



BOARD CHARACTERISTICS

Copper Layer Count: 8
 Board overall dimensions: 3.550" x 3.982"
 Min track/spacing: 0.006" / 0.006"
 Copper Finish: ENIG
 Castellated pads: No
 Edge card connectors: Yes, Bevelled, 30 degrees.
 Board Thickness: 0.062"
 Min hole diameter: 10.00 mils
 Impedance Control: Yes
 Plated Board Edge: No

Layer Name	Type	Material	Finished Thickness (mils)	Base Thickness (mils)	Construction Details
F.Mask	Top Solder Mask		0.5 mils	0.5 mils	
F.Cu	copper		1.45 mils	0.35 mils (0.25oz)	1x1080
Dielectric 1	prepreg	N7000-2HT	3.648 mils	3.9 mils	
In1.Cu	copper		0.7 mils		1x106+1x1080
Dielectric 2	core	N7000-2HT	4 mils	4 mils (0.5oz/0.5oz core)	
In2.Cu	copper		0.7 mils		
Dielectric 3	prepreg	N7000-2HT	7.016 mils	7.8 mils	2x1080
In3.Cu	copper		0.7 mils		
Dielectric 4	core	N7000-2HT	28 mils	28 mils (0.5oz/0.5oz core)	2x2116+3x7628
In4.Cu	copper		0.7 mils		
Dielectric 5	prepreg	N7000-2HT	7.016 mils	7.8 mils	2x1080
In5.Cu	copper		0.7 mils		
Dielectric 6	core	N7000-2HT	4 mils	4 mils (0.5oz/0.5oz core)	1x106+1x1080
In6.Cu	copper		0.7 mils		
Dielectric 7	prepreg	N7000-2HT	3.648 mils	3.9 mils	1x1080
B.Cu	copper		1.45 mils	0.35 mils (0.25oz)	
B.Mask	Bottom Solder Mask		0.5 mils	0.5 mils	

FAB NOTES:

1. IPC-6012E Class 2
2. Matte Green soldermask, White silkscreen.
3. Fabricate on Nelco N7000-2HT
4. 1oz outer, 1oz inner copper.
5. Bevel edge connector 30 degrees minimum.
6. Fill all vias with non-conductive material.
7. Board target thickness shall be 0.062"

ASSEMBLY NOTES:

1. J-STD-001G Class 2
2. BOM provided with submitted files shall be the controlling document for component information.
3. Do not apply solder to pads of DNP components
4. Stake components specified in BOM with 3M 2216
5. Conformal Coat with Arathane 5750:
Ensure coating does not prevent electrical contact with connectors and mounting holes.

Sierra Lobo	
Sheet:	
File: mainboard.kicad_pcb	
Title: mainboard	
Size: B	Date: 2023-06-15
KiCad E.D.A.	kicad 7.0.1
Rev: C	Id: 1/1

