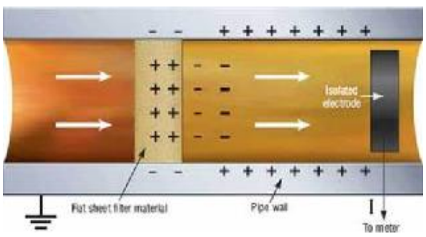
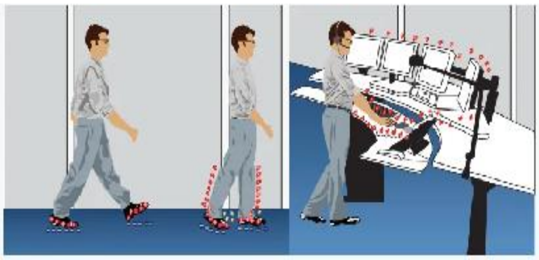


Figure. A6

In most workplace environments, the static generated when people walk is the biggest contributor to random ESD events (or problems caused by electrostatic discharge).



In most workplace environments, the static generated when people walk is the biggest contributor to random ESD events (or problems caused by electrostatic discharge)

Figure. A7

A3. Environmental Impact on ESD Generation & Control

The most significant environmental factor in ESD Control is the relative humidity (RH). II. In very dry areas, humidification is desirable because it makes antistatic materials with “sweat layers”, as it brings an overall reduction in triboelectric charging for all materials.

Table A1.: Typical Static Voltage level generated at different Humidity

Means of Generation	10-25% RH	65-90% RH
Walking across carpet	35,000V	1,500V
Walking across vinyl tile	12,000V	250V
Worker at bench	6,000V	100V
Poly bag picked up from bench	20,000V	1,200V
Chair with urethane foam	18,000V	1,500V

A4. Damages caused by ESD in PCB assembly

HOW STATIC ELECTRICITY CAN DAMAGE ELECTRONIC COMPONENTS

As far as electronics industry is concerned, electronics products tend to become miniaturized and multi-functionalized, integrity of some components maintains rising up. In addition, the internal insulating layer becomes increasingly thinner, interconnecting wire finer, the capacity to withstand applicable voltage reduced. A number of electrostatic sensitive SMDs (Surface Mount Devices) feature shock voltage that is lower than that of electrostatic voltage people are able to sense. However, the electrostatic voltage generated in the process of manufacturing, transportation and storage is far higher than shock voltage, which usually leads components to suffer from hard shock or soft shock. Finally, the SMDs will suffer from failure or their reliability will be dramatically reduced.

According to statistics, among all the causes for electronics failures, ESD accounts for 8% to 33% and the caused damage reach billions of US dollars. In the process of high-tech SMT assembly, therefore, the effective control of ESD is capable of increasing manufacturing efficiency, improving products' quality and gaining profits. So, it's of great significance to carry out effective measures to prevent ESD.

Static electricity, also known as electrostatic discharge (ESD), is an electrical charge at rest, it builds and looks for somewhere to go. The discharge can cause two types of damage: catastrophic failure and upset failure.

A5. Type of Failures:

a) **Parametric failures**
When ESD is capable of altering one or more parameters of a component, such an effect is called parametric failure. This type of ESD damage does not make the component non-functional, but it causes a shift in the values that are given in the datasheet.

b) **Latent failures**

ESD will bring forward **sudden failures or latent failures to sensitive components**. Sudden failure, also called hard damage, may lead components to lose overall functions, making components internal part suffer from permanent failure or opens. Latent failure, also called soft damage, may lead components to be degraded in terms of performance parameters, making them unstably run or partial functions to be degenerated or lost. When it comes to the overall failures caused by ESD, latent failures account for 60% to 90% while sudden failures for 10%, which means most failures belong to latent failure. The nature of latent failure is that the defects of components can be hardly exposed through inspection and it’s difficult to find out the real cause for failure. Plus, rework and processing feature high cost and components’ shelf life will be also shortened.

C) Catastrophic failure

A **catastrophic failure** of an electronic component can be the least costly type of ESD damage as it may be detected and repaired at an early manufacturing stage. Latent damage caused by ESD is potentially costlier since damage occurs that cannot be felt, seen or detected through normal inspection procedures. Latent defects can be very expensive as the product passes all inspection steps and the product is completed and shipped. Latent defects can severely impact the reputation of a company’s product. Intermittent failures after shipping a product can be frustrating, particularly when the customer returns a product, reporting a problem which the factory again fails to detect. It consequently passes inspection and the product is returned to the customer with the problem unresolved

It is critical to be aware of the most sensitive items being handled in your factory. As electronic technology advances, electronic circuitry gets progressively smaller. As the size of components is reduced, so is the microscopic spacing of insulators and circuits within them, increasing their sensitivity to ESD. The need for proper ESD protection increases every day.

