PCB Assembly Techniques

by Harvey Twyman

I'M SURE YOU'VE NOTICED THE RAPID DECLINE IN THE AVAILABILITY OF CERTAIN DUAL-IN-LINE COMPONENTS. THE DEVICES MIGHT STILL BE AVAILABLE BUT ONLY IN THEIR SURFACE MOUNT FORM. SO HOW DO YOU CONSTRUCT PCBS USING THESE TINY DEVICES?

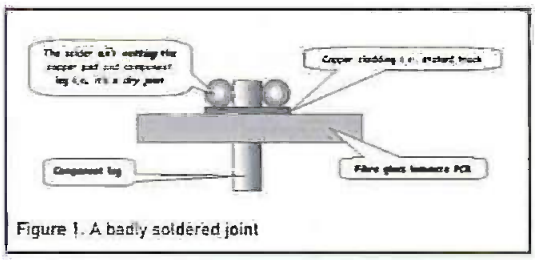

Figure 1. A badly soldered joint

To enable you to keep up with the technology, this article contains a series of special techniques that only require low cost hand tools. It's mainly intended for Prototype PCB designs where manufacturing costly Plated-Through-Hole PCBs for a 'One-Off' is uneconomic.

Working Practices

We know the term 'Design for Manufacture', however here we need to 'Design for Prototype'. The techniques described in this article require certain 'working practices'. These techniques described later can only be successful if the guidelines below are followed:

CAD Considerations

The PCB Design will need modification at the CAD Level to enable these techniques to be workable: VIA pads need to be at least 55 to 60mil diameter if the drilling is to be done by hand. SMT Rectangular Pads must be longer so that they protrude out and are visible even when the device is sitting on the pad. This is so that you can observe and create a reliable joint when manual soldering.

Lighting

Use 2 standard 60W desk lamps one either side of your work area and placed low to the bench to give maximum intensity. This will illuminate the work area adequately and not produce shadow. Visual inspection is the key to success. Most faults are visible with the lighting described above. With these techniques you have to rely totally on your own eyes for quality control. The more light you have, the more you'll see.

Keeping Everything Clean

Before any soldering work is done on the pads they need to be cleaned with IPA (see below) using a soft toothbrush style brush to remove oxides. The PCB will re-oxidise within an hour, so clean just before you intend to solder. IPA (Isopropyl Alcohol) is used in aerosol sprays as a solvent and is a common component of such products like magnetic tape head cleaning solutions. It has the advantage that it evaporates quickly.

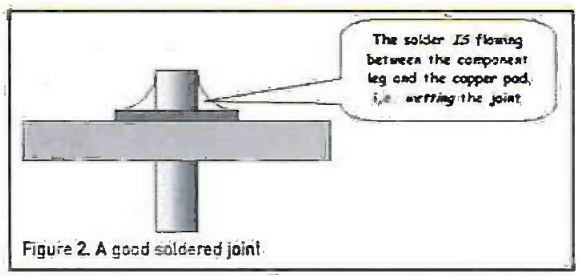

Figure 2. A good soldered joint

Hand Tools

Standard tools are adequate for the job, except for the ones mentioned below:

Soldering Iron

Modern Temperature Controlled types are adequate. The solder tip needs to be pointed thus giving access to individual SMT joints. Pointed tips when used on SMT Pads have an added advantage of less heat capacity. So once placed on an SMT pad the temperature drops rapidly. This has the advantage that quick joints can be made on the tiny pads without them getting damaged.

Tweezers

Don't use sharp ones! These may damage delicate SMT resistors. Flat ended types are preferable as rounded ones can't be used to pick up the smaller SMT devices. Solder Braid: use the narrowest grade you can. Fat ones take too long to heat up which may damage the pads.

Inspection Eyeglass

These are available in different magnifications. A 8x magnification is adequate. Using any higher doesn't display enough board area. There are some types available with a built-in measuring facility. These are important for checking actual PCB pad and drill hole sizes.

Cut and Crop Tool

Also known as Cut and Clench or Cut and Crimp. The tool automatically squashes the wire flat before cutting above it. This tool is used in the VIA Technique described later.

