LASERS - A FLEXIBLE TOOL FOR PCB AND DEVICE REWORK

Bob Wettermann, CIT, BEST Inc.

ABSTRACT
While there are many rework processes outlined by both rework equipment vendors and the IPC 7711/7721PCB rework and repair documents there are lack of repositories of application notes for non mainstream laser-based PCB rework processes. Outlined in this writing are a variety of applications stories which make use of lasers as PCB rework tools.

Creative rework and repair solutions for PCBs are required where there are challenges in salvaging populated PCBs. One such innovative approach involves the use of lasers, which have become commonplace in PCB and stencil fabrication and are becoming more common so in the rework area.

While novel laser rework processes cannot be described in an all encompassing "how to" applications guide, examples of their use as rework tools in the PCB and electronic device rework areas help to expand the possibilities in users minds of salvaging valuable PCBs. Several examples, including the demarking of a device, the creation of traces on an already-populated PCB and PCB marking will be described herein as case studies on the use of lasers for reworking PCBs.

COMPONENT DE-MARKING
There are several reasons why demarking or remarking of components may take place including but not limited to:

- Change of solder ball alloy type on an area array device[i]
- Protecting the intellectual property of a PCB or device design
- Marking of the date code or revision level marking for a firmware or software change
- Marking the serial number for part tracking purposes

There are several types of de-marking methods for devices including the mechanical abrasion, re-labeling, chemical etching or laser machining.

The *Micro-abrasive* removal technique[ii] uses high velocity particles to "etch away" part print markings. This method, which requires excellent dexterity, uses high-velocity abrasive particles accelerated and directed through a nozzle, as shown in Figure 1.

Figure 1-Microablasion Technique

Commonly used abrasive media include ground walnut shells, glass or plastic beads, and sodium bicarbonate powder. Care must be taken so that the abrasive media does not embed itself into soft conformal coatings such as those which are silicon-based. Masking the surrounding areas with an ESD-safe cover prevents nearby components or areas of the PCB to become damaged. Damaging static charges can be generated as high-velocity particles travel through the removal tools' delivery lines. Many micro-abrasion systems have anti-static ionizers and grounding points contained within the unit.

Although *mechanical grinding* is the least preferable method for the remarking of a component, it is the easiest procedure. A multitude of equipment rework stations containing drills, grinders, rotating brushes, and the like are commercially (Figure 2) available which the rework technician can use to grind off the markings. Care must be taken to control the depth and avoid undue stresses on device leads. The lack of control in this process means that only the most experienced and skilled operators should perform this technique. If the size of the markings is very small then the use of this technique might not be appropriate.

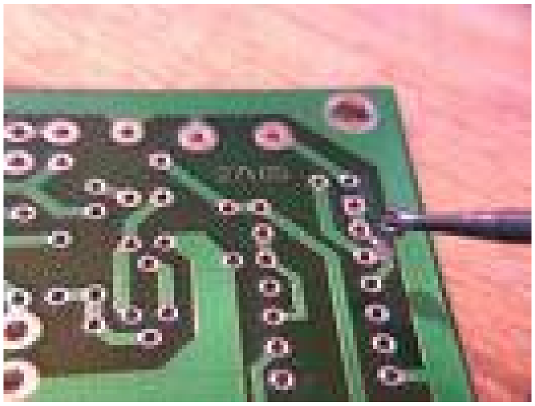

Figure 2 Mechanical Removal Tool

Chemicals can be used to remove markings from electronic components. Care must be taken to insure that the chemicals do not contain any chelating ingredients such as EDTA, phosphates, soy, caustics or chlorinated solvents. Some etchants can be overly aggressive and can remove too much of the mask or the epoxy body of the component. The parts need to be neutralized and cleaned so they do not interfere with subsequent process steps.

