Board Characteristics:

- All dimensions are given in inches unless specified otherwise.
- Material FR4 with Tg>170°C, e.g. FR4066.
- Minimum trace width and clearance: 0.0005" on Signal 1, 2, 3, 10 (Top and Bottom).
- Minimum trace width and clearance: 0.0005" on Signal 2, 3, 10 (all stripline traces).
- 0.008 oz copper for all power layers and for Signal 1, 2, 3, 10 (Top and Bottom).
- 0.008 oz copper for Stripline traces layers (Signal 1, 2, 3, 4, 5).
- Immersion Gold over copper, with Min. Ni: 2.5-5 µm; Au: 0.05-0.2 µm.
- Apply solder mask over bare copper.
- Board Thickness: 0.093 +/- 0.008.
- Solder mask on Component and Solder Sides.
- 45 degree chamfer.
- FMS tolerances: +/- 0.002 unless specified otherwise.
- Underlayer spacing as specified.
- This is a pressfit technology thru hole with the following specs:
  1. Finished plating hole size: 0.039 ± 0.025 mm.
  2. Drill hole size: 0.754 +/- 0.025 mm.
  3. Min. Thickness of thru hole plating: Cu: 25 µm; Ni: 2.5-5 µm; Au: 0.05-0.2 µm.
  4. Plated via holes for 0.005" stripline and 0.003" microstrip traces on all layers.
  5. Perform TDR test for all signal layers, using existing test traces on board.
  6. Present TDR test results for all signal layers.

UNIVERSITY OF CHICAGO ELECTRONICS DEVELOPMENT GROUP

TDC Board Specification Drawing

M. Bogdon graphics

M. Bogdon graphics