STELLER STUDIOS

PCB Soldering

PCB Soldering

Table of Contents:

1-Overview

Circuit boards can be repaired by contract electronics repair companies, however this is lengthy and costly process and may require such repair companies to develop specific test jigs for the individual boards. Even with a test jig, these external companies still do not have a method to test the applied repairs in the field. If there is an error in the repair, the board will have to be returned, which could cost you much more down time.

Due to these reasons, it is prudent to refresh or increase your electronics repair skills. This seminar is designed to help you make component level repairs.

2-Tools Set

2a-Soldering Station

A typical soldering station is shown in figure 1.

Figure 1: OKI Soldering/Desoldering Station

The soldering iron features a fixed temperature control system, which automatically selects the correct soldering temperature based on the tip inserted into the soldering iron (figure 2).

Figure 2: Soldering Iron

The soldering iron has been ordered with 2 soldering tip types. One tip is a round conical fine point tip, and the other is flat edge tip.

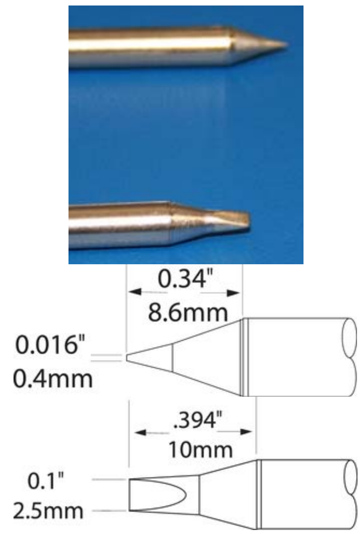

Figure 3: Soldering Tips

Each tip can be used for a variety of tasks. The smaller tip is best used on smaller components where less heat and more precise heating is required. Example of areas to use the conical tip are SMT parts, Dual in Line Packaged (DIP) integrated circuits (IC's), and small leaded (<0.05" diameter) components.

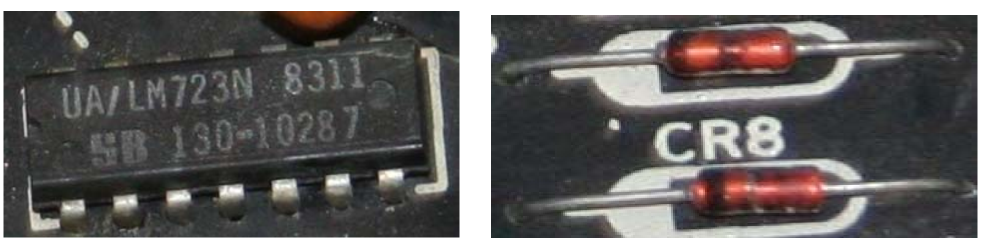

Figure 4: Example of DIP16 and DO-35 Packages

The flat tip is better suited for areas that require more heating and are less sensitive to overheating. For example a through hole connector, T0220 packages, and larger leaded (>0.05” diameter) components.

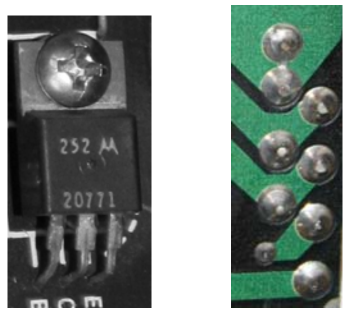

Figure 5: Example of TO220 and Larger Connector Solder Pads

A continuously tinned surface must be maintained on the soldering iron tip's working surface to ensure proper heat transfer and to avoid transfer of impurities to the solder connection.

Before using the soldering iron the tip should be cleaned by wiping it on a wet sponge or driving into the metal sponge. When not in use the iron should be kept in a holder, with its tip clean and coated with a small amount of solder.

The Desoldering Iron shown in figure 6, is used to heat and vacuum up old solder. The OKI unit has a built in vacuum pump and filter cartridge.

Figure 6: Desoldering Iron

The handle of the desoldering iron can be used as a pencil or gun like position. To clean the various desoldering tips an insertion tool can be used. When the desoldering tool is hot insert the tool to clear any blocking solder balls.

Figure 8: Using the Desoldering Iron Cleaning Tool

The suction capabilities of the desoldering tool can be significantly impacted by infrequent maintenance, so checking the filter once per session is a good idea.

Figure 8: Changing the Filter in the Desoldering Iron

There are two desoldering tips, one with a 0.052" suction hole, the other with a 0.061 suction hole. The 0.062" can be used on larger though hole devices. The 0.052 can be used for smaller leaded components.

2b-Solder Wick

There are two types of solder wick available. One is 0.03" wide and the other is 0.08" wide.

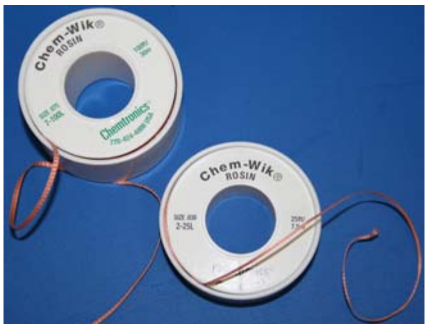

Figure 9: Solder Wick in Stock

Both types have rosin flux in the braid to aid in solder removal. Braid is often used for SMT removal, PCB clean up, and with care, through hole removal. The thinner 0.03” is useful for smaller SMT work, and the thicker 0.08” is great for larger electronic parts.

