

NOTES: (UNLESS OTHERWISE SPECIFIED)
THIS FAB SHOULD BE RoHS COMPLIANT .

1. FABRICATE TO IPC-A-600, CURRENT REVISION
2. BOARD SHALL MEET THE INSPECTION CRITERIA OF
 - a- ACCEPTABILITY AS PER IPC-A-600 (LATEST REVISION) CLASS II
 - b- QUALIFICATION AND PERFORMANCE AS PER IPC-6012 (LATEST REVISION) CLASS II.
3. MATERIAL: Isola_370HR (RoHS COMPLIANT MATERIAL) OR EQUIVALENT. GLASS TRANSITION TEMPERATURE MUST MEET OR EXCEED THE TEMPERATURE EXHIBITED WITH HIGH TEMPERATURE PROCESSES ASSOCIATED WITH LEAD FREE ASSEMBLY.
4. APPLY SOLDER MASK OVER BARE COPPER (SMOBC) IAW IPC-SM-840, BOTH SIDES, USING LPI, COLOR BLUE.
5. LPI SOLDER MASK TAIYO PSR4000 (RoHS COMPLIANT MATERIAL) OR EQUIVALENT WILL BE USED ON BOTH SIDES.
6. SOLDER MASK REQUIREMENTS FOR VIAS:
 - a) ALL VIAS SHOWN IN DATA WITH SOLDER MASK RELIEF OF 6 MILS OVER VIA DRILL SIZE FROM BOTTOM SIDE, SHALL HAVE SOLDER MASK ENCROACHED (DRILL PLUS 6 MILS).
 - b) ALL TENTED VIAS MUST BE PLUGED WITH SOLDERMASK.
7. SOLDER MASK REGISTRATION TO BE WITHIN DIAMETRICAL TRUE POSITION OF + / - 0.002 WITH APPLICABLE HOLE / PAD.
8. FINISH: ENIG.
9. SILKSCREEN USING WHITE - HAVEN PC421 (NON-CONDUCTIVE OR EQUIVALENT RoHS COMPLIANT MATERIAL) BOTH SIDES DISTORTION OF SILKSCREEN IS ACCEPTABLE OVER TRACES. EPOXY INK ON SOLDER LANDS IS NOT ACCEPTABLE.
10. VENDOR LOGO AND DATE CODE TO BE MARKED SOLDER SIDE IN SILKSCREEN. MAXIMUM HEIGHT 0.12 INCHES.
11. 100% ELECTRICAL TEST REQUIRED FOR CONTINUITY. BOARD SHALL HAVE A UL-RATING OF 94V-0. UL SYMBOL AND RATING SHALL BE MARKED SOLDER SIDE IN SILKSCREEN.
12. REMOVE ALL UNUSED PADS FROM INTERNAL LAYERS.
13. 274X GERBER/ODB++ USED FOR FAB MUST BE VERIFIED AGAINST THE PROVIDED IPC356 NETLIST. COPPER SLIVERS THAT ARE LESS THAN 0.003 IN WIDTH BETWEEN ANTI-PAD TO PLANE EDGE, ANTI-PAD TO SPLIT PLANE AND ANTI-PAD TO ANTI-PAD MUST BE REMOVED FROM THE MANUFACTURING ARTWORK. A NETLIST COMPARISON MUST BE PASSED WITH NO VIOLATION AFTER THE REMOVAL OF SLIVERS. ANY REQUIREMENT FOR SLIVER REMOVAL ABOVE OR EQUAL TO THE 0.003 COPPER WIDTH MUST BE ADDRESSED AND APPROVED IN WRITING BY SUPPLIER.
14. Finish:ENIG 3-8/180-220 microinches