There are several types of *laser* sources that can be used for marking active electronic components. The most common type is near infrared (NIR) laser that produces laser energy in the 1,064 nm range of the spectral field. The most common type of NIR laser used in marking is the Nd:YAG (neodymium yttrium aluminum garnet) laser. Recently, second harmonic generation or *green lasers* have become available for this application. $C0_2$ lasers which are able to machine away any organic materials such as epoxy resin, underfills and plastics are also being employed to mark these packages.

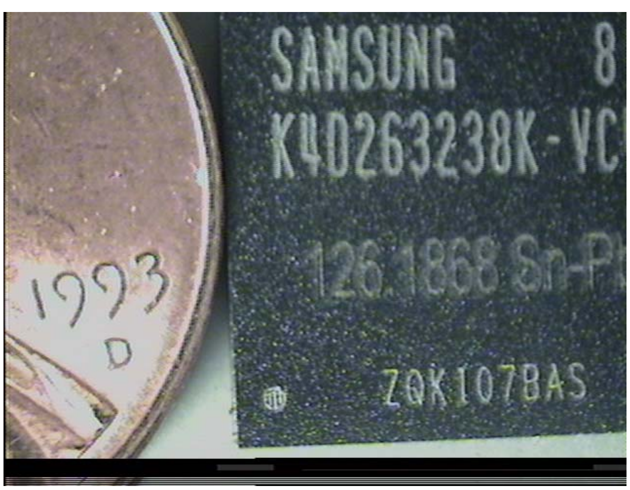

Figure 3 Lead Free Marking using laser on component

One of the main concerns during remarking (Figure 3) for manufacturers of silicon-packaged components is the potential risk of damaging the device during remarking. With silicon-based packaging, there is no plastic, metal or ceramic in the package body to protect the silicon chip from the outside. With the proper laser characteristics and optimized marking parameters, the component package can be permanently marked without incurring damage to the device.

JUMPER WIRES

Jumper wires are used to change a circuit path, such as a BGA site, for engineering changes and modifications. Jumper wires[iii] fall into the following categories:

1. Those which are considered wires and are added or installed during the assembly
2. Those that are added after the assembly to effect a change or modification in the electronic functionality
3. Those that are added to correct a defect

There are two common methods for placing jumper wires including adding or building traces and a wire re-routing method on circuit boards.

There are several issues the rework engineer needs to be aware of when using wires for the jumpers:

- When a jumper wire is added prior to part assembly, screen printing of the jumper wire is not possible.
- Stress relief must be incorporated into the jumper wire routing
- Wire cannot impede the potential for the subsequent reworking of the PCB

Conversely, traces can be built or modified right on the PCB. The benefits of using this method are the following:

- You can solder anywhere along the trace path thereby increasing the flexibility of further connections or testing
- Better aesthetic appeal
- The elimination of messy tape dots, tape strips, quick set adhesives, hot melt adhesives, hot bonding or other potential “foreign” matter on the PCB

A trace modification technique can be used if and when a jumper wire is in close proximity to another. In this method a replacement trace is added to the PCB trace layout scheme by either using a 2-part epoxy or by applying heat and pressure to a dry film-backed replacement trace. In another method (Figure 4) mask can be lased away to expose the copper effectively creating a trace which has very fine line definition. This method [1] while having engineering and set-up costs not present to near the degree of the addition of the manual trace methods, has several benefits. First, line definitions achieved by the laser are very tight with 1 mil traces and spaces well within the capability spectrum of most lasers. Second, very complex trace paths, which are very time consuming to hand fabricate when the traces are added manually, are easy to fabricate with a laser. Finally, the cost for rework is greatly reduced. The cost per piece part for rework is a fraction of the same work done manually, with the difference becoming more pronounced as the patterns get more complex and the volumes greater. In both cases these newly created traces are solderable surfaces.