A6.ESD Workstation

Figure. A8

A8. Static charges are generated in humans and other insulators by friction and should be dissipated properly before handling with ESD sensitive electronic circuits and components.

ESD Workstations are ergonomically designed to improve productivity and all its accessories are made of ESD safe materials and is grounded properly creating ESD safe work environment.

Figure.A9

A9. Professional series ESD workstation are ergonomically designed, heavy duty ESD workstation for medium to large electronics Labs and Industries handling ESDS boards and components. Static charges generated by turbo charging or by friction on humans and other insulated materials should be properly dissipated before bringing in contact with sensitive electronic components and circuits. This ESD workstation is designed with an aim to provide complete robust ESD safe work environment that boosts maximum productivity and safety to sensitive electronic components and

A7. Setting Up of sample EPA Workstation

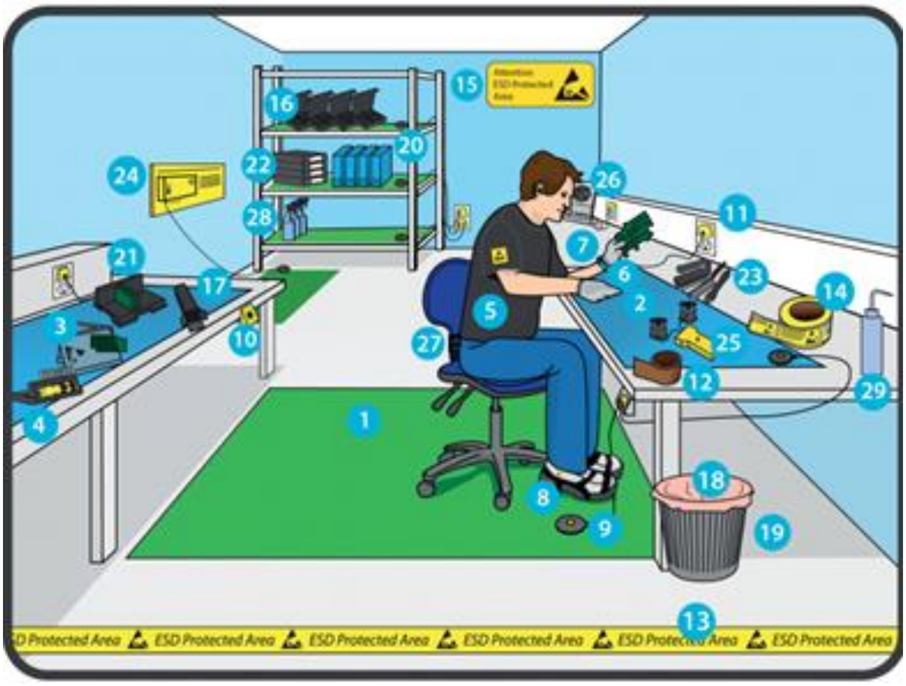


Figure. A10

A8. Labelling of EPA Workstation Content:

1. ESD Matting	**16. ESD safe Transit Packs**
2. Bench Matting	**17. ESD safe Component Boxes**
3. ESD Bags	**18. ESD Bin Liners**
4. ESD Screwdrivers	**19. Conductive Waste Bin**
5. Antistatic Clothing	**20. Antistatic Ring binders**
6. ESD Gloves	**21. Conductive PCB Rack**
7. ESD Wrist Strap	**22. Conductive Tote Bins**
8. ESD Foot Grounder	**23. Conductive Brushes**
9. Grounding Cords	**24. Wrist Strap & Footwear Tester**
10. Earth Bonding Points	**25. Testing Equipment**
11. Earth Bonding Plugs	
12. Antistatic tapes	

13. Floor Marking Tape **14. ESD Labels** **15. EPA Signage**	**26. Ionizing Blower** **27. EPA Chair** **28. IC Workstation & Mat Cleaner** **29. ESD Wash Bottle**

ANNEX-B: TERMS & DEFINITIONS

B1. The following definitions apply to the use of this document.

PCA – Printed Circuit Assembly

Rework – the act of reprocessing noncomplying articles, through the use of original or equivalent processing, in a manner that assures full compliance of the article with applicable drawings or specifications.

Modification – the revision of the functional capability of a product in order to satisfy new acceptance criteria. Modifications are usually required to incorporate design changes which can be controlled by drawings change orders, etc. Modifications should only be performed when specifically authorized and described in detail on controlled documentation.

Repair – the act of restoring the functional capability of a defective article in a manner that does not assure compliance of the article with applicable drawings or specifications.

