PCB Laser Depanelizing Using a UV Laser

Bob Wettermann

BEST Inc.

bwet@solder.net

One of the methods gaining in popularity for singulating rigid/flex, rigid and flex circuit boards post assembly is through the use of laser routing. This method has the advantage of speed, positional accuracy, no tooling wear and lastly no induced mechanical stresses on components during the singulating process.

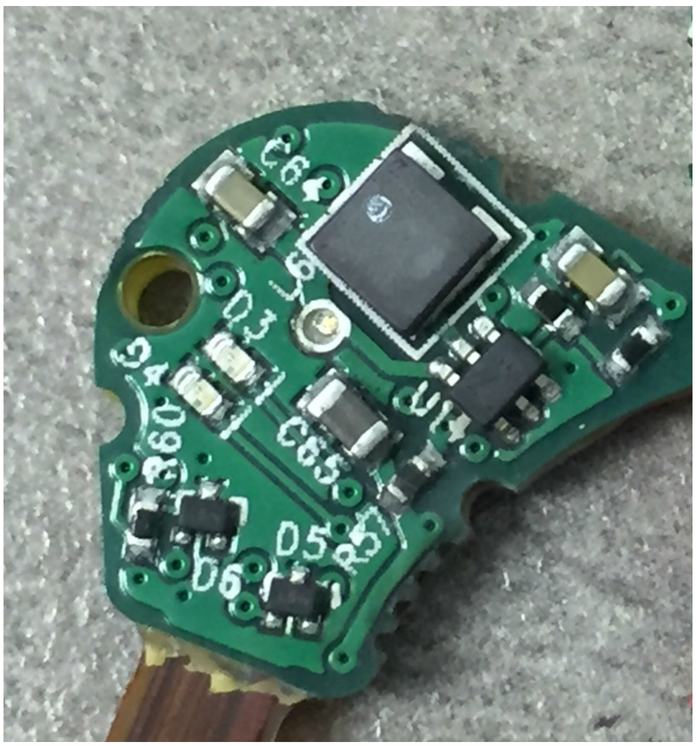

Figure 1-An IoT PCB Using Rigid-Flex Construction

There are several cases where laser routing of printed circuit boards is desirable:

- when a high degree of precision of the final assembly is required (think "tight fit" of a board in to an assembly) or
- when numerous materials need to be cut through and the number and type of depanelizing steps is to be limited (think "not compromising

cutting one material by using a technique that does not work well on a second or third material") or when

- unusual board shapes come up in a design or finally when
- highly sensitive components are placed near the cut out lines of the PCB

need to be treated gingerly.

There are a variety of depaneling options available for PCBs, each with its own set of advantages and disadvantages.

PCB Depaneling Options

The methods for straight line PCB singulating, which are set up for rectangular-shaped PCBs, all cut or crush the edge of the board edge. These methods include die cutting, punching or V-scoring the assembly or by using a wheel cutter or a saw.

The sawing method typically uses a single rotating blade spinning at high RPM to cut the panel into the shapes required. This method produces heat in the cut out area as well as creating debris as a byproduct of the cutting operation.

In V-scoring the depth of the thickness of the board is 30-40% of the original board thickness as it is cut from both sides of the board. After assembly the board is broken at this v-score line. Alternately a "pizza cutter" cuts through the V-score of the panel and cuts the remaining web until the boards are in their final cutout shape thereby putting strain on the components and solder joints-especially those near the board edge.

In another method, the singulated board outline can be punched out form the panel. This requires that a new punch be used for every single type of circuit board which means it is not a flexible method of board cut out. The punch force can also bend or deform the edges of the PCB. Sharp edges of a well-maintained die must be the norm in order to get a defect-free cutout.

In board routing and subsequent "nibbling" on a board outline can also be cut out. Boards are routed prior to assembly. The remaining attached points are drilled with a small drill thereby making it easier to break the boards out from the panel post board assembly. This leaves so-called "mouse bite" patterns behind. The routing takes up panel space as there are keep out requirements around the edge of the board cut out area. The routing can also reduce panel stiffness which is typically required and desire during the initial stages of the PCB assembly process. The advantage the routing process is that curved and non-linear line patterns can be cut using the router bit.

