



Design for Manufacturing, Assembly & Testing

Top DFMAT Considerations for Your Next PCB Design

Layout & Fabrication Best Practices

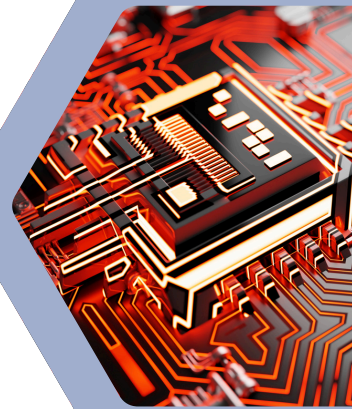
- Use standard board stackups & FR-4 material as possible; Specify dielectric thickness for impedance control.
- Utilize standard drill sizes and avoid unique microvias; Limit small drills (<0.3 mm) to protect yield and cost.
- Keep trace widths and clearances conservative ($\geq 6-8$ mil standard). BENCOR can support 3/3.
- Prefer plated-through vias for signals; Use blind or buried vias only when necessary.
- Place thermal reliefs on thru-hole pads tied to large planes to simplify soldering & reflow.
- Minimize via-in-pad unless via filling and planarization are planned.
- Use multiple small apertures for BGAs to reduce solder doming.
- Plan stencil apertures with standard paste ratios.
- Separate high-speed/analog & noisy digital nets.
- Implement controlled impedance traces & solid return planes.
- Maintain sufficient annular ring ($\geq 0.15-0.2$ mm) and soldermask expansion; Avoid exposed copper near edges.
- Group high-power components & provide thermal reliefs, heatsinking, thick copper, or thermal vias for good dissipation.

Assembly & Test Optimization

- Allow adequate pad & courtyard clearances for reliable placement, inspection, & rework; Clearly mark polarity & Pin 1.
- Reserve keep-out areas for tooling holes, connectors, & mounting hardware w/ good edge clearance ($\geq 0.5-1.0$ mm).
- Include accessible test points & group testable nets on one side as possible for efficient ICT & functional testing.
- Provide fiducials (min. 3 for odd-shaped boards or single-sided arrays) & clear placement references.
- Design for consistent component orientation to simplify automated pick-and-place & reduce errors.
- Maintain minimum component-to-component spacing compatible with pick-and-place nozzles.
- Use IPC-compliant pad shapes and toe fillets for QFPs, BGAs, and fine-pitch components.
- Utilize standard reflow profiles; Avoid mixed solder alloys unless required.

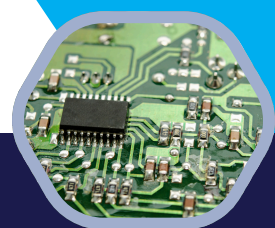
Process & Documentation Excellence

- Clearly label all parts in the BOM and assembly drawings (reference designators, polarity, orientation, manufacturer, and manufacturer part numbers).
- Explicitly call out surface finish (e.g., HASL, ENIG).
- Include placement drawings and centroid (pick-and-place) files.
- Consider panelization early: include routing tabs, V-scores, tooling holes, and array symmetry.
- Ideally, BENCOR will determine final panelization for best results.



Why Partner with BENCOR for DFMAT?

Early collaboration
with our team eliminates
costly revisions,
accelerates your time to
market, and delivers
higher first-pass yields.
— providing expert,
practical guidance
tailored to real-world
production.



Ready to Strengthen Your Next PCB Design?



Contact us today for augmented engineering services.

1-800-836-8268

Sales@BENCOR-LLC.com

ISO 9001:2015 • ITAR Registered
BENCOR, LLC • Brenham, Texas • www.bencor-llc.com