Heat Gun

This type of tool is required for removing (reworking) SMT components, This is the only tool that needs a certain amount of training to use successfully. This will be described later. Heat Guns come in 2 forms: Paint Strippers are relatively cheap and generate a wide column of very hot air with limited control over the air flow. These are more suited for removing the larger SMT devices or where heating a larger surface area of the PCB is required. Hot Air Tools are designed to heat small areas and usually have a proper air velocity control to regulate air flow and thus the temperature. Thus these are more suited for removing the smaller SMT devices. Generally though, as we're more interested here in a low cost solution, the techniques described later will be referring to the paint stripper type of tool.

Good and Bad Joints

The knowledge of how to create good and bad Joints is essential. For those of you already familiar with the subject, please bear with me as beginners may be participating.

The reasons for a bad joint

- The copper cladding is oxidised – Clean with non-abrasive methods: Scrub with

small brush using IPA solvent, re-solder within a few minutes before oxidisation occurs again.

- Soldering iron bit needs wetting – press the tip into the special wetting paste provided.
- Not applying the solder tip and solder to the joint simultaneously.
- Not enough heat being applied to both the component leg and pad.

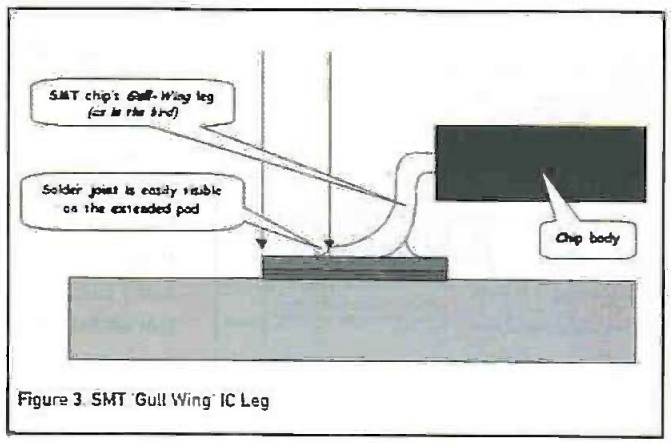

Figure 3. SMT 'Gull Wing' IC Leg

Facts about PCBs and Soldering

- The copper cladding is glued to the fibre glass laminate.
- The melting point of the solder is 180°C.
- The melting point of the glue is 150°C.

Therefore when the solder is molten, the copper cladding is 'floating' on molten glue! The reason why the glue's temperature is lower than the solder's is by design. This is to insure that the copper cladding's expansion is unimpeded by the glue. This means that PCB pads are very vulnerable during soldering.

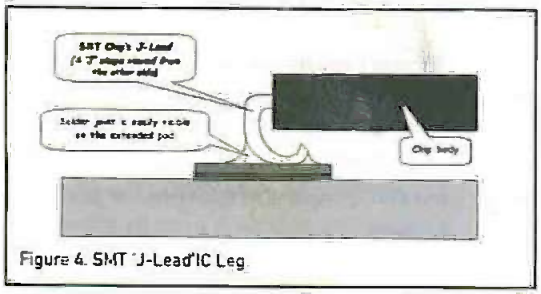

Figure 4. SMT 'J-Lead' IC Leg

Figure 1 shows an example of a badly soldered joint; Figure 2 shows an example of a good one.

Soldering Rules

1. Take care not to press too heavily on the pads, otherwise the pads will lift up and will be very difficult to repair.
2. Moving components before the solder has solidified may again either lift the copper tracks or cause a dry joint.
3. Apply the soldering iron tip and solder to the joint simultaneously and keep there until the solder flows correctly over the joint.
4. Remove the soldering iron tip immediately after the smoke from the flux within the solder disappears, approximately 2 seconds after the solder melts.
5. Only use silver-loaded solder for SMT joints (2% Silver).

