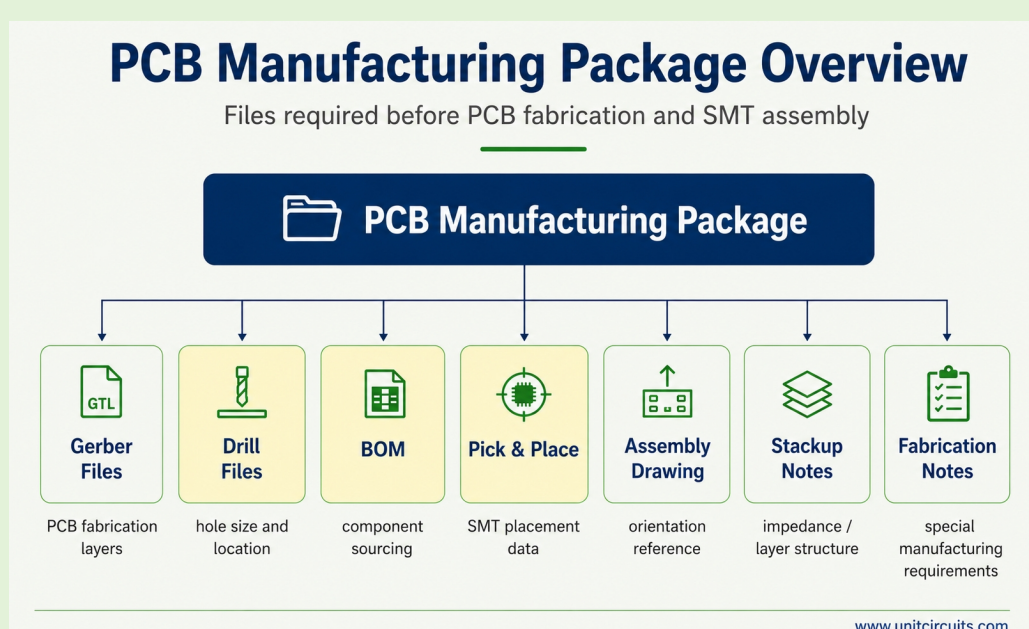


Pre-Manufacturing Gerber Validation Checklist

Essential manufacturing file checklist for PCB fabrication and SMT assembly projects.

Why Gerber Submission Errors Still Cause PCB Manufacturing Delays?

Even when Gerber files are exported successfully, manufacturing issues can still occur due to inconsistent layer naming, incorrect drill mapping, centroid rotation errors, outdated BOM revisions, or missing assembly references.



Critical PCB Manufacturing File Validation Points

Validation Area	What to Verify	Real Manufacturing Risk
Gerber Layer Naming	Consistent top/bottom identification	CAM processing confusion
Drill File Mapping	Correct tool size association	incorrect hole dimensions
Solder Mask Alignment	Aperture consistency	exposed copper defects
Pick & Place Rotation	Rotation standard validation	component polarity failures
BOM Revision Control	Latest approved revision	sourcing mismatches
Fiducial Placement	SMT optical recognition	placement instability
Stackup Documentation	Controlled impedance notes	signal integrity failures

Most Overlooked PCB Submission Errors

Common Oversight	Actual Production Consequence
Missing NPTH definition	drill interpretation errors
Inconsistent reference designators	SMT placement mismatch
Wrong centroid rotation convention	reversed IC orientation
Missing panelization notes	production delays
No controlled impedance indication	failed RF performance
Outdated assembly drawing	operator confusion

Before Sending Files to Your PCB Manufacturer

- ✓ Verify fabrication notes
- ✓ Validate drill pairing consistency
- ✓ Confirm BOM revision alignment
- ✓ Review polarity indicators
- ✓ Ensure centroid rotation accuracy
- ✓ Confirm stackup requirements
- ✓ Request supplier DFM review before mass production

Common Missing Files
Missing manufacturing files can delay PCB fabrication and SMT assembly.

- ✗

Missing Pick & Place File
SMT placement programming cannot be verified.

📋
- ✗

No Stackup Documentation
Impedance and layer requirements may be misunderstood.

📄
- ✗

Missing Assembly Drawings
Component orientation and polarity errors may occur.

📐

Manufacturing Tip | Always review Gerber, BOM, Pick & Place, and assembly drawings together before production.

www.unitcircuits.com

Recommended Manufacturing Preparation Resources

- 👉 [SMT Factory Audit Checklist](#)
- 👉 [PCB BOM Submission Guide](#)
- 👉 [PCB Stackup Design Guide](#)
- 👉 [PCB Assembly Cost Optimization Guide](#)