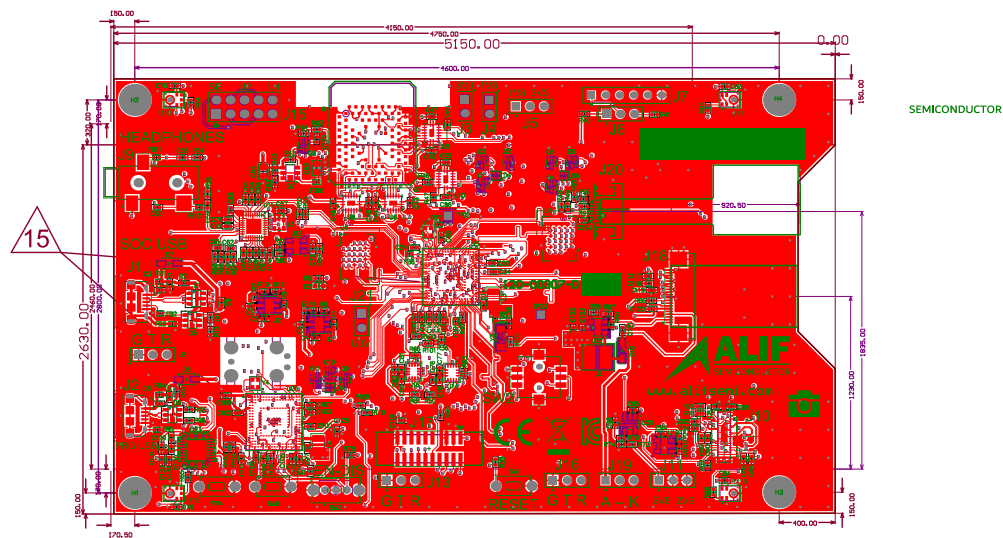
15. ALL 6 MIL VIA-IN-PAD ARE TO BE FILLED.

100 OHM DIFFERENTIAL PAIRS

MIPI_CSI_0_N
MIPI_CSI_0_P
MIPI_CSI_1_N
MIPI_CSI_1_P
MIPI_CSI_C_N
MIPI_CSI_C_P
MIPI_DSI_0_N
MIPI_DSI_0_P
MIPI_DSI_1_N
MIPI_DSI_1_P
MIPI_DSI_C_N
MIPI_DSI_C_P

90 OHM DIFFERENTIAL PAIRS

USB_CYP_N
USB_CYP_P
USB_DEV_N
USB_DEV_P
USB_D_N
USB_D_P
USB_FTDI_N
USB_FTDI_P
USB_U_N
USB_U_P



Symbol	Count	Hole Size	Plated	Via/Pad	Hole Tolerance (+)	Hole Tolerance (-)
■	1	31.50mil (0.800mm)	NPTH	Pad		
◇	1	43.31mil (1.100mm)	NPTH	Pad		
☆	2	59.00mil (1.496mm)	PTH	Pad		
□	2	66.93mil (1.700mm)	NPTH	Pad		
✕	3	31.00mil (0.787mm)	PTH	Pad		
▼	3	40.00mil (1.016mm)	PTH	Pad		
△	3	42.00mil (1.067mm)	NPTH	Pad		
✕	4	19.69mil (0.500mm)	PTH	Pad		
☆	4	21.65mil (0.550mm)	PTH	Pad		
✕	4	38.00mil (0.961mm)	NPTH	Pad		
✕	4	96.50mil (2.451mm)	NPTH	Pad		
✕	4	125.00mil (3.175mm)	PTH	Pad		
●	5	38.00mil (0.966mm)	PTH	Pad		
○	6	49.21mil (1.250mm)	PTH	Pad		
○	10	45.00mil (1.143mm)	PTH	Pad		
○	27	40.16mil (1.020mm)	PTH	Pad		
B	41	8.00mil (0.203mm)	PTH	Via	0.00mil (0.000mm)	8.00mil (0.203mm)
E	93	8.00mil (0.203mm)	PTH	Via		
C	120	6.00mil (0.152mm)	PTH	Via		
D	767	10.00mil (0.254mm)	PTH	Via		
1104 Total						

Slot definitions: Routed Path Length = Calculated from tool center position to tool end center position.
Hole Length = Routed Path Length + Tool Size = Slot Length as defined in the PCB layout

Layer	Name	Material	Thickness	Constant	Board Layer Stack
	Top Overlay				
	Top Solder	SM-001	0.70mil	4	
1	L1	CF-003	1.40mil		
	Dielectric 1	PP-008	3.10mil	4.1	
2	L2 (GND)	CF-003	0.60mil		
	Dielectric 3	Core-035	4.91mil	4.7	
3	L3-SIG	CF-003	0.60mil		
	Dielectric 2	PP-006	41.58mil	4.1	
4	L4-SIG	CF-003	0.60mil		
	Dielectric 6	Core-035	4.91mil	4.7	
5	L5 (PWR)	CF-003	0.60mil		
	Dielectric 8	PP-008	3.10mil	4.1	
6	L6	CF-003	1.40mil		
	Bottom Solder	SM-001	0.70mil	4	
	Bottom Overlay				
Total board thickness:			63.40mil		