11 Fundemental Design Rules for the Panelization of PCBs

Panelization Guidelines

Panelization Guidelines

By following a few key guidelines in the design of panels, significant savings can be made in the production of printed circuit boards. The following 11 recommendations ensure an effective and cost-efficient processing:

1. Size of the panel

The minimal and maximal dimensions of the panel are limited by the machines used for each process of the PCB manufacturing line. On the one hand, to enable a proper handling of the panels, at least one dimension of the panel must be larger than 50 mm. On the other hand, the maximum dimensions of the panel are limited by several machines, e.g. for the AOI or assembly of components.

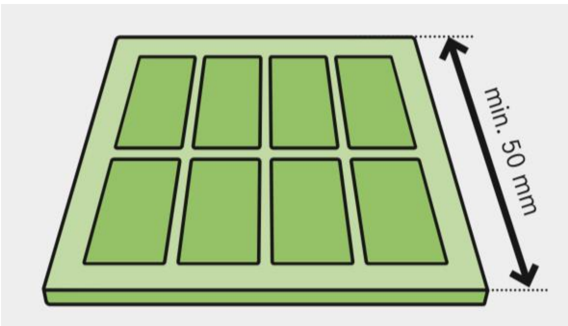

Figure 1: Minimal size of a panel

2. Form of the panel

A lot of printed circuit boards have irregular contours to meet the requirements of limited installation spaces and therefore do not have two parallel edges. To enable a proper handling by the production machines material must be added around the irregular shaped PCBs. In general, in the field of PCB production all panels have a rectangular shape.

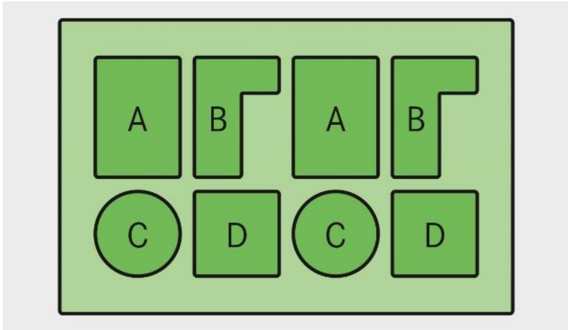

Figure 2: Typical forms of PCBs

3. Panel composition

Basically, there are two different ways of compiling the panel: heterogenous or homogenous, which means that either only the same or a predefined mixture of different PCBs is on the panel. Both methods offer various advantages. Due to its simplicity and flexibility, the homogenous composition has become widely accepted in practice.

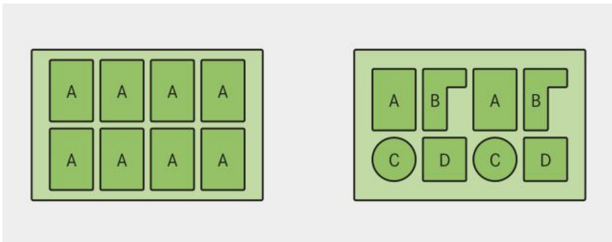

Figure 3: Schematics of panel composition

4. Handling-Margin

To ensure a practical handling of the panels a narrow strip must be left free all around at the edge of the panel. This way the transportation of the panels with conveyors, transport belts or similar gadgets is simplified.

Depending on the structure of the panel/printed circuit board the required widths vary between 12.5 mm (0.5 inch) for one-/two-sided and 25 mm (1 inch) for multi-layer boards.

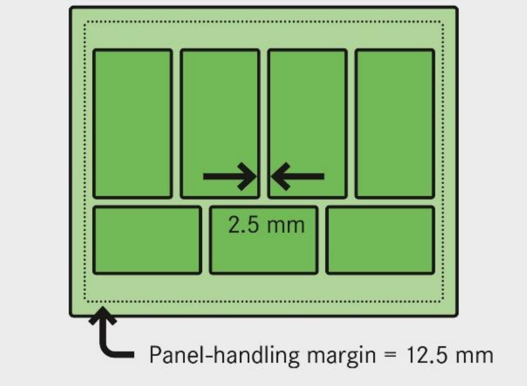

Figure 4: Margin for handling of the panel

5. Preparation of singulation

In general, there are two different ways to prepare the panel for the final singulation: tabs and v-groove. Those preventive measures protect the panel and the PCBs from damage and increase the cutting speed. In both cases specific issues must be considered.

a. Tab Design

For tabs, the contours of the circuit board are pre-milled. Predefined gaps are left in the milling paths, which are provided with individual holes at fixed intervals. Typical dimensions of milling paths, diameters and distances are shown in the following illustration.

