



Board Drawing

DRILL TABLE

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad
	307	8.00mil (0.203mm)	PTH	Round	Top Layer – Bottom Layer	Via
	1980	8.00mil (0.203mm)	PTH	Round	Top Layer – Bottom Layer	Via
H	39	10.00mil (0.254mm)	PTH	Round	Top Layer – Bottom Layer	Via
	60	10.00mil (0.254mm)	PTH	Round	Top Layer – Bottom Layer	Pad
C	20	22.00mil (0.559mm)	PTH	Round	Top Layer – Bottom Layer	Pad
B	2	24.00mil (0.610mm)	PTH	Slot	Top Layer – Bottom Layer	Pad
	2	25.59mil (0.650mm)	NPTH	Round	Top Layer – Bottom Layer	Pad
	2	27.56mil (0.700mm)	PTH	Slot	Top Layer – Bottom Layer	Pad
☆	3	31.00mil (0.787mm)	PTH	Slot	Top Layer – Bottom Layer	Pad
◇	2	35.43mil (0.900mm)	PTH	Slot	Top Layer – Bottom Layer	Pad
✕	43	38.00mil (0.965mm)	PTH	Round	Top Layer – Bottom Layer	Pad
A	4	40.00mil (1.016mm)	NPTH	Round	Top Layer – Bottom Layer	Pad
⌘	24	40.95mil (1.040mm)	PTH	Round	Top Layer – Bottom Layer	Pad
G	6	41.00mil (1.041mm)	NPTH	Round	Top Layer – Bottom Layer	Pad
D	12	41.00mil (1.041mm)	PTH	Round	Top Layer – Bottom Layer	Pad
	15	43.00mil (1.092mm)	PTH	Round	Top Layer – Bottom Layer	Pad
O	5	50.00mil (1.270mm)	PTH	Round	Top Layer – Bottom Layer	Pad
	2	61.02mil (1.550mm)	PTH	Round	Top Layer – Bottom Layer	Pad
E	2	62.00mil (1.575mm)	PTH	Round	Top Layer – Bottom Layer	Pad
⊕	4	78.00mil (1.981mm)	PTH	Round	Top Layer – Bottom Layer	Pad
□	4	120.00mil (3.048mm)	PTH	Round	Top Layer – Bottom Layer	Pad
F	2	130.00mil (3.302mm)	NPTH	Round	Top Layer – Bottom Layer	Pad
	2540 Total					

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

LAYER STACK

Layer	Name	Material	Thickness	Constant	Board Layer Stack
	Top Overlay				
	Top Solder	Solder Resist	0.40mil	3.5	
1	Top Layer	CF-003	1.40mil		
	Dielectric 1	370HR	8.00mil	4	
2	L2-GND	CF-004	0.70mil		
	Dielectric 2	370HR	8.00mil	4	
3	L3-SIG	CF-004	0.70mil		
	Dielectric 3	370HR	8.00mil	4	
4	L4-GND	CF-004	0.70mil		
	Dielectric 4	370HR	8.00mil	4	
5	L5-PWR	CF-004	0.70mil		
	Dielectric 5	370HR	8.00mil	4	
6	L6-SIG	CF-004	0.70mil		
	Dielectric 6	370HR	8.00mil	4	
7	L7-GND	CF-004	0.70mil		
	Dielectric 7	370HR	8.00mil	4	
8	Bottom Layer	CF-003	1.40mil		
	Bottom Solder	Solder Resist	0.40mil	3.5	
	Bottom Overlay				

- FABRICATION NOTES:

1. MATERIALS:
ISOLA 370HR with 1/2 oz starting Copper and 1oz. copper plating.
2. Board is 8 layers, finished board thickness = .062" +/- 10%.
Shall conform to IPC-2221, UL 94V-0 and European Directive 2011/65/EC (ROHS2).
3. Finished board shall meet requirements of IPC-A-600.
Finished trace width and outer diameter of plated holes shall be within .001" or less of image on photo tool master.
Annular rings to meet IPC-600J class 2.
4. Holes to be plated through in accordance with IPC-600J.
Dimensions shown are for finished plated through holes.
5. All exposed copper to be finished with ENIG, as per IPC-4552.
6. Screen legend with white non-conductive ink, both sides.
Contact areas of SMT components to be free of ink.
7. SMOBC: BLACK LPI solder mask. Both sides.
8. Board to be 100% bare board tested and report provided.
9. Design rules used: 5mil trace, 5mil space.
10. Plated holes have a +/- 0.003 inch tolerance.
Non-plated +/- 0.0015 unless otherwise specified
11. DO NOT adjust Stack-up or Trace widths without contacting NEXTGEN RF.