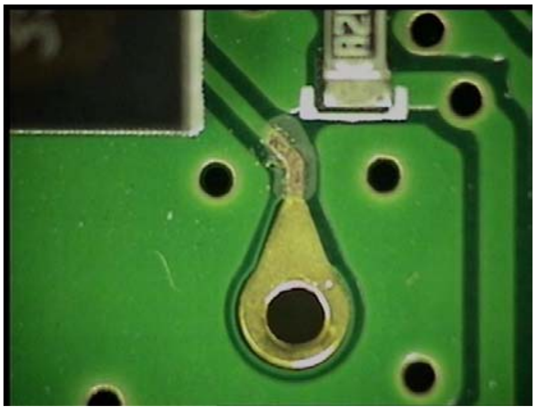

Figure 4 Laser etching of solder mask to extend a trace

PCB MARKING

There are cases where the PCB needs secondary marking as part of a rework process. In some instances a date code needs to be marked on the assembly which is more permanent or the marking needs to withstand the temperature and environmental conditions of the electronic assembly. In other cases board information needs to be permanently erased in order to protect intellectual property.

In the case of PCBs, laser marking and labeling are two of the most common marking options along with their associated attributes:

Marking Attributes	Laser	Labeling
Minimal downtime due to equipment malfunction, setup or part changeover requirements.	YES	NA
Capability to print alphanumeric text, barcodes, 2-D codes and graphic logos anywhere on a PCB	YES	YES
Invariant to PCB base material color	No	YES
Ability to accommodate multi-up PCBs without significant reduction in cycle time per panel	YES	No
Minimal Inventory of stocked items to support process	YES	No
Low cost operation per panel or board	YES	No
Supports reworking of boards at subsequent processes	YES	Sometimes

The application of a label is a common method for board identification in the PCB assembly industry. They are able to withstand rigorous manufacturing processes including soldering and cleaning. They are simple to apply and many times the ability to generate a new one already exists in the manufacturing area.

Labels have several disadvantages. One them is per board marking cost. The average cost of a 1.00 x 0.25" preprinted polyester label is $0.04 per label. Individual label costs increase as label stock requirements change from the low cost polyester label to the high cost ESD polyimide label. Labels sometimes can fall off, curl, shrink or absorb chemicals which later leach out. Fluxes can also be entrapped on labels. Their [4] thorough removal is essential for certain types including OA, RA, and RSA. Otherwise ionic contaminants will cause humidity sensitive electrical leakage currents and long term corrosion. Another side effect can be unwanted contaminants on the board. Widely used labeling standards[iv] require a minimum of 50% reflectance for spaces and in effect there is a required bar-space reflectance ratio, which some labels have a hard time reaching.

The demand for PCBs to be more completely identified can be more readily met by laser marking systems. Customers are continually requesting that each board be viewed as unique in its function and marked as an end product. Laser marking meets this demand as its "on-the-fly" machining flexibility allows each board to be uniquely marked by feeding new data into the laser machining system.

There are several advantages to utilizing laser marking as a step in the rework process. Densely populated PCBs with little or no room for product information are well-suited to laser marking as they have an ability to print very small, high aspect ratio markings. Laser marking (Figure 5) provides a permanent mark on a wide variety of materials. In addition this technique allows for a very fast writing rate which in turn allows for greater rework throughput.

Figure 5 Laser markings on stainless steel

CONCLUSION

Lasers can be used to mark, create pads or traces for jumpers.. While there is a capital cost associated with the procurement of the laser there are specific application do the proximity of neighboring parts, [5] the specialty requirements of a device or PCB marking or the precision of the light source which makes a laser a unique rework tool.

REFERENCES

[1] IPC 1066-"Marking, Symbols and Labels for Identification of Lead-Free and Other Reportable Materials in Lead-Free Assemblies, components and Devices "

[2] IPC 7721 2.3.6 "Coating Removal, Micro Blasting Method"

[3] IPC 7721 6.1 "Jumper Wires"

[5] ANSI MS 10.8-1983 and MIL STD 1189A