DEPANELING OF CIRCUIT BOARDS

Ahne Oosterhof, Eastwood Consulting, Hillsboro, OR
Ahne@Oosterhof.com
Thomas Nether, LPKF Laser & Electronics AG, Garbsen, Germany
Thomas.nether@lpkf.com

ABSTRACT
Most often circuit board manufacturers use 'size standardized' panels in production. This standardization simplifies their processes and sometimes can also benefit the assembly process when multiple boards can be processed as one item.
When producing large boards sometimes only one may fit in the panel, for small boards many can fit. But in either case at some point in the overall process individual boards have to be removed from the panel. Multiple solutions exist, like punching, V-scoring, wheel cutting, routing and lately also laser depaneling. We will be reviewing some of the impacts these methods have on the final product and determine how the newer method of laser depaneling can be advantageous.

KEYWORDS: depaneling; laser, UV, ultra-violet

DEPANELING METHODS.

PUNCHING / DIE CUTTING
This method requires a different die for each new circuit board making it not a practical solution for small production runs. The action can be either a shearing and/or crushing method, but either can leave the edges of the board somewhat deformed. To minimize damage care must be taken to maintain sharp die edges.

V-SCORING
Typically the panel is scored on both sides to a depth of about 30% of the board thickness. After assembly the boards can be manually broken out of the panel. This puts bending strain on the boards which can be damaging to some of the components, especially those close to the edge of the board.

WHEEL CUTTING / PIZZA CUTTER
An alternate method to manual breaking the web after V-scoring is to use a "pizza cutter" to cut the remaining web. This requires careful alignment between the V-score and the cutter wheels. It also induces stresses in the board which may affect some components.

SAWING
Typically machines that are used to saw boards out of a panel use a single rotating sawblade that cuts the panel from either the top or the bottom.
Each of the methods mentioned so far are limited to straight line operations, thus only for rectangular boards and each of them to some degree crush and/or cut the board edge.

WATER JET
This is a technology which some say can be done, however I have found no actual users for it yet. Cutting is done with a high-speed stream of slurry, which is water with an abrasive. I expect it will require careful cleaning after the fact to remove the abrasive portion of the slurry.

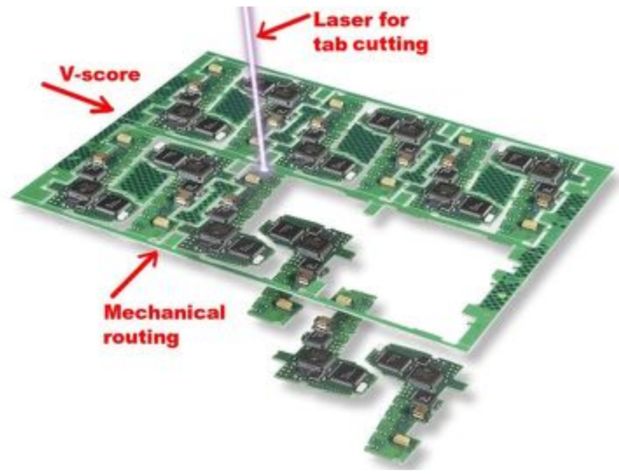

Fig 1. Combining various singulation methods in the same panel

ROUTING (+NIBBLING)
Most of the time boards are partially routed prior to assembly. The remaining attaching points are drilled with a small drill size which makes it easier to break the boards out of the panel after assembly, leaving the so-called mouse bites. A disadvantage can be a significant loss of panel area to the routing space as the kerf width typically takes up to 1.5 to 3 mm (1/16 to 1/8") plus some additional space for inaccuracies. This means that a significant amount of panel space will be needed for the routed traces.
Laser routing provides a significant advantage as the kerf width is only a few micrometers.
As an example, the small boards in Fig 2 were initially laid out expecting the panel to be routed. In this manner the panel yielded 124 boards. After designing the layout for laser depaneling the number of boards per panel increased to 368.
So for every 368 boards needed only one panel has to be produced rather than three, which is a very significant cost saving.

Routing can also reduce the stiffness of the panel to the point that a pallet may be required for support during the earlier steps in the assembly process.

But unlike the previous methods routing is not limited to cutting straight line paths only.