2c-Solder

The solder used for most repair work will be no clean solder. No clean solder contains a small amount of rosin in its core. There is not enough to leave to large rosin deposits on the PCB, making the clean up after the re-work much more manageable. The type solder used is often referred to as Eutectic. Eutectic solder is made up of 63 % tin and 37% lead and has lower melting temperature (183 degrees Celsius).

Figure 10: 63/37 Eutectic No Clean Solder

2d-Alcohol and Cotton Swabs

Isopropyl alcohol is used to clean the rosin and dirt off of PCBs. Steller Studios uses 99% pure Isopropyl alcohol, typically rubbing alcohol is 70% isopropyl and 30% a mixture of oils and water. Using standard rubbing alcohol on a PCB is not as effective for cleaning, and tends to leave a small layer of oil residue on the circuit board. A bath of 99% pure Isopropyl will help loosen and clean the circuit board.

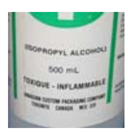

Figure 11: 99% Pure Isopropyl Alcohol

Figure 12: Cotton Swabs

Alternatively a cotton swab soaked in alcohol can clean a particular area of the PCB. Take care to ensure the Isopropyl supply does not get contaminated, do not re-dip dirty swabs.

2e-Side Cutters and Needle Nose

Electronics hand tools, like electrical hand tools, have dedicated purposes. The cutters are delicate and should only be used to trim wire or component leads. The needle pliers are useful for component insertion and lead bending. The wire strippers can be used to strip smaller gauge wire.

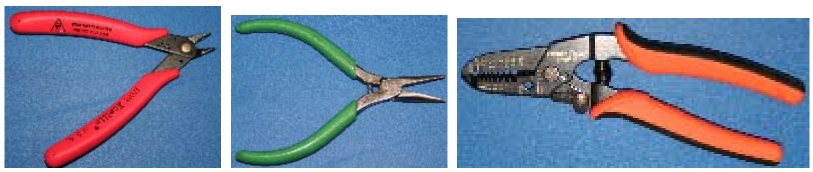

Figure 13: Side Cutters, Needle Nose and Wire Strippers

2f-Tweezers

For SMT tweezers are essential. The curved tweezers are most useful soldering SMT part on the PCB and the straight for removing SMT. For through hole you may find the small needle nose pliers more useful.

Figure 14: Straight and Curved Tweezers

2g-Magnifier

The magnifier is an essential tool used for inspection and fine soldering work. PCB trouble shooting is about 70% visual inspection, the magnifier greatly assists this. Sometimes it is easier to do SMT soldering using the magnifier. When the eye can see more detail, the hand will automatically compensate, and steady itself to a higher degree.

Figure 15: Magnifier

Alternatively a photo can be taken and blow up on the computer to do visual inspections.

2h-Ohm Meter and Continuity Meter

A multi-meter is also essential. A continuity meter will help checking connections and mapping traces, as well check diodes. An ohmmeter can check impedances.

Figure 16: Multi-meter

3-PCB Desoldering Methods

This section deals with ways to de-solder components from a PCB. Depending on the type of component, there are different methods to safely remove the component. The main goal in de-soldering to remove the component without damaging the PCB, the components themselves are thrown away. Components that have seen excessive heating from a soldering iron are more likely to fail. Unless the original component can not be replaced, the original components should be thrown away.

3a-Through Hole

Background

Many people believe through hole components are easier to work on. They are typically larger and can be easier to hold, and the extra lead length can help dissipate the heat. It is also believed that through hole components are more robust and can be designed to better handle vibration issues. A common trick is to wind two or three turns in the component leads, this dampen the vibration the component experiences.

Figure 17: Coiled Resistor Leads to Reduce Vibration Shearing

However through hole components could be seen to have several disadvantages. Through hole components typically take more space on the PCB. The body of device plus the spacing of lead length expands the PCB footprint requirements. Through hole components have more mass. The additional mass makes vibration issues <u>more</u> prevalent.

In an assembly through components require more manufacturing steps. For instance to install a 0.4" resistor (typically considered a ¼ watt resistor) requires 5 steps.

1- The component will have to cut from its tape reel.
2- The leads will have to be bent to the correct size.
3- The component will be placed in the through holes on the PCB
4- Each copper pad will have to soldered to the lead.
5- The soldered components will likely need an additional trim after soldering.

6-After allowing the PCB's to cool, it will have to be washed. For water soluble flux, hot water and scrub brush are used, or a dishwasher without soap. For rosin flux use an Isopropyl bath and a tooth brush to scrub the PCB clean.

7- Visual inspection then looks for misaligned components, solder bridge, or missing solder.

8-Lastly any necessary rework must be done.

If the designer wishes to add the aforementioned vibration coils, another step will have to be added. In the world of electronics manufacturing, every production step added, increases price to the consumer. Careful cost/benefit analysis of the PCB assembly must be done.

