

Zephyr Engineering Inc. Fabrication Drawing Notes

PCB Name: Clock Fanout Board
PCB Rev.: A
PCB P/N: BTI-13062
Date: 4/12/2013

NOTES: - UNLESS OTHERWISE SPECIFIED

1. Material: Epoxy fiberglass **FR-4** or equivalent, **0.062"** +/- 10% finished thickness.
2. Finished copper thickness to be **1.0 oz.** all layers.
3. Minimum copper plating 0.001" thick for plated through holes. Annular ring to be 0.002" minimum.
4. **SMOBG** – Solder mask over bare gold both sides, L.P.I., **green** in color.
5. Solder mask registration +/- 0.003". All traces adjacent to pads shall be completely covered with solder mask.
6. **ENIG** – Surface finish to be Electroless-Nickel Immersion Gold prior to solder mask.
7. Primary Silkscreen 1 shall be **white** epoxy ink.
8. Primary Silkscreen 2 & Secondary Silkscreen shall be **black** epoxy ink.
9. Hole diameter tolerance is +/- 0.003" after plating.
10. Hole location +/- 0.003". Maximum layer-to-layer mis-registration shall be 0.003".
11. Finished conductor width shall not vary more than +/- 10% from artwork master.
12. Warp and twist of board shall not exceed 0.005 inch per inch.
13. All dimensions are in inches. Tolerances: .XX = +/- 0.010", .XXX = +/- 0.005".
14. Acceptability requirements per IPC-A-600D, Class 2.
15. Vendors UL approved logo to be located on solder or secondary side of board.
16. Drawing is viewed from primary side.
17. This is a 4-layer board.
18. Drill size units are in inches.
19. Do not trim silkscreen that flows over via holes.
20. Use Gerber generated data for board outline and cutouts.
21. Compare IPC-D-356 netlist with Gerber generated netlist prior to any workmanship.
22. Differential pair specifications:

Value (Ohms)	Trace Width / Spacing (mils)	Tolerance (Percent)	Found on Layer(s)
90	5.5 / 7.5	+/- 10	1
100	4.5 / 8.5	+/- 10	1