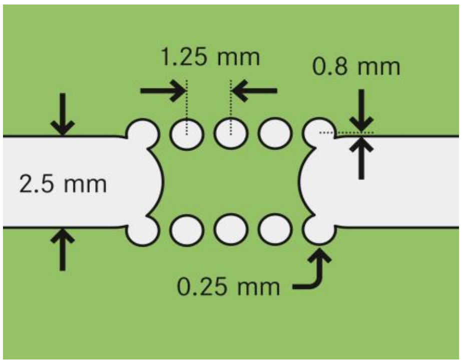

Figure 5: Typical tab design

b. V-Groove

Another approach to prepare the panel is the so-called v-groove. Thereby the panel is scored from above and below. The depth of the score is about one third of the total thickness of the panel. As with the cutting process itself, a specific distance to the components must be taken into account.

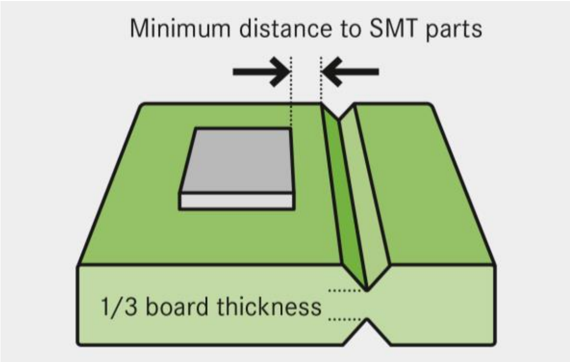

Figure 6: Schematic illustration v-groove

6. Geometries of PCBs

Printed circuit boards can have different and quite complex geometries due to the individual requirements of the assembly and the specific installation space. Mechanical separation processes like sawing or routing are limited regarding their possible contour, so that for example saws can only process straight lines and routers are limited regarding their minimum radii. The laser, on the other hand, can be used as a tool for any contours but is limited for panel thicknesses greater than 2 mm.

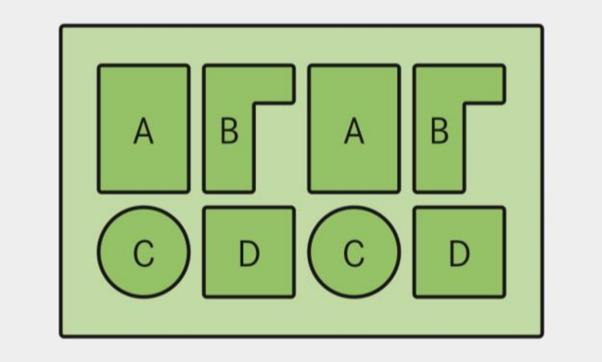

Figure 7: Different geometries of PCBs

7. Technical Cleanliness

The technical cleanliness of surfaces and cutting edges is of high importance for many industries (e.g. medical technology) and specific applications. The various panel separation processes differ significantly with regard to their possibilities in terms of the degree of purity they can achieve. Some methods produce many large particles that cannot be easily removed by suction units.