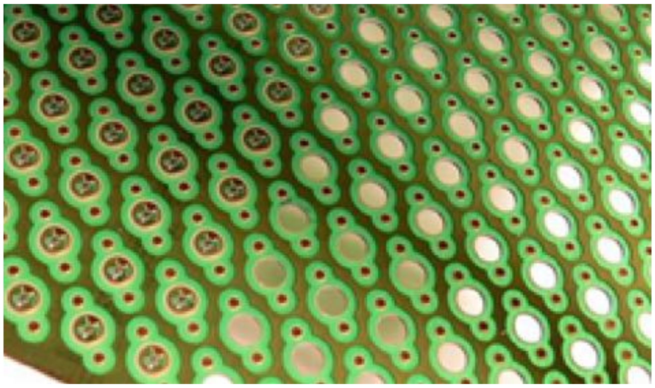

Fig 2. Example for a panel with very small circuits

Most of these methods mentioned so far exert some or a lot of mechanical stress on the board edges, which can lead to delaminating or cause a space to develop around the glass fibers. This can lead to moisture ingress which in turn can reduce the long term reliability of the circuitry.
Additionally, when finishing the placement of the components on the board and after soldering, the final connections between the boards and the panel have to be removed. Often this is accomplished by breaking these final bridges, causing some mechanical and bending stress on the boards. Again, such bending stress can be damaging to components that are placed close to the areas that need to be broken in order to remove the board from the panel.

It is therefore imperative that when doing the layout for the board and for the panel the production methods have to be taken into account, so that certain parts and traces are not placed in areas known to be subject to stress when depaneling.

Room is also required to allow for the precision (or lack of it) with which the tool path can be placed and to take into account any non-precision in the board pattern.

LASER CUTTING

The most recently added tool to delaminate flex and rigid boards is a laser. In the SMT industry several types of lasers are being employed. CO2 lasers (~10um wavelength) can provide very high power levels and cut through thick steel sheets and also through circuit boards. Neodymium-YAG lasers and fiber lasers (~1um wavelength) typically provide lower power levels at smaller beam sizes. Both these laser types produce infrared light and could be called "hot" lasers as they burn or melt the material being cut. As an aside, these are the laser types, especially the Nd-YAG lasers, which are typically used to produce the stainless steel stencils for solder paste printing.

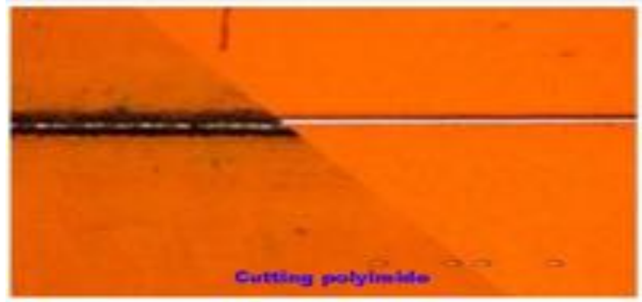

Fig 3. 'Hot' laser on the left, 'Cold' UV laser on the right

UV lasers (typical wavelength ~355 nm) on the other hand are used to ablate the material. A localized short pulse of high energy enters the top layer of the material being processed and essentially vaporizes and removes this top layer explosively turning it into dust. (See fig 3.)
The choice of a 355 nm laser is based on the compromise between performance and cost. In order for ablation to occur the laser light has to be absorbed by the materials to be cut. In the circuit board industry these are mainly FR4, glass fibers and copper. When looking at the absorption rates for these materials (Fig 4.) the shorter wavelength lasers are the most appropriate ones for the ablation process. However the cost of lasers with wavelengths shorter than 355 nm increases very rapidly.