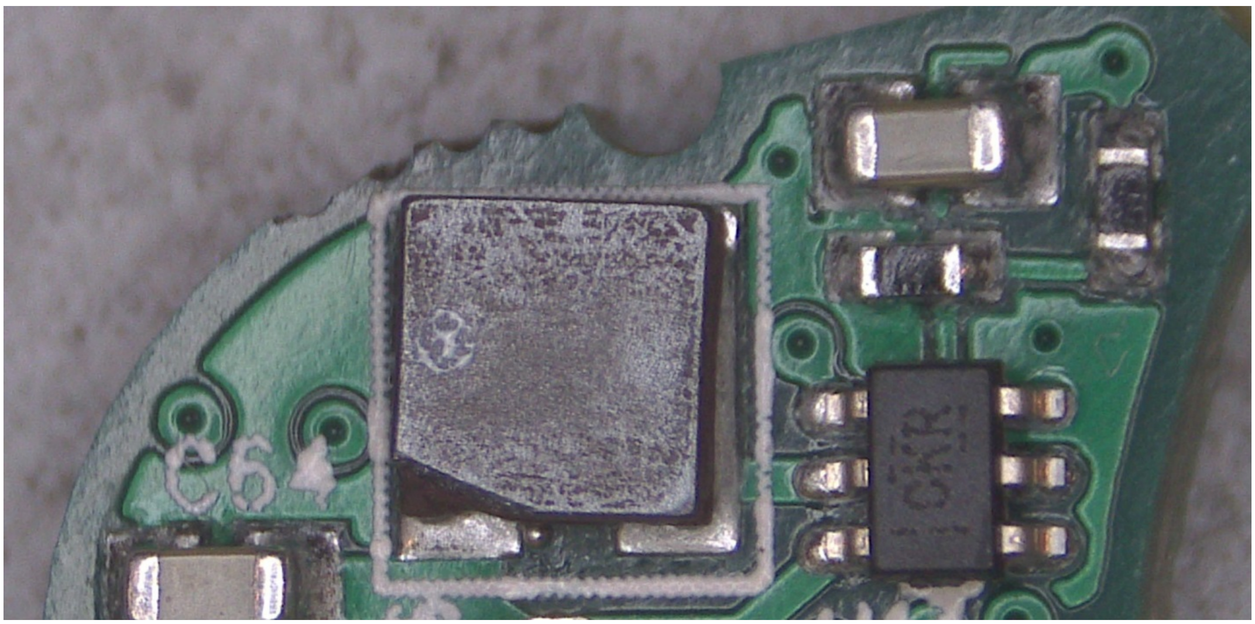

Figure 2-"Nibbling" out the board outline produces some rough edges

In each of the above techniques, all of which are mechanical in nature, the board edges, parts near the cut out area as well as the solder joints will have a high degree of stress placed on them during the cutting process. This stress may cause delamination of the board near its edges or develop space in and round the glass fibers near the board edge. Both of these anomalies may lead to moisture ingress in to the board which may lead to reliability problems down the road. These potential problems enlarge the "keep out" areas of the components along the periphery of the boards.

Laser Machining Options

There are several methods by which laser machining can be used to create the board cutout patterns including perforations, hold-in tabs and scoring. The precision of the laser cutting source allows for much smaller PCB geometries to be singulated. The precision of the laser is quite tight especially when compared to other mechanical methods. The C02 laser sources are positionally and dimensionally within 2 mils (50um) while the UV sources are within 1 mil (25 um)

Perforations

Similar to scoring or v-grooves, laser perforations are another option for tool-less PCB removal from a panel. Perforations can be laser formed to any size and spacing to meet the desired removal and securement forces.

Hold-in Tabs

Hold-in tabs are small uncut sections around the board used to secure the board in the panel. The hold-in tabs are used due to the ease of handling small parts or part securement for additional processing. The hold-in tab width is chosen based on the amount of force desired to remove the individual board from the panel/sheet or known forces to be applied by downstream processes like component loading or electro-polishing. The laser can create tabs in most any material and to any width and location about the board.