Tin Plating

Don't tin plate your PCB unless the plating solution is new! The solution can become heavily contaminated from being used just a few times. These contaminants make it much more difficult to solder. This rule is essential particularly for SMT PCBs. Prototype PCBs don't need tin plating anyway. They still remain clean looking years later. They won't go green unless exposed to severe climatic conditions.

SMT Pads

Surface Mount Pad sizes seem to vary with every designer. They all have their own criteria. Here we are Designing for Prototype so need to have pads that are longer than normal.

PAD Specification for SMT Components:

The SMT Pad must be designed in the CAD stage specifically for hand soldering and...

- can be the same width or wider as the SMT leg.
- must be longer than the IC leg positioned on it, to allow ease of visual inspection and hand soldering of the joint.

Figure 3 is an example of an SMT 'Gull Wing' IC Leg; Figure 4 shows an SMT 'J-Lead' IC Leg.

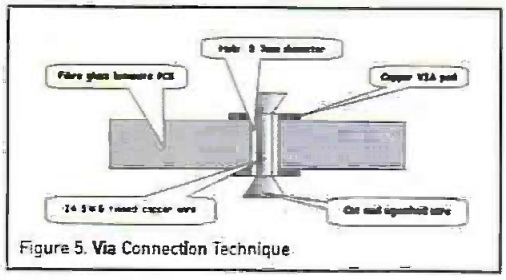

Figure 5. Via Connection Technique

VIA Holes

Double Sided PCBs made as a Prototype probably won't have Plated-Through-Holes. So how can you cope with VIA holes?

Some of the tracks on a double sided PCB connect from one side of the board to the other. Most of them connect via a component's leg, but some don't. These connections that don't are called VIAs.

On commercially manufactured PCBs these VIAs are produced by a system called Plated-Through-Hole. This process electroplates all the inside of the holes with copper.

However a simple technique is described below that doesn't require the expensive Plated-Through-Hole technology.

VIA Connection Technique (Figure 5)

A special Cut and Crop Tool is used to produce a reliable connection through the VIA hole pads. There are various names used for this tool, including 'Cut and Clench' and 'Cut and Crimp'.

Using 24 SWG tinned copper wire: Place the wire in the VIA hole. (Hole should be 0.7mm diameter); Use the cut and crop tool to produce the result; Solder both sides of the wire link.

IC Sockets

Without Plated-Through-Hole Technology, IC Sockets have to be soldered both sides to act as VIA connections.

There are 2 common types of IC sockets in use today:

DIL (Dual-In-Line) Sockets

Available in various sizes from 8 to 64 pins. However their quality can vary. By far the best is the turned PIN type.

Unfortunately they're the most expensive. However they do give better electrical contact with the IC leg, which is very important for reliability.

When creating a double-sided PCB tracks need to be connected to the DIP socket from both sides of the board. The solder side connections are easy, but the component side connections get obstructed by the socket's plastic body, making soldering difficult if not impossible.

A special jig is used to extract the pins from their former. This makes the soldering of the component side pin much easier.

The JIG described in Figure 6 is for extracting the DIL IC socket pins. The IC sockets used are a 16 way turned PIN type. These are used as they represent the cheapest way of purchasing the socket pins. The JIG shown below can be constructed from a 4" vice mounted vertically.

PLCC (Plastic-Leaded-Chip-Carrier) Sockets

Available in various sizes from 20 to 84 pins. The surface mount type described here (Figure 7, Left Hand Side) are purchased with a centre section which needs to be removed by gently pressing it out with a screwdriver. The result is shown on the right-hand side of

Figure 7.

The reason for removing the PLCC socket's centre is so that VIA hole links, discussed previously, can be placed in the centre of the PLCC socket, i.e. underneath the PLCC device. The socket raises the PLCC device off the PCB enough to accommodate for the height of the via links but only when the centre section of the PLCC socket is removed.

If Plated-Through-Hole technology isn't available to you, but you need to produce double sided PCBs, then this technique described is a solution.