Tack Solder – A solder connection commonly used to teem- polarity align and retain a multiload component in place on a PCB during the soldering of the other leads. A tack sol- der connection typically requires additional reflow to form the final solder connection.

B2. Class of Product The user of the product is responsible for identifying the Class of Product. The procedure selected for action to be taken (rework, repair, modification, ...) must be consistent with the Class identified by the user. The three Classes of Product are:

Class 1 – General Electronic Products

Includes products suitable for applications where the major requirement is the function of the completed assembly.

Class 2 – Dedicated Service Electronic Products

Includes products where continued performance and extended life is required, and for which uninterrupted service is desired but not critical. Typically, the end use environment would not cause failures.

Class 3 – High Performance/Harsh Environment Electronic Products

Includes products where continued high performance or performance-on-demand is critical, equipment downtime cannot be tolerated, end-use environment may be un-commonly harsh, and the equipment must function when required, such as life support and other critical systems.

B3. Board Types There are a variety of printed board types that the procedures in this document apply to. When selecting the appropriate rework, repair or modification procedure, the printed board type being worked should be considered. Select a procedure that applies to the printed board type as listed on the procedure. Printed board types include the following:

R. Rigid Printed Boards and Assemblies – A printed board or assembly using rigid base materials only. These may be single-sided, double-sided or multilayered, and may be constructed from base laminate material that spans all approved commercial grades of laminate and includes glass fabric reinforced epoxy and polyimide resin laminates.

F. Flexible Printed Boards and Assemblies – A printed board or assembly using flexible or a combination of rigid and flexible materials, which may utilize electrically non-functional stiffeners and/or cover layers. These may be single-sided, double-sided or multilayered.

W. Discrete Wiring Boards and Assemblies – A printed board or assembly using a discrete wiring technique to obtain electrical interconnections.

C. Ceramic Boards and Assemblies – A printed board or assembly using ceramic as the base material with inter- connections separated by dielectric. The board layers are usually formed by alternate printing or depositing of interconnections and dielectric.

Annexure C: References

C1. Introduction of PCBA repair and rework process

https://www.ipcb.com/technical/7480.html

C2. Reflow and Rework Rules for Flex PCBAs

https://resources.pcb.cadence.com/blog/reflow-and-rework-rules-for-flex-pcbas

C3.PCB Assembly Rework - PCB Rework - Technotronix-Pb free

https://www.technotronix.us/pcb-assembly-rework.html

C4. Circuit Board Rework and Repair Blog

https://www.cuitrewircork.com/general/features.html

C5. QUAD FLAT PACK (QFP) REWORK AND REPAIR PROCEDURE

https://www.ape.com/quad-flat-pack-qfp-rework-and-repair-procedure/

C6. BGA Rework Process | BGA Repair at Solder.net

https://www.solder.net/technical-info/bga-rework-processes/

C7. Tips for Rework of Large Scale BGA Components

https://www.circuitrework.com/features/609.html

C8. Rework and Repair on Flex Circuits

https://smt.iconnect007.com/index.php/article/103748/rework-and-repair-on-flex-circuits/103751/?skin=smt

C9. Flex Circuit Repair and Rework Services - Solder.net

https://www.solder.net/services/flex-circuit-repair/

C10. SMT Rework/Repair and Assembly SMT Rework System Packages

https://www.bomir.com/online/categories,244,SMT-Rework-Repair-and-Assembly.html

C11. Repair and Rework Requirements for SMT Assemblies: SMT Article

https://www.rayprasad.com/repair-and-rework-requirements-for-smt-assemblies

C12. Rework and Repair-DS board

http://pcb.iconnect007.com/index.php/article/45677/step-10-rework-and-repair/45680/?skin=smt

C13. SolderQuik(R) BGA Re-balling

http://www.winslowautomation.com/bga.asp?gclid=Cj0KCQjwu-KiBhCsARIsAPztUF33Jj_hpqFNXqYTnQ8iqZNpqHE3Ze30MuaLrWoAIzQoUv4YvighiNYaAvmVEALw_wcB

C14. The Complete Manual For PCB Rework and Repair by solder4 - Issuu

https://issuu.com/solder4/docs/the_complete_manual_for_pcb_rework_and_repair

C15. Circuit Board Rework and Repair Guides

https://www.circuitrework.com/guides/guides.html

C16. The essential guide to PCB rework and repair - MPE Electronics

https://www.mpe-electronics.co.uk/2021/07/13/the-essential-guide-to-pcb-rework-and-repair

C17. Ultimate Guide to Benchtop PCB Rework & Repair

https://www.chemtronics.com/ultimate-guide-to-benchtop-pcb-rework-repair

C18. PCB Rework - NexLogic Technologies

https://www.nexlogic.com/pcb-assembly/pcb-assembly-rework/