

LAYER-STACK_	Sym	No	Mils	MM	Qty	Plated
01-16 01-20	+	1	12	0.30	11	YES
	×	2	18	0.46	2	YES
CONTROL CONTRO		3	25	0.64	2	NOT
	♦	4	39	1.00	34	YES
	X	5	40	1.02	37	YES
	M	6	102	2.60	4	NOT
	4	7	125	3.18	2	NOT

- 1. This drawing is CAD generated. No manual changes authorized after release.
- 2. All printed wiring boards to be manufactured in accordance with IPC-6011 and IPC-6012. In case of conflict this drawing governs all other specifications.
- 3. Material, copper sheet laminate, copper clad FR4 170Tg/290Td. Interior layers (if applicable) to be 35um copper. Exterior layers to be 70um copper finished. Final thickness to be ~1.60mm.
- 4. All product manufactured to this drawing shall be RoHS compliant.
- 5. All holes are plated through holes unless noted otherwise. Minimum plated through holes to have 25um thick walls.
- 6. All finished trace widths to be +/- 20% measured at the bottom, minimum finished spacing to be 100um.
- 7. All exposed copper to be finished with ENIG solder finish over clean bare copper.
- 8. Bow and twist shall not exceed 0.50% measured diagonally, per IPC-TM-650.
- 9. Drill boards using drill data, drill patterns, and drill table provided. Hole locations shall be 0.075mm (radial error) about true position. All finished holes to be +/-0.075mm diameter unless noted otherwise. All finished vias to be +0.075mm to closed diameter.
- 10. Minimum annular ring to be 50um external layers and 25um internal layers (if applicable).
- 11. Layer to layer misalignment shall not exceed 75um (radial error) about true position.
- 12. Solder mask construction to be SMOBC using green LPI solder mask material. Solder mask to be 18 ~ 50um thick after curing, both sides.
- 13. Solder mask misalignment shall not exceed 0.075mm. Solder mask may overlap through hole solder pads by 0.025mm maximum, but shall not overlap SMT pads.
- 14. Silkscreen should be applied over solder mask using a white epoxy-based ink. The silkscreen should not overlap solder pads unless noted otherwise.
- 15. Dimensions shown reflect the board size after plating.
- 16. The circuit board manufacturer shall apply a name and date code on the bottom side of the board in copper etch.