Typically removing through hole components involves more steps than surface mount. Though hole removal can lead to PCB damage if excessive heating is used. A typical method involves reheating the solder on the PCB, removing the molten solder (There are several methods for this), removing and trashing the component, and finally cleaning the PCB.

Desoldering Method for Through Hole

There are several ways to successfully desolder a through hole component, the following procedure is a method you can use to remove through hole components. The method is based on the equipment Steller Studios currently has.

<u>Step 1: Clean the PCB</u>

1) Remove the problematic PCB in a static save method, store the PCB in an antistatic bag.

Figure 18: Use an Antistatic Bag for all PCB Transportation and Storing

Seal and label the bag. Include on the label the model, serial number, install location, and date.

2) Once back in the shop use filtered compressed air to clean any major dust and dirt off the PCB.
3) Next soak the PCB in an isopropyl alcohol bath. Allow the PCB to soak for at least 2 hours. Be carefully to cover the bath, as the fumes are noxious and combustible.
4) Remove the PCB from the alcohol bath and use filtered compressed air to blow the circuit board completely dry.

Step 2: Inspect the Circuit Board

1) Use the magnifier and a strong light to carefully inspect the circuit board.

   Look for burnt and /or cracked components. Burned components are typically caused from an over current condition. Look for burn marks which are brown or black depending on the fault. Sometimes the PCB will be damaged too, however before condemning the PCB, take a photo and clean it thoroughly. Sometimes burns look worse than they really are, the only way to find out if there is real PCB damage is too clean the PCB and re-exam it.

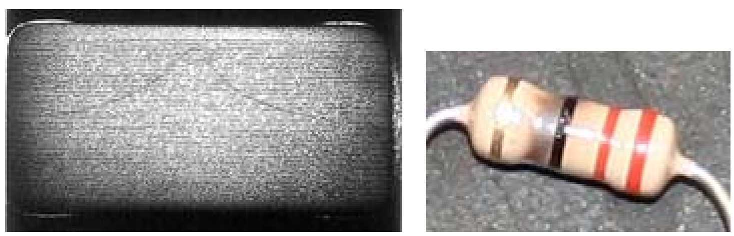

Figure 19: A Crack in a Ceramic Capacitor and a Burnt Resistor

   Cracked components can be caused by vibration. However over voltage conditions on ceramic components is also a very common cause. Often cracks in ceramic capacitors are hard to see. If you suspect a ceramic capacitor, just replace it.

   Look for broken and/or lifted traces. Often heat will cause copper traces to lift and/or break. Sometime traces can shear off if the PCB is stressed. Look for moving bands of copper, scratches or small holes.

Figure 20: Cut Trace

Look for discoloured and/or disfigured components. Typically when capacitors see an over voltage condition they will be damaged. Ceramic capacitor will likely burn or darken in colour. Electrolytic capacitors will bloat or explode. If the board gave off some puffs of smoke, most likely it was a capacitor pushed beyond its specifications.

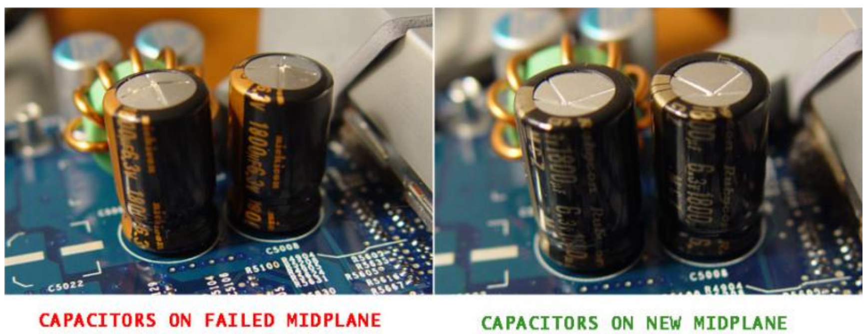

Figure 21: Over Heated Capacitors

Look for cold and/or cracked solder joints. Inspect the solder joints for cold solder joints (not silver and shinny). This means that the solder was cooled and immediately reheated. The cold solder may stay solid while new molten solder is added. When molten and solid solder is heterogeneously mixed in a solder joint, it can lead to cracking. With newer lead free circuit solder it is often hard to tell a cold joint from a good joint because the cooled lead free solder has grayed appearance.

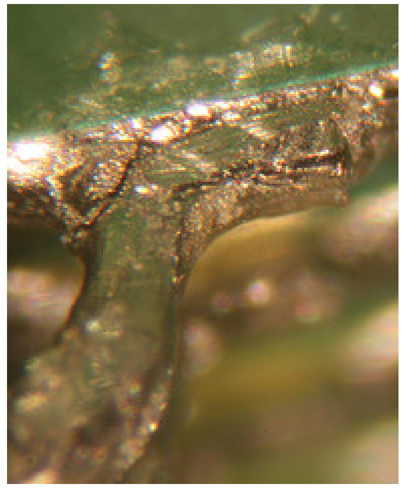

Figure 22: Cracked and Cold Solder Joint

Also note that most modern electronics manufacturing facilities use water soluble flux. Water soluble flux can be easily washed off with hot water, unlike rosin flux which requires isopropyl alcohol or stronger cleaners. The danger of water soluble flux is that when it dries becomes conductive. If water soluble flux is not cleaned off it can cause short circuits.

