Pre-Production PCB Design Checklist

SCHEMATIC REVIEW

- All net connections verified against design specification
- No floating inputs or unconnected pins anywhere
- Power supply decoupling: 100nF per power pin pair
- ESD protection on all external-facing connectors
- Pull-up/pull-down on configuration pins (BOOT0, NRST, etc.)
- Test points on critical nets: power rails, reset, debug interface
- All component values and tolerances confirmed with BOM
- BOM cross-checked for EOL/alternative part availability

PCB LAYOUT REVIEW

- Board outline and mounting holes match mechanical drawing exactly
- Keep-out zones defined for connectors, heatsinks, and tall parts
- Trace width verified for current capacity per IPC-2221 standard
- Differential pairs routed with length matching (skew less than 5ps)
- Controlled-impedance traces per stackup calculation and fab spec
- Via types specified: through-hole, blind, buried, or microvia
- No 90-degree trace corners; use 45-degree or arc routing
- Copper pour on all unused layers, stitched to GND with via fence
- Thermal relief spokes on pads connected to large copper areas
- Sensitive analog traces kept away from high-speed digital signals

DRC VERIFICATION

- Signal-to-signal clearance: 0.15mm minimum for standard density
- High-voltage to low-voltage clearance: 2mm minimum (creepage)
- Annular ring: 0.15mm minimum on all vias and PTH pads
- Silkscreen: no overlap on pads, minimum line width 0.15mm
- Solder mask: 0.1mm expansion minimum on all pads
- Paste mask: 1:1 ratio with copper pad or per assembly specification
- Board edge to copper: 0.3mm minimum clearance

STACKUP AND FABRICATION

- Layer stackup fully specified: cores, prepregs, copper weights
- Total board thickness: 1.6mm standard (confirm if custom needed)
- Surface finish selected: ENIG / HASL-LeadFree / OSP / Immersion Silver
- Solder mask color: Green / Blue / Red / Black / White
- IPC class: Class 2 standard / Class 3 high-reliability
- Controlled impedance tolerance: plus/minus 10% standard, plus/minus 5% tight
- Minimum trace/space capability confirmed with fabricator

ASSEMBLY PREPARATION

- BOM complete with manufacturer part numbers and approved alternates
- Pick-and-place (centroid) file generated from CAD export
- Fiducial marks: minimum 3, placed near board corners
- Panelization: V-score or tab-routing with mouse-bites
- Tooling holes: 3mm diameter, minimum 3 locations
- Component orientation unambiguously marked on silkscreen
- Polarity indicators on all polarized components (diodes, caps, ICs)

DOCUMENTATION PACKAGE

- Schematic PDF, fully annotated with design notes
- Assembly drawing with component designators and outlines
- Fabrication drawing with dimensions, stackup table, and notes
- NC Drill file (Excellon format) with complete tool list
- IPC-D-356 netlist for bare-board electrical testing
- Gerber RS-274X: all copper layers, masks, paste, silkscreen, outline
- ODB++ or IPC-2581 if required by fabricator
- Readme file with revision history and change notes