Board Characteristics:
0. All dimensions are given in inches unless specified otherwise.
1. Material FR4 with Tg>170°C (e.g., FR408)
2. Minimum trace width: 0.008" and clearance: 0.005" on Signal 1, 3, 5, 6 (Top and Bottom);
3. Minimum trace width and clearance: 0.008" on Signal 2, 3, 5, 7, 8, 9, 10 (all striplines);
4. 1 oz copper for all power layers and for Signal 1, 2 (Top and Bottom)
   1/2 oz copper for stripline trace layers (Signal 2, 3, 4, 5, 7, 8, 9, 10).
5. Immersion Gold over copper, with min. Ni: 2.5-5 μm; Au: 0.05-0.2 μm.
   Apply Solder Mask over bare copper.
6. Board Thickness: 0.032" ± 0.008".
7. Will the Top and Bottom of board on the solder side to a thickness of 0.063" ± 0.008
   0.060" on Component and Solder Sides.
8. 45 degree chamfer.
9. 0.008" tolerances: ± 0.002" unless specified otherwise.
10. Layer spacing as specified.
11. Test layer spacing as specified.
12. Use 5 Ohm ± 5% for 0.005" stripline and 0.006" microstrip traces on all layers.
   Perform TDR test for all signal layers.
   Present TDR test results for all signal layers.