



## Design for Manufacturability (DFM) General Guidelines

Listed below are the general guidelines for design engineers to review, prior to releasing a new design to Distron:

- Circuit board fiducials are required on three corners on both sides of the board. Solid, round, copper fiducials work best. Fiducials are also required on the panel frame. These fiducials need to be in .200" from board edge.
- V-scored rails are preferred. Maximum board/panel size is 15.5" x18.75". Rail width should be approximately .250". If connectors or components overhang the board edge, the rail width should be slightly wider so that the panel can be assembled on conveyORIZED equipment. For routed areas, the opening should be .100" wide x .450" in length to allow for a nibbler tool to cut each board out of the panel. Components should be placed at least .200" from the board edge.
- SMT and PTH components should be spaced properly and be spaced properly from board edges.
- Distron prefers ENIG (Electroless Nickel/Immersion Gold) as the surface finish for new designs.
- Remove solder mask between pads of passive components where there are no traces: this will reduce risk of tombstone defects and flux entrapment.
- Minimize the number of through hole components to improve assembly efficiency and reduce labor costs. Ensure bottom side clearance between through hole lead and SMT devices for selective or wave soldering; preferred clearance is approximately .075".
- Unused vias should be tented with solder mask to reduce risk of solder bridging.
- If wave soldering is required, use un-plated holes on the bottom side of the board if possible to reduce masking requirements

These are other considerations to be discussed in the design review phase, and prior to quoting/manufacturing:

- A complete bill of material (BOM) including approved vendor information with valid part numbers and reference designators for each component. If possible, please provide alternate approved vendors to avoid single sourced items and ensure components are not obsolete.
- A drill drawing and gerber files are required including board stack-up and PCB material.
- Specify the soldering chemistry preference (Lead-free or Tin Lead, water soluble or No Clean)
- Provide an overall test strategy.
- Detail final packaging requirements.