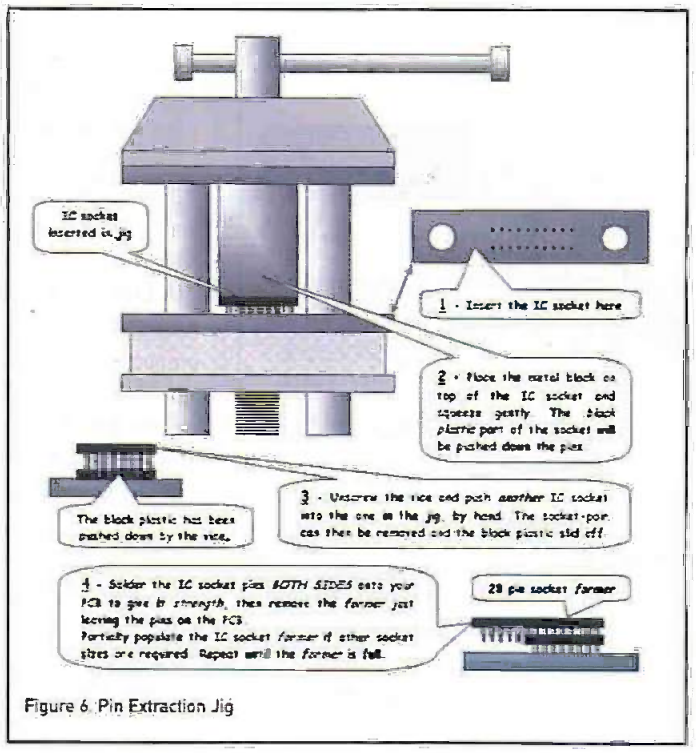

Figure 6. Pin Extraction Jig

Solder Levelling (Figure 8)

Surface mount pads need to be tinned with solder prior to mounting the component.

Commercial PCB manufacturers use a process called hot solder levelling which produces a 'flatter' solder joint. If this process is not available there is an alternative technique using solder braid to achieve the same result.

Scrub the pads with a small brush while applying IPA solvent. Apply the solder (must use the 2% silver type) to the SMT pad.

Place the solder braid on the pad near to the track connecting the pad.

Apply the soldering iron to the top of the braid resting on the pad. While the solder is molten, slowly drag the braid away from the pad in the opposite direction to the track connecting it.

Take care when dragging the solder braid. As stated previously, the pad is floating on molten glue which is holding the pad on the board. This is the reason for dragging the braid away from the track that connects to the pad. The track prevents the pad from moving out of position.

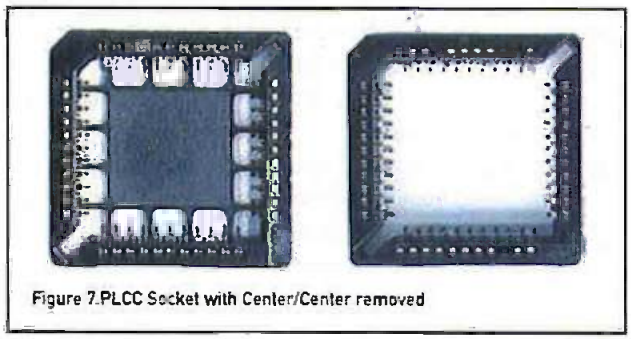
Figure 7.PLCC Socket with Center/Center removed

Fine Pitch SMTs

Components are shrinking in size all the time so the ability to be able to solder these tiny devices 'by hand' is essential.

The fact that SMT components are getting smaller is common knowledge. The problem is that the larger SMT components are becoming unavailable too!

Therefore the techniques described previously have to cope with the smaller more common components as well.

A Special Technique

Normally the device pads are soldered individually. However this is not possible with these fine pitch devices as the pitch between the device's leads are too small.

Therefore place the soldering iron between 2 adjacent leads.

Apply solder and heat as normal. Remember not to keep the heat applied for very long, particularly as the pads are so small, the glue holding the pad will melt. This will almost certainly short circuit the 2 leads together.

The excess solder can be removed using solder braid described in the solder levelling technique mentioned previously.