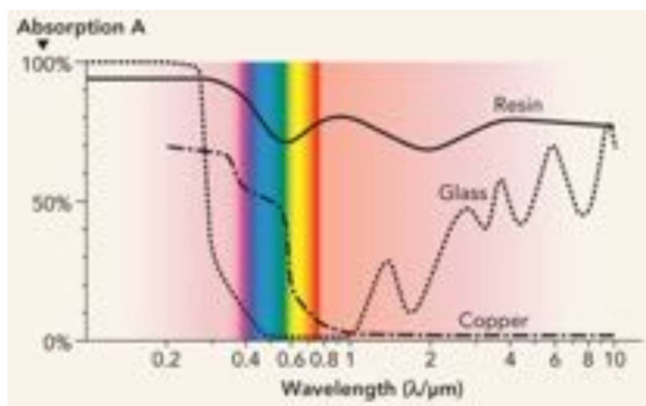

Fig 4. Energy absorption vs. wavelength

The laser beam has a tapered shape as it is focused from a relatively wide beam to an extremely narrow beam and then continuous in a reverse taper to widen again. This small area where the beam is at its most narrow is called the throat. The optimal ablation occurs when the energy density applied to the material is maximized which occurs when the throat of the beam is just inside the material being cut. By repeatedly going over the same cutting track thin layers of the material will be removed until the beam has cut all the way through.
In thicker material it may even be necessary to adjust the focus of the beam as the ablation occurs deeper into the kerf being cut into the material.

The ablation process causes some heating of the material but can be optimized to leave no burned or carbonized residue. Because the cutting is done gradually, the heating is minimized.
The earliest versions of UV laser systems had enough power to depanel flex circuit panels. Present machines have more power and can also be used to depanel circuit boards up to 1.6 mm (63 mil) in thickness.

Fig 5. Example of a laser depanleing system in an in-line setting for automatic load and unload.

Temperature
The temperature rise in the material being cut depends on the beam power, beam speed, focus, laser pulse rate and repetition rate. The repetition rate (how quickly the beam returns to the same location) is determined by the path length, the beam speed and whether a pause is added between passes.

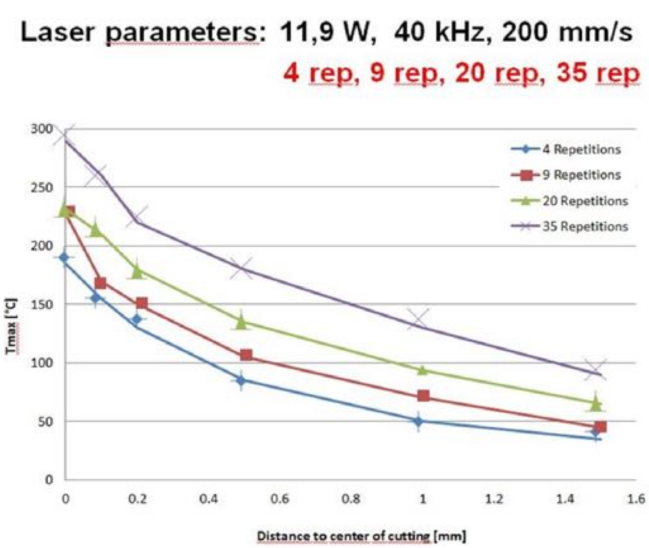

Fig 6. Heating along cutting path.

A knowledgeable and experienced system operator will be able to select the optimum combination of settings to assure a clean cut free of burn marks. There is no straight forward formula to determine machine settings as they are influenced by material type, thickness and condition. Depending on the board and its application the operator can choose fast depaneling by allowing some discoloring or even some carbonization versus a somewhat slower but completely ‘clean’ cut.
Careful testing has shown that under most conditions the temperature rise within 1.5 mm from the cutting path is less than 100 C, way below what a circuit board sees during the soldering process. (See fig 6.)

Expelled material
In the laser used for these tests an airflow goes across the panel being cut and removes most of the expelled dust into an exhaust and filtration system. (See fig 8.)
To test the impact of any remaining expelled material a slot was cut in a 4 board pattern on FR4 material with a thickness of 800 um (31.5 mil). (See fig 7.)
Only few particles stayed behind and consisted of powdery epoxy and glass particles. Their size ranged from an average of 10 um to a high of 20 µm and some may even consisted of burned or carbonized material. But as their size and their amount is extremely small there is no conduction expected between traces and components on the board. If so desired a simple cleaning process can be added and will remove the remaining particles.
Such a process step can consist of the use of any kind of wiping with a smooth dry or wet tissue, using compressed air or brushes.

One can also use any kind of cleaning liquids or cleaning bathes with or without ultrasound, but normally one would avoid any kind of additional cleaning process, especially the expensive ones.

Surface resistance
After cutting a path in these test boards (fig 7, slot in the middle of the test pattern) the boards were subjected to a climate test (40 C, RH=93%, no condensation) for 170 hours and the SIR values exceeded 10E11 Ohm indicating that no conductive material is present. See fig 9.

