

Panel Plating versus Pattern Plating

The two predominate plating processes used in the manufacture of a printed circuit boards are pattern and panel plating.

Pattern plating, also known as print, plate and etch, requires only the image of the circuit to be plated. Only those unexposed areas of the photo resist, including the plated through holes, are to be plated with copper. Tin is then plated onto those areas to serve as an etch resist to allow the removal of the unwanted copper prior to solder mask operation.

Panel plating is the reverse of pattern plating whereas it is plate, print and etch. The entire panel is plated after the drilling operation, printed with the artwork (opposite of pattern) and then etched. In this case, the photo resist serves as the etch resist and the unwanted copper is etched away and then the board is sent onto be solder masked.

Panel plating was used before pattern plating became the “norm” of our industry. However, panel plating has become more popular because its advantage is the uniformity of the electroplating with some even saying there is improved circuit definition. The disadvantages are the additional cost of the copper being plated over the entire panel and the possibility of photo resist breakdown of a tented hole causing the plating to be etched away—a void.

The advantage of pattern plating is that copper is electroplated only where it is needed. However, the major disadvantage is that depending on the current density, the copper thickness will vary considerably. For example, a ground plane will plate less than a lone trace along the edge of the printed circuit board.

Usually, it is either one or the other plating process that is used to produce a printed circuit board. However, depending on the technology and/or the size of the printed circuit board, both panel and then pattern plating may be used to manufacture the same board to ensure the proper amount of copper is in the hole and a much more evenly plated surface exists.

Research for the above information may be from, but is not limited to, IPC reference manuals, the PCB Handbook, the Bare Board PWB Design Manual and consultation with industry professionals. Please consult a process engineer familiar with your company's PCB assembly process before making any procedure changes.