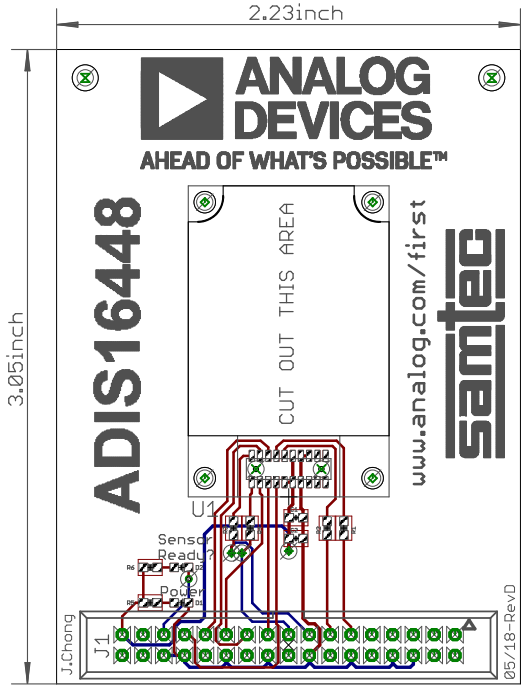

1. This drawing is CAD generated. No manual changes authorized after release.
2. All printed wiring boards to be manufactured in accordance with IPC-610 Class 2. In case of conflict this drawing governs all other specifications.
3. Material, copper sheet laminate, copper clad FR4 170Tg. Interior layers (if applicable) to be 0.5oz copper. Exterior layers to be 1oz copper finished. Final thickness to be 0.063" (1.6mm).
4. All holes are plated through holes unless noted otherwise. Minimum plated through holes to have 25um thick walls.
5. All finished trace widths to be +/- 20% measured at the bottom, minimum finished spacing to be 100um.
6. All exposed copper to be finished with ENIG Solder finish over clean bare copper.
7. Drill boards using drill data, drill patterns, and drill table provided. Hole locations shall be 0.003" (0.075mm) about true position (radial error). All finished holes to be +/-0.003" (0.075mm) diameter unless noted otherwise. All finished vias to be +/-0.003" (0.075mm) to closed diameter.
8. Minimum annular ring to be 50um external layers and 25um internal layers (if applicable).
9. Layer to layer misalignment shall not exceed 75um (radial error) about true position.
10. Solder mask construction to be SMOBC using green LPI solder mask material. Solder mask to be 18 ~ 50um thick after curing, both sides.
11. Solder mask misalignment shall not exceed 0.003" (0.075mm). Solder mask may overlap through hole solder pads by 0.001" (0.025mm) maximum, but shall not overlap SMT pads.
12. Silkscreen should be applied over solder mask using a white epoxy-based ink. The silkscreen should not overlap solder pads unless noted otherwise.
13. Dimensions shown reflect the board size after plating.
14. The circuit board manufacturer shall apply a name and date code on the bottom side of the board in copper etch.

LAYER-STACK
01-16

Sym	N°	Mils	MM	Qty	Plated
+	1	18	0.46	3	YES
×	2	25	0.64	2	NOT
⊡	3	39	1.00	34	YES
◈	4	102	2.60	4	NOT
⊠	5	125	3.18	2	NOT