Cutting path location
The laser beam typically uses a galvanometer scanner (or galvo scanner) to trace the cutting path in the material over a small area, 50x50 mm (2x2 inch). Using such a scanner allows moving the beam at a very high speed along the cutting path, in the range of approx. 100 – 1000 mm/sec. This assures that the beam is only a very short time in the same location which minimizes local heating.
A pattern recognition system is employed, which can use fiducials or any other panel or board feature in order to precisely find the location where the cut has to be placed. High precision X and Y movement systems are used for large movements in combination with a galvo scanner for local movements.

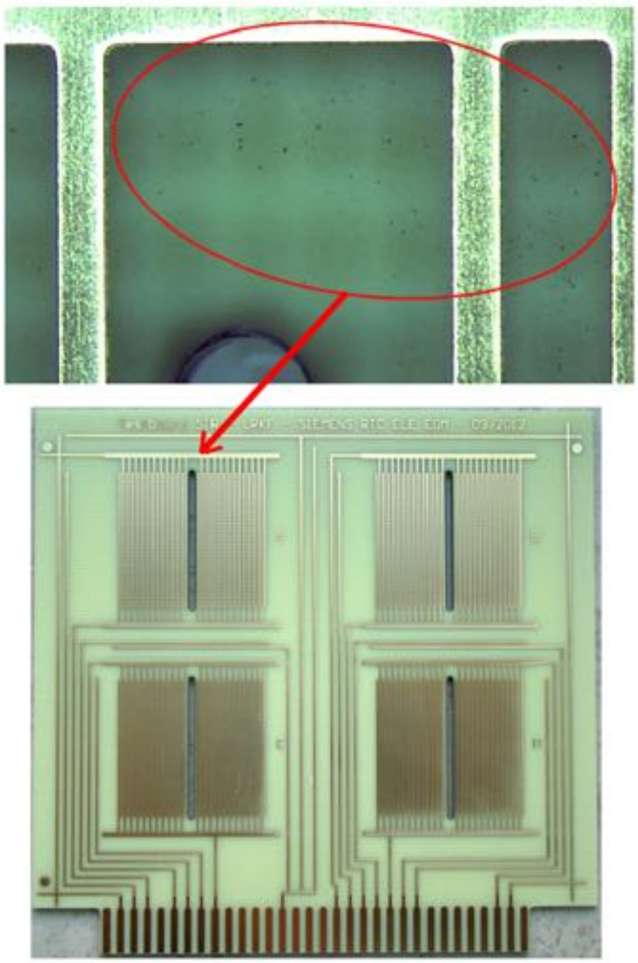

Fig 7. Surface after laser cutting.

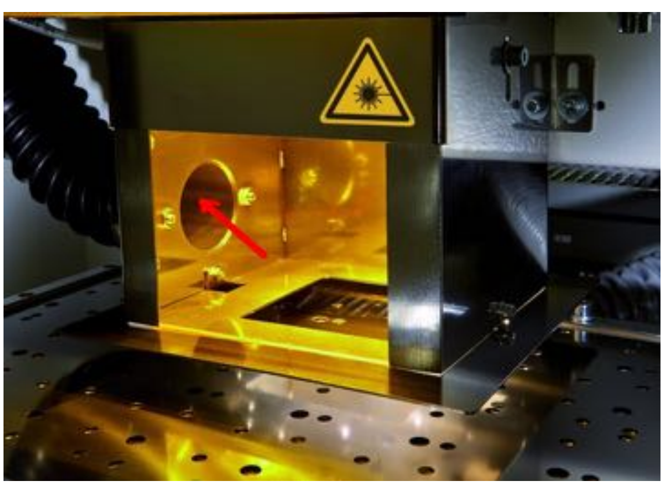

Fig 8. Removal of dust.

In these types of machines, the cutting tool is the laser beam and it has a diameter of approximately 20 um. This means the kerf cut by the laser is about 20 um wide and the laser system can locate that cut within 25 microns with respect to either panel or board fiducials or any other board feature.
The boards can be therefore be placed very close together in a panel. For a panel with many small circuit boards additional boards can therefore be placed leading to significant cost savings.