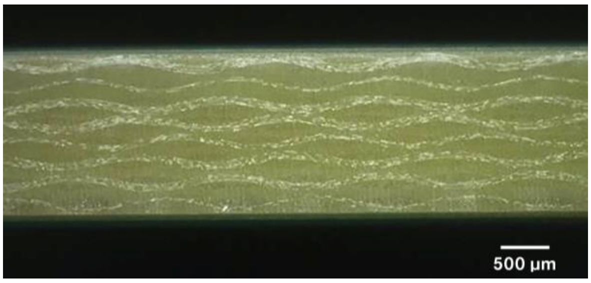

Figure 8: Technically clean cutting edge by LPKF's CleanCut-technology

8. Stress

Mechanical and thermal stress are of central importance for the yield of a SMT production line. By inducing stress, especially sensitive components like sensors can be damaged and malfunctions of the circuit be created. The generation of mechanical stress and the associated risk of component failure can be completely avoided by using a laser.

9. Mounting of PCBs/Panels

The assembly is the centerpiece of each printed circuit board and guarantees the function for the respective application. Depending on the separation method, different distances between the cutting channel and the component must be taken into account in order to avoid a collision. In case of laser depaneling only about 100 µm are necessary to avoid any thermal stress for even sensitive components. In this way the size of the circuit board can be reduced and material can be saved.

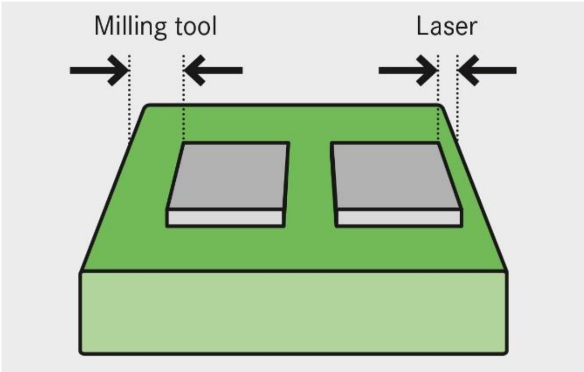

Figure 9: Distance from cutting edge to component

10. Space between PCBs

In order to be able to cleanly cut out the printed circuit boards with the respective tool, certain distances between the PCBs must be taken into account during the design phase. These vary depending on the separation process: e.g. 2-3 mm wide cutting channels are required for milling, while lasers only need a few hundred micrometers (~ 200 µm). The differences show that, depending on the size and design of the circuit board significant saving potentials can be achieved.

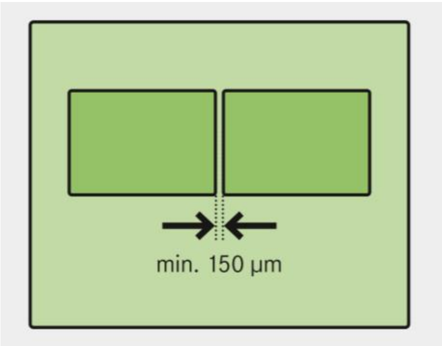

Figure 10: Space between components for laser depaneling

11. Full cut

An alternative to the two-step separation process (preparation and final singulation) is the full cut of the PCB. With mechanical processes, this is often not possible without damaging the PCBs through stress. Using the laser this is possible without any problems and enables a significantly higher utilization of the available surface area by reducing the distances between printed circuit boards. On average, savings of more than 30 percent have been achieved in this way (see figure). The producer benefits from these savings in two ways: directly through material savings and indirectly through reduced process costs of other production steps.

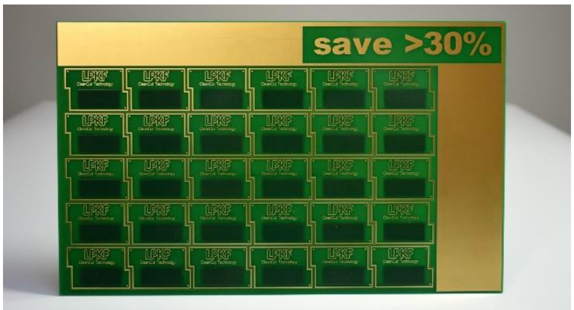

Figure 11: Potential material savings by full cut

Laser & Electronics

LPKF Laser & Electronics AG

Patrick Stockbrügger
Osteriede 7 Tel.: +49 (0) 5131 7095-1755
30827 Garbsen Mail: patrick.stockbruegger@lpkf.com
www.lpkf.com