<u>Step 3: Desolder the Damaged Through Hole Components</u>

1) Optionally you can use a small eraser to remove any oxidation on dry solder joints.

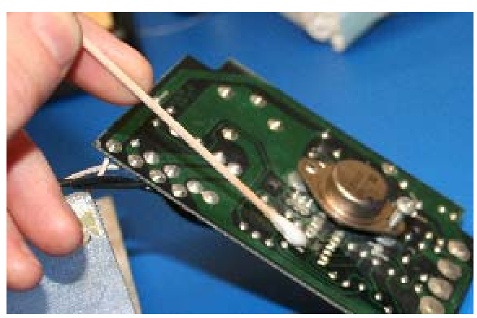

Figure 23: Re-Clean the Desoldering Area

2) Apply a small amount Isopropyl alcohol to each solder joint that needs to be desoldered. Use the cotton swab to this.
3) Identify the solder joints that need to be desoldered, use a small marker to place a dot beside each pad to be desoldered.
4) Double check to ensure you have marked the correct pads.

5) Using the OKI soldering station turn the power on.

Figure 24: Power Button on the OKI Soldering Station

Figure 25: Select the Soldering Iron on the OKI Soldering Station

6) Select the soldering iron and apply a small amount of solder to each pad to be desoldered. Try to do this quickly, as to not over heat each joint. The goal is to see the entire joint become molten. You will find as the new solder is added the old joint become molten much faster, than had you not added any fresh solder. As solder ages its melting point increases, therefore it will take more heat to make it all melt. Mixing in some new solder will help ensure all the solder melts at a lower temperature and more quickly under the desoldering iron.
7) Select the correct tip size for the desoldering iron. The hole on the desoldering iron should fit over the lead of the component to be desoldered.
8) Select the desoldering tool on the OKI soldering station. The iron will begin to heat.
9) Place the tip of the solder joint into the hole of the desoldering iron.
10) Allow the solder to become molten, as it heat gently rock and desoldering tip around the pad.
11) When the solder is molten, push the trigger on desoldering iron. This will activate the vacuum and should suck up the molten solder. If it doesn't clean the joint perfectly, move onto the next joint and let the first joint cool. After the first round of solder joints is finished return to any the need additional work. In some case it may be easier to apply a bit more fresh solder to the joint, in an attempt to mix the old solder with the new.

Step 4: Pad Clean up

1) Clean the desoldered pad with a cotton swab and Isopropyl.
2) Use the rosin solder wick to suck up any remaining solder.
3) Move the wick in a soft scrubbing fashion until the pad appears like new.
4) Clean the pad with a cotton swab and Isopropyl.

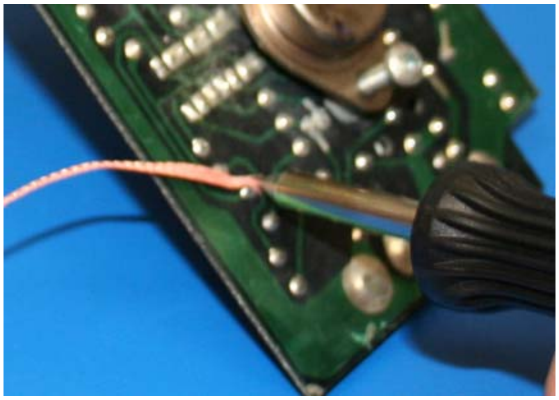

Figure 26: Pad Clean Up With Rosin Solder Wick

3b-Surface Mount(SMT)

Background

Surface mount technology allows the designer to create smaller more densely populated circuit boards. Surface mount components typically have a lower profile. Since surface mount components have no leads, they are soldered directly to the PCB on a single side (or both sides). This means stray inductance, and stray capacitance are minimized.

Manufacturing cost can be greatly minimized as the amount of production steps involved is reduced.

Desoldering Method for SMT

There are several ways to successfully desolder a surface mount components, the following procedure is a method you can use to remove SMT components. The method is based on the equipment Steller Studios currently has.

<u>Step 1: Clean the PCB</u>

1) Remove the problematic PCB in a static save method, store the PCB in an antistatic bag.

Figure 32: Use an Antistatic Bag for all PCB Transportation and Storing

   Seal and label the bag. Include on the label the model, serial number, install location, and date.

2) Once back in the shop use filtered compressed air to clean any major dust and dirt off the PCB.
3) Next soak the PCB in an isopropyl alcohol bath. Allow the PCB to soak for at least 2 hours. Be carefully to cover the bath, as the fumes are noxious and combustible.
4) Remove the PCB from the alcohol bath and use filtered compressed air to blow the circuit board completely dry.

Step 2: Inspect the Circuit Board

1) Use the magnifier and a strong light to carefully inspect the circuit board.

Look for tombstones. A tombstone is an expression used to describe a component stands straight up or soldered to one pad only. It happens frequently in production with two terminal SMT devices, such as resistor and capacitors. When the component is in the reflow stage if one terminal has less solder then the other, the mass and movement of the extra solder becoming molten can pull the component off the opposite pad. Often the part stands straight up, like a tombstone.