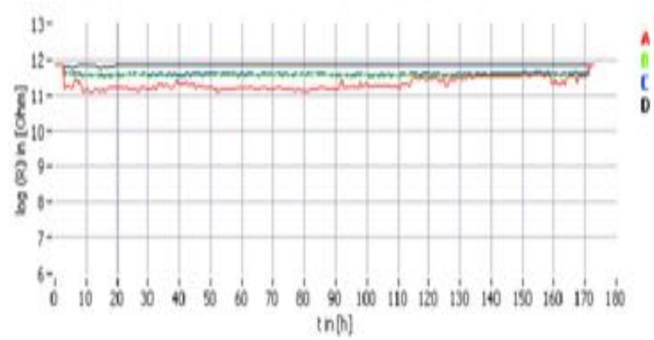

Fig 9. Surface resistance.

As the laser beam can be freely and rapidly moved in both X and Y directions, cutting out irregularly shaped boards is very simple. This contrasts with some of the other described methods which can be limited to straight line cuts. From my own experience this becomes especially advantageous with flex boards, which are often very irregularly shaped and in some instances require extremely precise cuts, for example when conductors are close together or when ZIF connectors need to be cut out. (See Fig 10.) These connectors require precise cuts on both ends of the connector fingers while the fingers are perfectly centered between the two cuts.

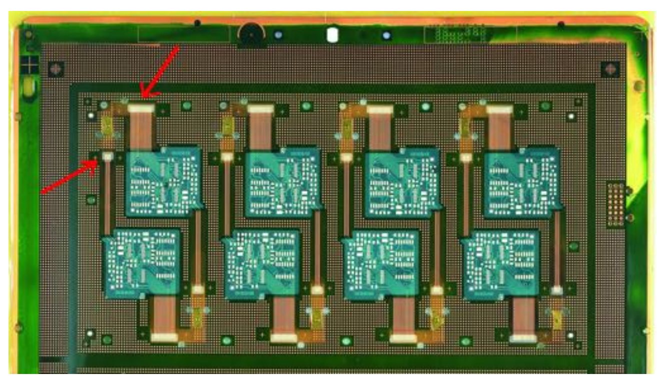

Fig 10. Flex circuit with ZIF connectors.

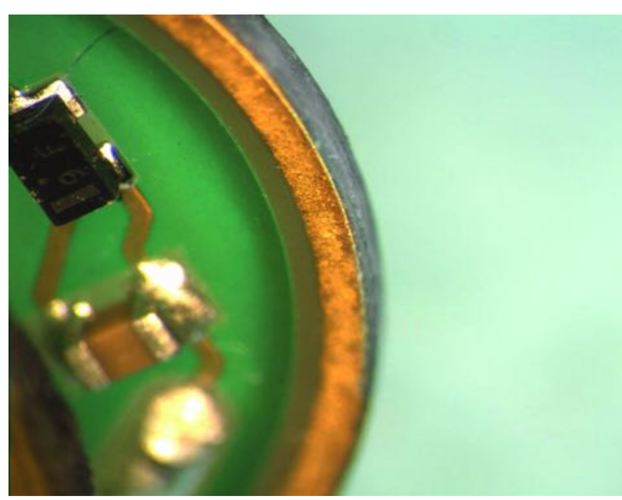

Fig 11. Singulated board from Fig 2, slightly oblique.

A potential problem to consider is the precision of the board images on the panel. I have not yet found an industry standard indicating an expectation for board image precision. The closest I have come is "as required by drawing".
This problem can be overcome by adding more than three panel fiducials and allowing the cutting operation to be divided into smaller sections with their own area fiducials. Fig 11 shows in a sample board cut out in Fig 2, that the cutline can be placed very precisely and closely around the board, in this case, next to the outside of the copper edge ring.
Even when ignoring this potential problem the minimum space between boards on the panel can be as little as the cutting kerf plus 10 to 30 um depending on the thickness of the panel plus the system accuracy of 25µm xxx

Within the area covered by the galvo scanner the beam comes straight down in the middle. Even though a large collimating lens is used, towards the edges of the area the beam has a slight angle. This means that depending on the height of the components near the cutting path, some shadowing might occur. As this is completely predictable the distance some components need to stay removed from the cutting path can be calculated. Alternatively the scan area can be reduced to side step this problem.