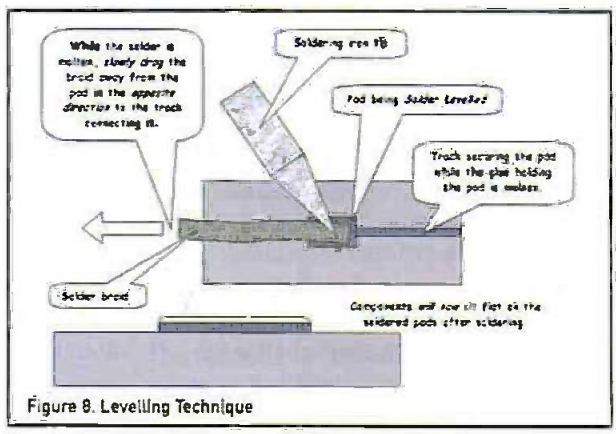

Figure 8. Levelling Technique

Handling SMTs (Figure 9)

Handling SMT devices correctly is essential to prevent both damage and contamination.

The physical size of SMT components demands the use of tweezers. However SMT components are very fragile and thus a blunt nose type is essential. Sharp tweezers can easily damage SMT resistors. The resistive film on SMT resistors for example is very thin and damage could either change the value or even make it completely open circuit!

Resistors come in various physical sizes:

- 1206 - 0.12' by 0.06' This size is the easiest to handle.
- 0603 - 0.06' by 0.03'
- 0402 - 0.04' by 0.02' These will blow away if breathed on!

The component value is printed on them in the form:

- 1K5 1500 Ohm
- 390R 390 Ohm
- 1M8 1,800,000 Ohm

Capacitors are available in the same sizes as above but have no identification marks on them at all! So DON'T MIX THEM UP!

It's important that you use the largest of the SMT components available, for ease of handling.

The pads on SMT components are pre-tinned and sealed in airtight packaging. This is to prevent the pads oxidising. They also have use-by dates similar to food stuffs. Therefore only open the packaging just before use.

The use of carousels to hold lots of unpacked SMTs is not advised for the above reasons.

All SMT components must only be touched with tweezers! Fingers will contaminate them with grease, which will be very difficult to remove and subsequently difficult to solder.

Electrostatic damage can occur when CMOS inputs are exposed to high voltages as

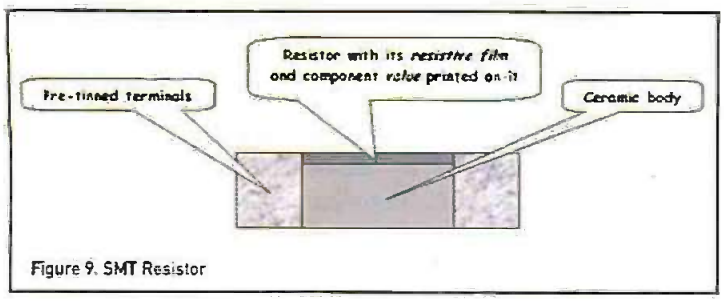

Figure 9. SMT Resistor

low as 60 Volts. However it's more likely to be KVolts.

The use of wrist straps and earth mats are essential while handling any CMOS device.

When handling devices away from the electrostatic-free area, always hold the device tightly in the palm of your hand, keeping its pins discharged through your damp skin.

Removing SMTs (Figure 10)

This is called 'rework' in the manufacturing trade, as the job has to be repeated. Reworking a job costs a company money. Therefore it's important to get things right every time by monitoring quality at all stages of assembly to reduce the necessity for rework.

Excessive heat during rework may damage the components, the PCB's pads, and de-laminate the fibre glass PCB itself – i.e. a bulge appears in the PCB due to trapped gases expanding. Therefore priorities have to be established when reworking as to which is more important, saving the component or the PCB's pads.

The pads are usually treated more importantly than the components. PCB repair is always more difficult and time consuming when compared to the cost of purchasing new components.

Removing a 2 Pinned SMT Device

Place the soldering iron tip 'flat' over the 2 component pads. Place extra solder to the tip to improve 'thermal conduction'. The 'surface tension' of the extra molten solder will 'suck' the component off the PCB and onto the tip.

