

PCB Fabrication Process

A Step-by-Step Guide to Bare PCB Manufacturing

From engineering review and inner-layer imaging to lamination, hole metallization, outer-layer processing, and final inspection.

Scope

This guide describes a conventional multilayer plated through-hole construction using outer-layer pattern plating. Double-sided boards do not require multilayer inner-layer lay-up and lamination. HDI, blind or buried vias, and sequential-lamination designs may require additional or repeated operations.

Manufacturing flow

Engineering review > material preparation > inner-layer imaging and etching > inspection and bonding preparation > lay-up and lamination > drilling and metallization > outer-layer imaging and pattern plating > stripping and etching > solder mask and surface finish > marking, profiling, testing, and final inspection.

Process conditions and acceptance requirements depend on the approved production data, material system, board construction, and agreed specification. This document is an overview, not a substitute for a controlled manufacturing instruction or acceptance standard.

Source page: <https://www.pcbelec.com/pcb-fabrication-process>

Document edition: October 2026

1. Pre-Production Engineering

Engineers review the fabrication data, drill files, board outline, stack-up, materials, copper requirements, finished thickness, surface finish, tolerances, and any controlled-impedance requirements. Manufacturability questions should be resolved before production release. The output is an approved production plan and manufacturing instructions.

2. Multilayer PCB Inner-Layer Manufacturing

2.1. Material Selection and Laminate Cutting

Select copper-clad laminate that matches the approved stack-up and cut it into production panels. Panel dimensions account for equipment limits, board arrangement, tooling, and registration features.

2.2. Inner-Layer Dry-Film Lamination

Clean and prepare the copper surfaces, then apply photosensitive dry-film resist using controlled heat and pressure. Uniform coverage and adhesion support accurate imaging.

2.3. Inner-Layer Exposure

Transfer the circuit image using a phototool and UV light or direct imaging. With negative-working resist, the exposed areas harden over the copper features that must remain.

2.4. Inner-Layer Developing

Remove unexposed resist to reveal copper intended for etching. Rinse and check the developed pattern. Developing creates resist openings; it does not remove the underlying copper.

2.5. Inner-Layer Etching

Dissolve the unprotected copper while the hardened resist protects traces, pads, and planes. Control etching to clear unwanted copper while limiting conductor narrowing.

2.6. Inner-Layer Resist Stripping

Remove the remaining resist and rinse the panel. Inspect the completed copper pattern, typically by automated optical inspection (AOI), before bonding preparation and lay-up.

2. Inner-Layer Manufacturing - Continued

2.7. Multilayer PCB Lay-Up

Prepare the inner-layer bonding surfaces and arrange inspected cores, prepreg, and outer copper foil in the approved sequence. Verify layer order, materials, orientation, and alignment.

2.8. Multilayer PCB Lamination

Apply a material-specific press cycle. Heat enables prepreg resin to flow; pressure consolidates the stack; curing bonds the layers. Controlled cooling follows. This is distinct from dry-film lamination.

2.9. PCB Drilling

Use production drill data derived from approved NC drill files to form the required holes. Establish registration with the internal layers. Deburring and desmear prepare hole openings and walls for metallization.

2.10. Through-Hole Metallization

Clean and condition the non-conductive hole walls. In a conventional electroless process, activation and chemical copper deposition establish a conductive seed layer. Subsequent electrolytic plating builds the required hole-wall copper thickness.

3. Multilayer PCB Outer-Layer Manufacturing

In this pattern-plating route, dry film covers the background copper while the required circuit features remain open for copper and tin plating. This differs from the inner-layer etch-resist arrangement.

3.1. Outer-Layer Dry-Film Lamination

Apply photosensitive dry film to the prepared copper surfaces. The film will act as a temporary plating mask.

3.2. Outer-Layer Exposure

Align the production image with the drilled panel. With negative-working resist in this route, expose and harden the background areas while leaving intended plating areas unexposed.

3.3. Outer-Layer Developing

Remove unexposed film to open the required circuit features and access to plated through-holes. Check for residues and mask damage before pattern plating.

3. Outer-Layer Manufacturing - Continued

3.4. Pattern Plating: Copper and Tin

Electroplate additional copper onto exposed circuit features and hole walls. Deposit tin over the plated copper as a temporary etch resist; it is not the final board finish.

3.5. Outer-Layer Resist Stripping

Remove the dry-film plating mask to reveal the background copper. The tin remains over the required circuit features.

3.6. Outer-Layer Copper Etching

Remove background copper using an etchant compatible with the tin resist. Tin protects the plated circuit features while the spaces between them are cleared.

3.7. Tin Stripping

Remove the temporary tin coating to reveal the completed outer-layer copper pattern. Rinse and inspect the circuitry before solder mask processing.

3.8. Solder Mask Application

In a typical liquid photoimageable process, clean, coat, pre-dry, expose, develop, and finally cure the solder mask. Check registration, pad openings, coverage, adhesion, and visible defects.

3.9. Surface Finish

Protect specified exposed copper and preserve solderability. Common finish families include HASL, lead-free HASL, ENIG, OSP, immersion silver, and immersion tin. Selection depends on assembly, geometry, storage, handling, and cost requirements; availability must be confirmed for the job.

4. Marking, Profiling, Testing, and Final Inspection

Legend printing adds specified identification. Routing, V-scoring, or other suitable profiling methods form the delivery outline; boards may remain panelized. The sequence of marking, finishing, and profiling can vary with the manufacturing route.

Electrical testing checks continuity and isolation against reference circuit data. Final inspection checks appearance, dimensions, markings, solder mask, surface finish, and other agreed requirements. Electrical testing alone does not verify every aspect of reliability. Accepted boards are packaged for handling and shipment.

Preparing a PCB Fabrication Inquiry

Provide Gerber files or another agreed manufacturing data format, NC drill files, a board outline, and fabrication notes. Include the required quantity and target delivery date.

Construction and materials

State the layer count, finished thickness, laminate requirements, and approved or proposed stack-up.

Copper and interconnections

Specify copper requirements, hole types, and any special via structures. Include controlled-impedance targets and associated stack-up requirements where applicable.

Finish and delivery

Specify the surface finish, solder mask and legend requirements, tolerances, acceptance requirements, and delivery-panel arrangement.

Questions to resolve before release

Identify unresolved stack-up choices, material substitutions, special inspections, or documentation requirements so they can be reviewed before production.

Request a Bare PCB Fabrication Quote

Send your manufacturing files and requirements to PCBELEC for quotation and engineering discussion. Confirm the final production specification and lead time before order release.

Website: www.pcbelec.com

Online guide: [PCB Fabrication Process](#)