Stress
As there is no mechanical contact with the panel during cutting, in some instances all of the depaneling can be performed after assembly and soldering. (See fig 11.)
This means the boards become completely separated from the panel in this last process step and there is no need for any bending or pulling on the board. Therefore no stress is exerted on the board and components near the edge of the board are not being damaged.

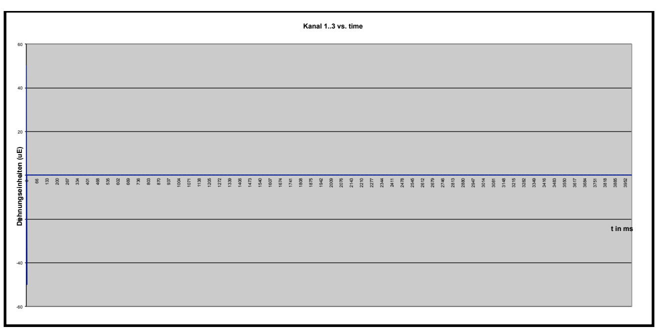

Fig 12. Stress during laser depaneling, not measurable.

Stress measurements were performed. During mechanical depaneling a significant snap is observed. (See fig 12 and 13).

It also means that during the earlier process steps, like paste printing and component placement the panel can maintain its full rigidity and no pallets are required leading to other cost savings in the process.

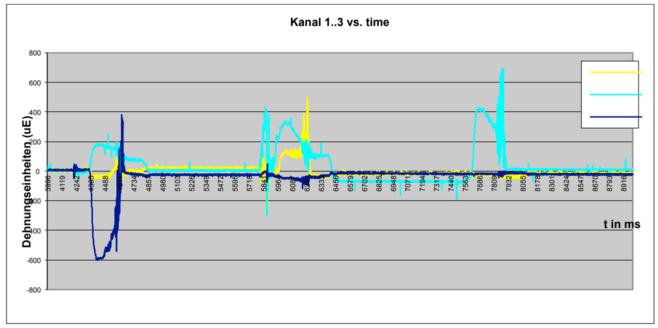

Fig 13. Stress during mechanical depaneling

Yet a common production method is to pre-rout the panel before assembly (mechanical routing with ~ 2 to 3 mm routing tool). The rigidity is then determined by the size and quantity of the breakout tabs. The final depaneling step will generate even less debris and by using this method the laser cutting time is reduced.

WALLS OF CUTTING PATH.
After many tests it has become clear that the sidewall of the cut path can be very clean and smooth regardless of the layers in the FR4 boards or polyimide flex circuits.

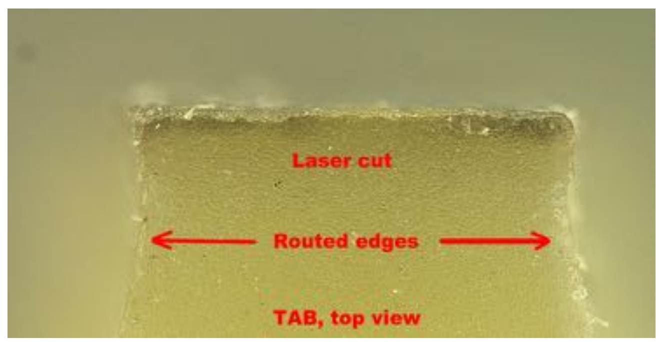

Fig 14. Laser cut tab

If the need for a clean cut is not very high, as in tab cutting of a pre-routed board, the cutting speed can be increased resulting in some discoloration as can be seen in Fig 14.

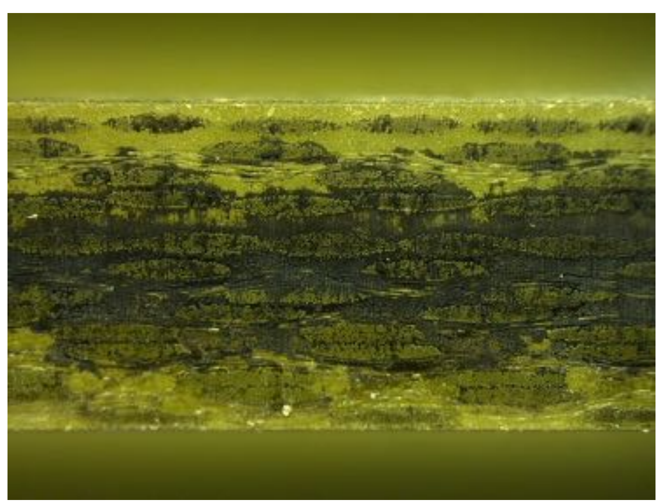
Fig 15. Laser cut edge

When cutting through epoxy and glass fibers, there are no protruding fibers or rough edges. Neither are there any gaps or delamination that would allow moisture ingress over time. (See fig 15.)

