

TPC-iTPC_PP-PCB-D (board A) fabrication notes:

revD 5/24/2016

1. Finished trace and line widths should match the supplied gerber files within +/- 0.001"
2. Material: must be Halogen/Bromide Free
3. Copper Weight: Inner layers : 1oz, Outer layers : 1oz
4. Solder Mask : none
5. Overall board thickness : 0.093" +/- 10%
6. Hole Diameter tolerances : +/- 0.003"
7. Hole location tolerances: +/- 0.002"
8. Electrical test to gerber data
9. Silk Screen : none
10. Minimum annular ring : 0.003"
11. Plane Registration : +/- 0.003" to top plane for all layers
12. Warp and Twist not to exceed 0.007" per inch across the diagonal of the PCB
13. Acceptability per latest revision of IPC-A-600, class 2
14. Quantity : 3 pieces
15. Total # of holes : 10,503 (includes blind/buried vias)
16. Min line : 0.005", Min space : 0.005", Tolerance: +/- 0.0002"
17. SMT Pads, Top : 4290, Bottom : 0, min pitch : 1.0mm
18. Board finish : Tin plating on top only, bottom should be bare copper
19. Turn time 12-15 days
20. No modifications permitted to copper layers, except upon explicit review and approval.

Size	Qty.	Plated	Thru/Partial	Tolerance
0.010"	3440	Yes	P1-2	+/- 0.003"
0.010"	3440	Yes	P2-3	+/- 0.003"
0.016"	3440	Yes	P3-4	+/- 0.003"
0.022"	71	Yes	Thru	+/- 0.003"
0.041"	110	No	*P1-3	+/- 0.003"
0.125"	2	No	Thru	+/- 0.003"

*Only need to be a depth of 0.055", don't have to reach L3

TPC-iTPC_INS-PCB-C(board B) fabrication notes:

1. Material: must be Halogen/Bromide Free
2. Board: 2 layer, 0.031" thickness, no copper on either side.
3. Quantity: 3 pieces
4. Turn time 12-15 days

TPC-iTPC_PP-ASY-A assembly notes:

1. Laminate boards A and B together using low flow prepreg (halogen/bromide free)
2. Overall finished thickness shall be: 0.125" +/- 10%
3. Board B shall be laminated to the Layer 1 side of board A.