Excessive heat can cause tombstones to occur occasionally when the PCB is heated enough in a fault condition. If a pad was lacking solder, and heat is applied the component can sometime lift off the pad.

Look for discoloured and /or cracked components. Burned components are typically caused from an over current condition. Look for burn marks which are brown or black depending on the fault. With SMT the PCB is often burnt, because the component is physically closer to the PCB. But again, before condemning the PCB, take a photo and then clean it thoroughly. Sometimes a burn looks worse than it really is. The only way to find out if there is real PCB damage is too clean the PCB and re-exam it.

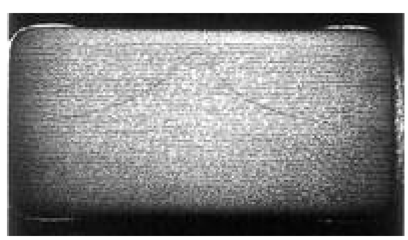

Figure 33: A Crack in a Ceramic Capacitor and a Burnt Resistor

Cracked SMT components are often caused by vibration or excessive heating. However over voltage conditions on ceramic components is a common cause of cracking. Often cracks in ceramic capacitors are hard to see. If you suspect a ceramic capacitor, just replace it.

With all SMT ceramics soldering is a serious issue. Thermal shock must be kept to a minimum. When soldering a SMT ceramic components follow the instructions in the section 4b.

Look for broken and/or lifted traces. Often heat will cause copper traces to lift and/or break. With SMT trace lift is less prevalent, because typically there is less mass involved. The SMT components are smaller and closer to the PCB, so the forces they exert on the copper traces are reduced. Look for moving bands of copper, scratches or small holes.

Figure 34: Cut Trace

Look for cold and/or cracked solder joints. Inspect the solder joints for cold solder joints (not silver and shinny). This means that the solder cooled and was reheated. The cooled solder may stay solid while new molten solder is added. When molten and solid solder are mixed in a solder joint it can lead to cracking. With newer lead free circuit solder it is often hard to tell a cold joint from a good joint because the cooled lead free solder has grayed appearance.

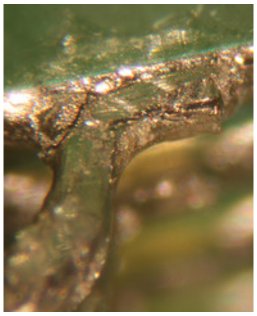

Figure 35: Cracked and Cold Solder Joint

Also note that most modern electronics manufacturing facilities use water soluble flux. Water soluble flux can be easily washed off with hot water, unlike rosin flux which requires isopropyl alcohol or stronger cleaners. The danger of water soluble flux is that when it dries becomes conductive. If water soluble flux is not cleaned off it can cause shorts.

Step 3: Desolder the Damaged SMT Components

1) Apply a small amount Isopropyl alcohol to each solder joint that needs to be desoldered. Use the cotton swab to this.

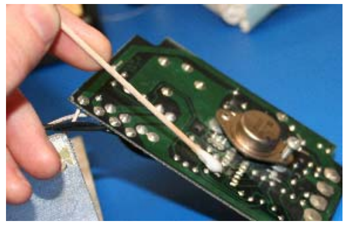

Figure 36: Re-Clean the Desoldering Area

2) Put the focusing nozzle on the hot air gun.
3) Set the hot air gun on mid temperature.
4) Use the tweezers in one hand to hold the component to be desoldered.
5) With the other hand gently heat the pads with a smooth circular motion. Be carefully not to overheat the good nearby components.
6) Gently pull the component away from the PCB. No force will be necessary, if the component resists at all do not pull it. You will likely damage the pads and traces.
7) Continue this procedure for each component.

Step 4: Pad Clean up

1) Clean the desoldered pad with a cotton swab and Isopropyl.
2) Use the rosin solder wick to suck up any remaining solder.
3) Move the wick in a soft scrubbing fashion until the pad appears like new.
4) Clean the pad with a cotton swab and Isopropyl.

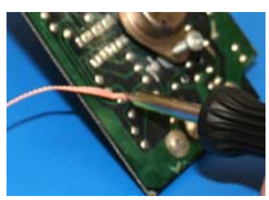

Figure 39: Pad Clean Up With Rosin Solder Wick

4-Soldering Methods

4a-Through Hole

Background

Soldering is the process of joining two metals by the use of a solder alloy. Faulty solder joints remain one of the major causes of equipment failure. The importance of workmanship in soldering cannot be overemphasized.

Solder used for electronics is a metal alloy, made by combining tin and lead in different proportions. You can usually find these proportions marked on the various types of solder available.

With most tin/lead solder combinations, melting does not take place all at once. Fifty-fifty solder begins to melt at 183°C (361°F), but it's not fully melted until the temperature reaches 216°C (420°F). Between these two temperatures, the solder exists in a plastic or semi-liquid state.

The plastic range of a solder varies, depending upon the ratio of tin to lead. With 60/40 solder, the range is much smaller than it is for 50/50 solder. The 63/37 ratio, known as eutectic solder has practically no plastic range, and melts almost instantly at 183°C (361°F).

