

What is PCB Depaneling?

PCB depaneling is the process of removing numerous smaller, individual boards from a larger panel during manufacturing.

The laser process used keeps delicate components, soldered connections, and fragile substrates from any mechanical stress. Due to minimal space between the boards, there is more value per panel. In addition, components can be placed adjacent to each other to minimize unnecessary bulk and weight.

PCBs are usually manufactured in large panels with multiple boards but can also be produced as single units. The depaneling process can be fully automatic, semi-automatic, or manual. This brings lower throughput, along with eliminating the added cost of tooling and waste removal associated with mechanical methods.

Laser routing offers all of the following benefits:

TEMPERATURE:

A knowledgeable and experienced operator can select the optimum settings to guarantee a clean cut with no burn marks. Material type, thickness, and condition are all factors that will be considered and will determine the speed (and consequently will affect the temperature) of the laser.

EXPELLED MATERIAL:

An exhaust or filter system removes any expelled material during the laser process.

If the application does produce an extremely small particle residue, compressed air or smooth tissue can be used.

STRESS:

Since there is no contact with the panel during cutting, lasers allow most or all depaneling to be done after assembly and soldering. This allows for the avoidance of any bending or pulling of the board and therefore no stress is applied, and no damage occurs.

[PCB Depaneling - A-Laser Precision Laser Cutting](#)

[A-Laser Precision Laser Cutting - Laser Ablation, UV and IR Lasers](#)