Polyimide, as used in flex circuits cuts really well and allows for extremely clean cuts, as seen in fig 3 and in the electron microscope picture in fig. 16.

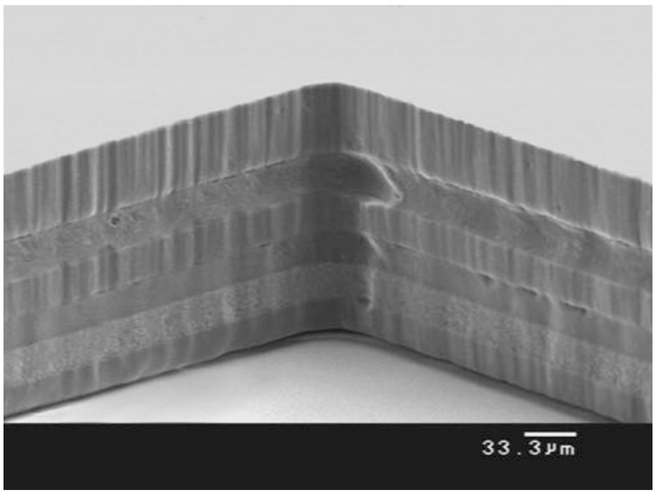

Fig 16. Electron microscope image of cutting path in multilayer polyimide.

KEEPING MATERIAL FLAT AND IN FOCUS.

As explained earlier it is necessary to keep the material to be cut by the laser as flat as possible for optimum cutting. In certain instances, as in cutting flex circuits it can be as easy as placing the flex on a downdraft honeycomb or an open cell foam plastic sheet.

For circuit boards it may be more difficult, especially for boards with components on both sides. For those instances it still may be desirable to prepare a fixture that can accommodate odd shapes and components.

CONCLUSION

From the above description it is obvious that laser depaneling can be done with very high precision. This makes it extremely useful in situations where parts of the board outline demand close tolerances. It also becomes very appropriate when very small boards are involved.

And because the cutting path is very narrow and can be located very precisely, individual boards to be placed closely together in the panel leading to economic savings.

The low thermal affects mean that even though a laser is involved, minimal temperature increases occur and therefore essentially no carbonization results.

The depaneling occurs without physical contact with the panel and without bending or pressing, therefore there is less chance of component failures or future reliability issues.

Control over the location of the cutting path is done in software, which means changes in boards can be handled very quickly.

Even though a laser depaneling system is expensive, there are sufficient reasons shown to consider using it.

REFERENCES

1. "Laser Ablation – Cutting systems for the Electronics Industry"; Ahne Oosterhof, Mark Hueske, Dieter Meier; Industrial Laser Solutions, June 2007
2. "Flex Circuit Depaneling: How UV Lasers Meet Today's Trends"; Shane Stafford ; Industrial Laser Solutions, September 2012
3. PCB Laser Technology for Rigid and Flex HDI – Via Formation, Structuring, Routing; Dr. Dieter J. Meier, Stephan H. Schmidt; IPC Printed Circuit Expo, Long Beach CA, 2002
4. "Failure Isn't an Option"; Maria Fontanazza ; Medical Device and Diagnostics Industry, March 4th 2010
5. "Laser Cutting of Printed Circuit Boards: Evaluation of possible corrosive effects of residuals using SIR-test and optical inspection", Manfred Zäske, Siemens AG, CT RTC ELE EOM-DE, June 2013

ACKNOWLEDGEMENTS

Images provided by LPKF USA and LPKF Laser & Electronics AG, Garbsen, Germany. www.lpkf.com

© Originally published in the Proceedings of the 20th Pan Pacific Microelectronics Symposium, Kauai, Hawaii, February 2-5, 2015.