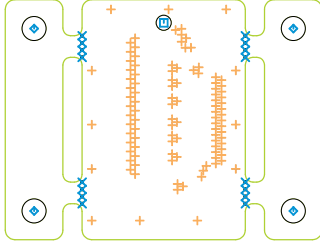


TOP SIDE



LAYER-STACK

SILKSCREEN TOP
SOLDER MASK TOP

LAYER 1 - TOP GROUND (1 OZ)
PREPREG: DIELECTRIC (8 MIL)
LAYER 2 - SIGNAL PLANE (1 OZ)
CORE: DIELECTRIC (FILLER)
LAYER 3 - POWER PLANE (1 OZ)
PREPREG: DIELECTRIC (8 MIL)
LAYER 4 - BOTTOM GROUND (1 OZ)

SOLDER MASK BOTTOM
SILKSCREEN BOTTOM

TOTAL BOARD THICKNESS: 62 MIL

DRILL CHART (ALL UNITS IN MILS)

Sym	Mils	Tol	Plated	Qty
+	10	0.02	YES	108
x	15	0.03	NOT	16
□	79	0.03	NOT	1
◇	125	0.03	NOT	4

OK WITH DEFAULT TOLERANCES

NOTES: UNLESS OTHERWISE SPECIFIED:

- PCB TO BE FABRICATED PER IPC GUIDELINES:
IPC-A-6012B OR LATEST REVISION, IPC-D-275 AND IPC-RB-276 CLASS 2 OR LATEST REVISION.
- FINISHED BOARD TO BE 0.062IN +/- 10% THICK.
MATERIAL: ITEQ IT-180A, OR 370HR FR4, OR EQUIVALENT.
4 LAYERS, LAYERS 1 AND 4 TO HAVE 1 OZ. MIN COPPER CLAD BEFORE PLATING,
AND PLATE PER IPC-6012.
COPPER PLATE ON WALL OF HOLES SHALL BE 0.001 AVERAGE, 0.0008 ABSOLUTE MINIMUM
- MINIMUM ANNULAR RING IS 0.002 FOR COMPONENT HOLES, TANGENCY FOR VIAS.
- MASTER ARTWORK (FILM/GERBER DATA) IS NOT ETCH COMPENSATED.
COMPENSATE WORKING FILM SUCH THAT TRACE WIDTHS
MATCH TRACE WIDTHS OF MASTER ARTWORK WITHIN +/- 20% OF TRACE WIDTH
- USE GREEN LIQUID PHOTO IMAGEABLE SOLDER MASK
CONFORMING TO IPC-SM-840C CLASS H.
BOARD TO HAVE SOLDER MASK OVER BARE COPPER ON BOTH SIDES.
NO SOLDER MASK TO PAD PERMISSIBLE, EXCEPT VIAS.
VIAS ARE TO BE MASKED PER ARTWORK MASTER.
- SILKSCREEN BOTH SIDES USING SILKSCREEN MASTER.
SILKSCREEN TO BE WHITE NON-CONDUCTIVE, NON-NUTRIENT EPOXY INK,
OR ACRYLIC PRINT TAYO UR-4000 MW300.
- WARPAGE NOT TO EXCEED 0.10 PER LINEAR INCH
- PCB MUST PASS 100% ELECTRICAL TESTING
FOR OPENS AND SHORTS IN ACCORDANCE WITH IPC-ET-652.
TRUE NETLIST TESTING REQUIRED USING SUPPLIED IPC NETLIST.
- ALL REWORK MUST FONCOFM TO IPC GUIDELINES:
IPC-R-700C (OR MOST CURRENT REVISION).
- MINIMUM CONDUCTOR WITH TO BE 0.007".
MINIMUM CONDUCTOR TO CONDUCTOR AND CONDUCTOR TO PAD SPACING TO BE 0.007"
- SURFACE FINISH: ENIG (ELECTROLESS NICKEL IMMERSION GOLD)
2-5 MICRO INCHES IMMERSION (SOFT) GOLD PLATE OVER 118-236 MICO INCHES NICKEL
- ALL SOLDER PADS TO BE FREE OF SILKSCREEN INK.
- PCB SHALL BE ROHS 2 COMPLIANT (2011/65/EU), REACH COMPLIANT (EC1907/2006)
AND COMPATIBLE WITH ROHS ASSEMBLY PROCESS
- VIA REQUIREMENTS: FOLLOW SUPPLIED GERBERS.
SOLDER MASK ON PAD - FOLLOW SOLDER MASK GERBER FILES
- OK TO ADD COPPER THIEVING TO SPARSELY POPULATED AREAS OF PCB AS NEEDED
- IF COPPER PAD OR PLATED HOLES TOUCH BOARD OUTLINE,
FOLLOW GERBER AND ACCEPT COPPER EXPOSED AT BOARD EDGE
- ADD TEARDROPS TO ALL PADS AT TRACE JUNCTIONS

UNLESS OTHERWISE SPECIFIED

DIMENSIONS ARE IN INCHES
TOLERANCES ON:
2 PL DECIMALS +/- 0.010
3 PL DECIMALS +/- 0.005
ANGLES +
FRACTIONS +

FABRICATION

AxionAdapter

9/22/20 12:25 AM

DWG 00000009

REV 1