Removing Multi Pinned SMT Devices

There are several options, but the cheapest are discussed below:

Using a Heat Gun

This may cause damage to the surrounding components and PCB if too much heat is applied. This can be a very effective method if the surrounding area is protected from the heat. This can be achieved by either using special high temperature selotape available or by keeping the Heat Gun in motion over the PCB area to control the temperature.

As long as care is taken to localise heating, the only problem will be that the solder on the adjacent small components may also become molten. They may even get blown away! However as long as the heat gun has a wide funnel of air, the air pressure shouldn't be enough for this to occur.

Once the solder is molten the component can be either taken off with tweezers or more easily by tapping the PCB on the bench. However beware, this may also dislodge other adjacent components that have molten solder.

Cutting The Chip's Legs Off

This method involves sacrificing the component, to save the PCB:

Cut all the legs off the component using either a fine pair of side cutters or if the legs are very small then a sharp knife can be used. Great care is required in not damaging the PCB's pads when pressing down with the knife.

Using solder wick, prepare the pads for a new component by applying the solder levelling technique described previously.

Of the 2 methods described above the Heat Gun method has an advantage that it is less damaging to the PCB's pads from the sharp knife as long as the Heat Gun is kept in motion as described above.

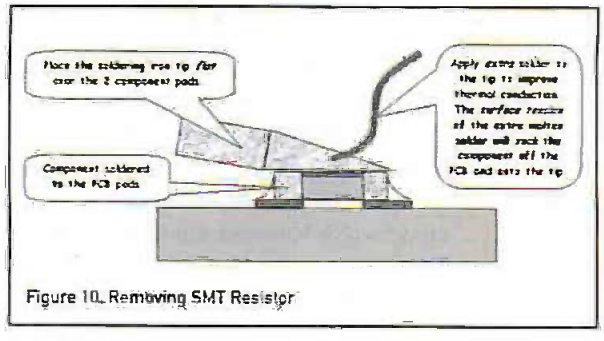

Figure 10. Removing SMT Resistor

And Finally

Another popular method of rework is to use a proprietary product called Low Melt™. This is a special low melting point solder but does require extra equipment support so isn't discussed here in detail. Further information is available from the manufacturers at www.zephyrtronics.com.

Frequently Asked Questions (FAQs)

There are always many ways of solving the same problem. Here are some Frequently Asked Questions (FAQs) from feedback emails I have received.

Question: You mention using a pointed tipped soldering iron – surely wouldn't a wide tip be better?

Answer: If your PCB has a solder mask layer then a wide tip can be used. A Solder Mask Layer will encourage the solder to fall either side of the mask so reducing bridgeing. This article is intended for prototype PCBs that don't have a Solder Mask Layer. Therefore a pointed tip is easier to solder individual SMT pads.

Observation: The fine tip has a very low thermal capacity because of its size. Therefore the temperature drops extremely quickly when it comes into contact with the pad. Once off the pad it regains its higher temperature also very quickly. The higher thermal capacity of the rest of the bit charges up the tip quickly as well. This drop in temperature has the advantage that the solder melts quickly on the pad but the temperature dropping protects the pad and its glue from overheating and causing damage.

Question: You use solder wire instead of solder paste - Why?

Answer: Solder Paste requires a lot of handling considerations: Storage in a Refrigerator; Thawing out for at least 30 minutes to room temperature; Manual application to SMD pads using applicators is messy due to its very thick consistency. Manual Soldering using solder paste can also cause solder balling where the temperature is accelerated too high that causes the solder to boil, bubble and spit its solder balls everywhere over the board with a possibility of causing short circuits. However by using Solder Wire none of these occur.

Question: Wouldn't it be better to use a hot air pencil instead of a soldering iron?

Answer: Hot Air Pencils are difficult to set up in terms of their air temperature. The air velocity controls are usually very fiddly to adjust the flow to achieve the correct temperature. If the air pressure is set too high, it can be strong enough to blow the SMT devices off their pads. You are usually heating an area of the PCB so it takes longer to melt the solder. However a soldering iron solders quickly without all the other problems mentioned.