



## Files Needed for PCB Fabrication: A Preparation Checklist

To ensure a smooth and accurate PCB fabrication process, include the following files in your data package:

### Gerber Files (RS-274X Format)

- ☐ Top and Bottom Copper Layers (circuit traces and pads)
- ☐ Solder Mask Layers (protects copper, exposes pads)
- ☐ Silkscreen Layers (component labels, logos, reference designators)
- ☐ Paste Mask Files (for SMT stencil creation)

### NC Drill Files

- ☐ Through-hole and via locations
- ☐ Hole sizes and plating requirements
- ☐ Mechanical mounting holes or special features

### Bill of Materials (BOM)

- ☐ Manufacturer part numbers
- ☐ Reference designators (e.g., R1, C5, U3)
- ☐ Quantities and preferred suppliers

### Pick-and-Place (Centroid) Files

- ☐ X/Y coordinates of SMT parts
- ☐ Rotation and polarity information
- ☐ Top/bottom layer identification

### Assembly & Fabrication Drawings

- ☐ Board outline and mechanical dimensions
- ☐ Cutouts, slots, and fiducials
- ☐ Layer stack-up and special assembly notes

### Netlist Files

- ☐ Lists all logical connections between components
- ☐ Used to compare against Gerber data to catch errors

### IPC-356 Test File (Optional but Recommended)

- ☐ For electrical continuity testing of traces and nets
- ☐ Helps ensure reliability and integrity of the final board



Tip: Submitting a complete and well-organized file package helps prevent delays, improves manufacturability, and ensures your PCB is built to specification.

Let us know if you need help generating any of these files—we're here to support your success.