Scoring

Laser scoring produces a limited depth ablation line in the board material or materials. The depth is generally 50% of the material thickness but can be controlled to a desired depth. The scoring acts similar to the hold-tab to secure the PCB in to the panel, but allows for individual parts to be 'snapped' out. Laser scoring lines can also be used as a deliberate path for stress relief or crack propagation.

How 355nm UV lasers function

UV lasers operating at 355nm are used to ablate or to disintegrate the board material. A high energy laser pulse vaporizes and removes the top layer explosively turning it into micro dust particles. This requires that the material, whether it be FR-4, polyimide, metals or some combination thereof needs to be able to absorb the pulse. The laser beam goes back and forth over the cut through location ablating a little bit at a time. This means that because only a small amount of material at a time is vaporized, the local heating effect is minimized. In fact, measurements taken near the cutting area (within 1.5 mm of the cut area) show that the temperature rise is well below 100°C- below that of the reflow profile (1). While other shorter wavelength lasers may be more optimal in terms of performance, their use is limited by their high capital requirements.

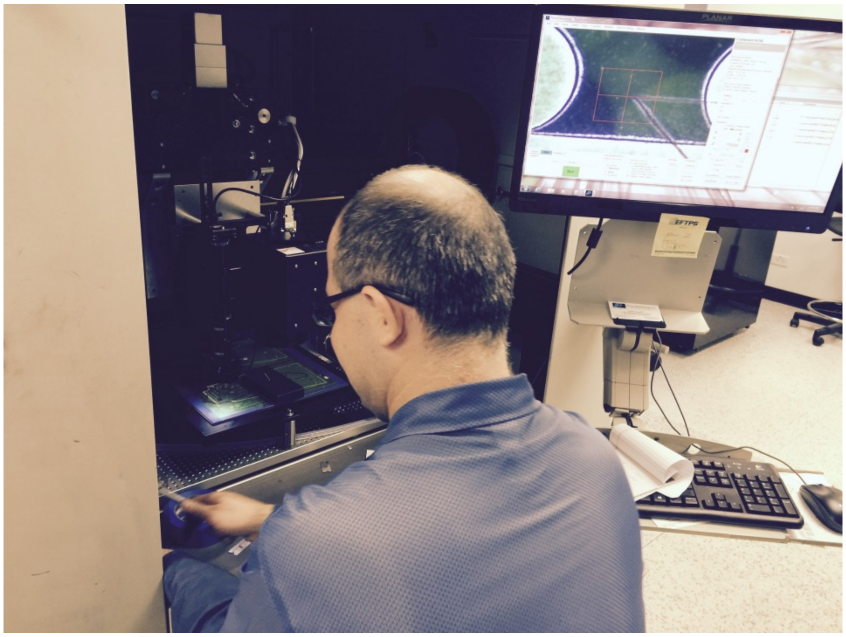

Figure 3- YAG laser system with XY micro stage controls and fiducial camera

In order to reduce the cut time and the associated expense the laser beam is "beam steered" using a galvanometer (galvo). This traces the cutting path in the material over a small area. This scanner approach arrows the beam to be moved at a very high rate of speed in the same location over a small area. This speed is in the range from 100 to 1000mm/sec. This makes sure that the beam is in the same area for only a short period of time thereby minimizing local heating affects.

The fiducials of the board can help determine the relative required location of the beam cutting location. An X-Y precision table is used for larger mechanical movements and adjustments while the galvo takes on the micro movements and adjustments of location.

Conclusion

Laser depaneling of PCBs has its application niche where boards have tight spacing, where geometric tolerances are critical and where the cutout location lines are very close to components. The method allows for precise cut out of the boards during the singulation process with very little heating of the board and very little mechanical stress compared to traditional board depaneling techniques.

Bibliography

1. Ahne Oosterhof and Thomas Nether, “Depaneling of Circuit Boards”, Circuits Assembly, 26 June 2015.

2. Jack Gilchrist ,“Laser Depaneling Technology for Medical Device Manufacturing, Medical Electronics Symposium, September 2015