The solders most commonly used for hand soldering in electronics are the 60/40 type and the 63/37 type. Due to the plastic range of the 60/40 type, you need to be careful not to move any elements of the joint during the cool down period. Movement may cause what is known as disturbed joint (Often referred to as a cold joint). A disturbed joint has a rough, irregular appearance and looks dull instead of bright and shiny. A disturbed solder joint may be unreliable and may require rework. Lead free solder often has appears dull even if the joint is not disturbed.

When the hot solder comes in contact with a copper surface, a metal solvent action takes place. The solder dissolves and penetrates the copper surface. The molecules of solder and copper blend to form a new alloy, one that's part copper and part solder. This solvent action is called wetting. Wetting can only occur if the surface of the copper is free of contamination and from the oxide film that forms when the metal is exposed to air. Also, the solder and work surface need to have reached the proper temperature.

Although the surfaces to be soldered may look clean, there is always a thin film of oxide covering it. An eraser can help wipe some of this off, is a good idea to rub an eraser over the pad to be soldered first. Clean the surface with alcohol and then add flux.

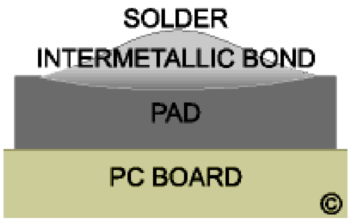

Figure 40:'Wetting' occurs when molten solder penetrates a copper surface.

Flux is very corrosive at solder melt temperatures and accounts for flux's ability to rapidly remove metal oxides. Flux must melt at a temperature lower than solder so that it can do its job prior to the soldering action. It will volatilize very rapidly; thus it is mandatory that flux be melted to flow onto the work surface and not be simply volatilized by the hot iron tip to provide the full benefit of the fluxing action. There are varieties of fluxes available for many purposes and applications. The most common types include: Rosin - No Clean, Rosin - Mildly Activated and Water Soluble.

When used, liquid flux should be applied in a thin, even coat to those surfaces being joined and prior to the application of heat. Cored wire solder and solder paste should be placed in such a position that the flux can flow and cover the joints as the solder melts.

Before solder is applied; the surface temperature of the parts being soldered must be elevated above the solder melting point. Never melt the solder against the iron tip and allow it to flow onto a surface cooler than the solder melting temperature. Solder applied to a cleaned, fluxed and properly heated surface will melt and flow without direct contact with the heat source and provide a smooth, even surface, filleting out to a thin edge. Improper soldering will exhibit a built-up, irregular appearance and poor filleting. For good solder joint strength, parts being soldered must be held in place until the solder solidifies.

If possible apply the solder to the upper portion of the joint so that the work surfaces and not the iron will melt the solder, and so that gravity will aid the solder flow. Selecting cored solder of the proper diameter will aid in controlling the amount of solder being applied to the joint. Use a small gauge for a small joint, and a large gauge for a large joint.

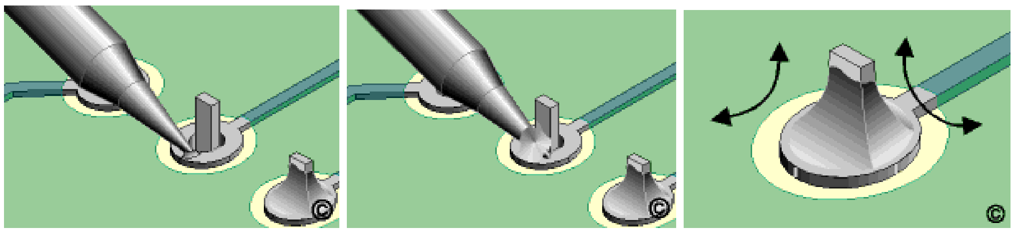

Figure 41: Proper Soldering Technique.

Solder joints should have a smooth appearance. A satin luster is permissible. With Lead free solder do not expect to see shiny solder fillets. Often lead free joints appear dull, but are still expectable.

The joints should be free from scratches, sharp edges, grittiness, looseness, blistering, or other evidence of poor workmanship. Probe marks from test pins are acceptable providing that they do not affect the integrity of the solder joint. Smooth clean voids or unevenness on the surface of the solder fillet or coating are acceptable. A smooth transition from pad to component lead should be evident. The following is a step by step method to solder through hole components.

1) Use a small eraser to remove any oxidation from the solder pads.
2) Apply a small amount Isopropyl alcohol to each pad that needs to be soldered. Use the cotton swab to this.
3) Select and install the appropriate soldering tip.
4) Using the OKI soldering station turn the power on.
5) Allow the soldering tip to warm up for 1 minute.
6) Insert the through hole component into the PCB.
7) Flip the PCB over and ensure the component does not shift or slip. If the component does shift, you can try bending the leads to keep the component installed. Another method is to solder the component from the top first, then flip the PCB over and solder the bottom pads as well.
8) Put a small dot of flux on each pad to soldered (not always necessary).
9) Tin the soldering tip by melting a small amount of solder to it.
10) Clean the tip with wire pad. The wire pad is preferable to the sponge as it does not lower the tips temperature.
11) Touch the soldering tip to pad and heat the pad. After 3 seconds or so, feed the solder into the joint as shown in figure 41.
12) After all the soldering on the PCB is completed, clean the board with isopropyl alcohol (or water if water soluble flux was used) to remove any flux residue.
13) With compressed air blow the board dry.

4b-Surface Mount(SMT)

Surface mount at first may appear more difficult to work with due to the smaller size. However after a few exposures to it, it quickly becomes clear how much more efficient surface mount can be. Basically there are two methods for SMT soldering; re-flow oven or hand.

The reflow oven can be as simple as toaster oven or as complicated as a multiple zone oven. Typically the manufacturing steps for a reflow oven are:

1-Apply solder paste to each pad that requires soldering. This typically down with a solder stencil and requires one swiping motion for the whole PCB. However in the case of repair it can be done by hand. On SMT IC such as the one in figure 27 a thin bead of solder paste works well. When the paste is heated it will naturally separate and be drawn to the pads.

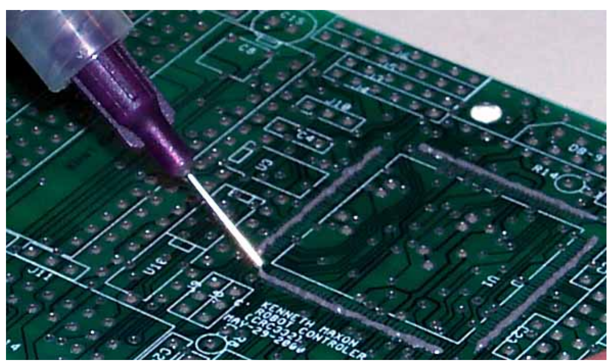

Figure 27: Appling Solder Paste By Syringe

The amount of solder paste used is important the following figure give a visual indication of what too much and too little paste can look like.

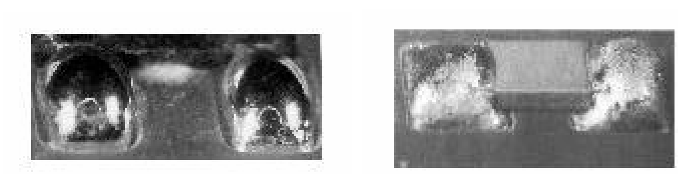

Figure 42: SMT Examples with Too Much Solder Used

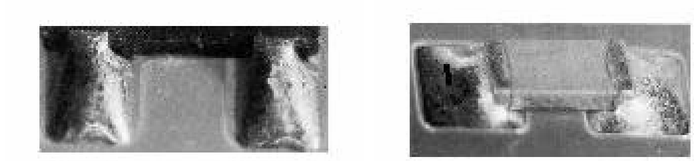

Figure 43: SMT Examples with the Correct Amount of Solder Used

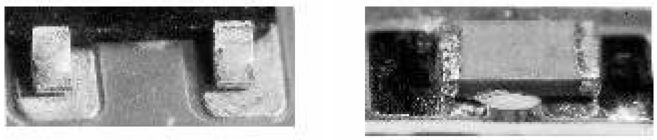

Figure 44: SMT Examples with Too Little Solder Used

2-Postion the component on the pads, the surface tension of the paste keeps the components from moving around.

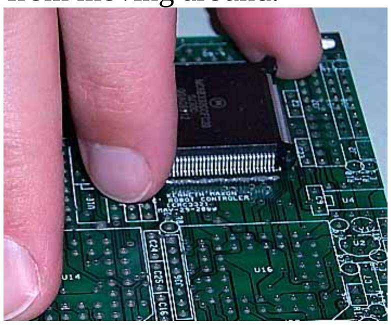

Figure 28: Position the Component

3- Place the PCB with the paste and components in a soldering oven. The soldering oven can be anything from a toaster oven to a 7 seven temperature zone high production unit.

Figure 29: Soldering Ovens

In a typical high output manufacturing oven, a conveyor moves the PCB through different temperature zones with in the oven. Each temperature zone is regulated to a different temperature. As the PCB passes through each zone, it builds a temperature curve for components on the PCB. Every electronic component has different temperature handling capabilities. In order to avoid thermal stressing the component, the oven must gradually ramp the temperature up and ramp it back down again. The more zones an oven has the more precise the temperature curve can be.

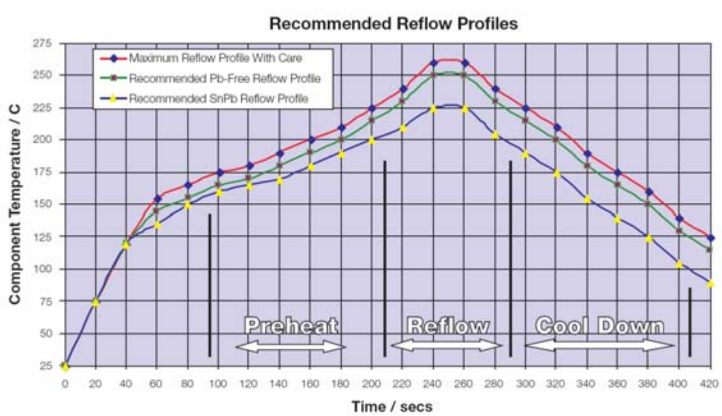

Figure 30: Typical Oven Program for Ceramic Capacitors

This can be done in a simple toaster oven by adjusting the temperature at specific times over a 6.5 min process described in the table bellow.

4 min.	200 deg.	Warm up board and allow temperatures to equalize.
2 min.	325 deg.	Bring temperature up to saturation.
30 sec +	450 deg.	Temperature raised until solder melts and beads at individual pins, then held for 30 additional seconds.
		Tap the oven before cool down...

Table 1: Toaster Oven Temperature Curve

4- After allowing the PCB's to cool wash the PCB. For water soluble flux use hot water and scrub the components with a toothbrush, or put the boards in a dishwasher without soap. For rosin flux use an Isopropyl bath and a tooth brush to scrub the PCB clean.
5-Inspect the boards for misaligned components, solder bridge, or missing solder.

6-Preform any necessary rework.

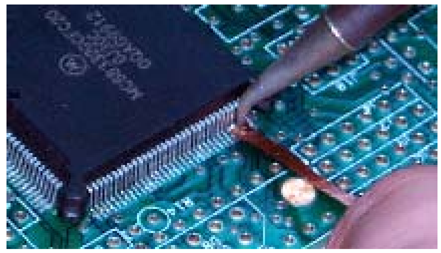

Figure 31: Removing a Solder Bridge

The second method for SMT soldering is hand soldering. This requires a good vision system and some practice time to do well. The type of components and the materials of those components define how much heat can be safely applied. The following a quick list of different types of SMT components.

Chip Resistors
The component body of chip resistors is made out of alumna; an extremely hard, white colored material. The resistive material is normally located on the top. Chip resistors are usually mounted with the resistive element facing upwards to help dissipate heat.

Ceramic Capacitors
These components are constructed from several layers of ceramic with internal metalized layers. Because metal heats up much faster than ceramic, ceramic capacitors need to be heated slowly to avoid internal separations between the ceramic and the metal layers. Internal damage will not generally be visible, since any cracks will be inside the ceramic body of the component.

NOTE
Avoid rapid heating of ceramic chip capacitors during soldering operations.

Plastic Body
Another style of chip component has a molded plastic body that protects the internal circuitry. There are a number of different types of components that share this type of exterior package. The termination styles for plastic chip component packages vary considerably.

MELF
Metal Electrode Face cylindrical components. These may be capacitors, resistors, and diodes. It can be hard to tell them apart - since there is no universal coloring or component designators printed on the component bodies.

The following procedure covers general guidelines for soldering surface mount chip components. The following surface mount chip components are covered by this procedure. While all of these components are different, the techniques for soldering are relatively similar

1) Add liquid flux to one pad.
2) Prefill the same one pad with solder.

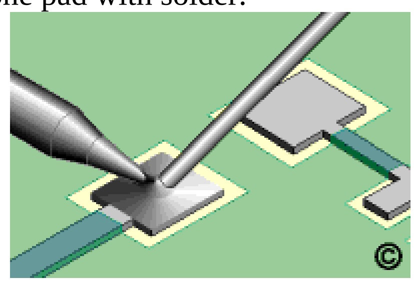

Figure 32: Adding Solder to One SMT Pad

3) Clean the area with Isopropyl alcohol and cotton swab.
4) Add more liquid flux to both pads.
5) Place the component in position and hold it steady with a wooden stick or tweezers so that the soldering iron won't push the component out of alignment.
6) Place the soldering iron tip at the junction between the prefilled pad and component lead. Flow the solder until the component drops down and is soldered in position. Apply additional solder as needed.

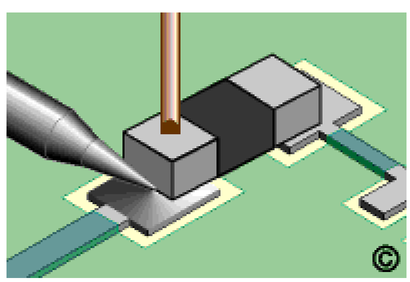

Figure 33: Fixing One Side of a Component

7) Remove the tip. Wait a moment for the solder to solidify before soldering the other side of the component.

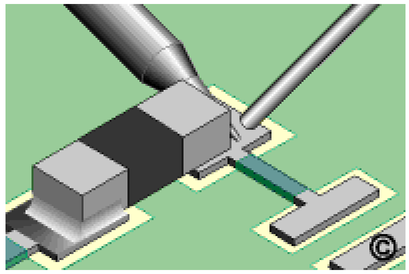

Figure 34: Fixing the Second Side of a Component

8) Clean with Isopropyl alcohol and a cotton swab, and inspect.

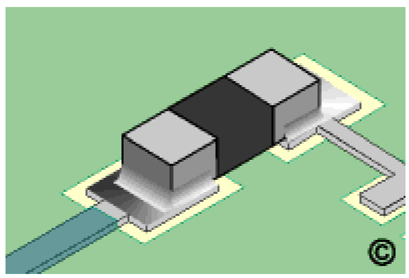

Figure 35: The Soldered SMT Component