SYNERGY ELECTRONICS - PCB Manufacturing Capabilities

	Item	Unit	Description or parameter
1	Arlon material model		AD350,AR1000,25FR,33N, Diclad527
2	Rogers material model		Ro4350, Ro4350B, Ro4003, Ro4003C, Ro3003, TR5880
3	Rogers PP model		Ro4403 (0.10mm), Ro4450B (0.10mm)
4	Taconic material model		RF-35, TLX-8, TLC-32
5	Taconic PP model		TP-32 (0.10mm)
6	High Tg material model		Shengyi Tg = 170 (S1141, S1170)
7	Halogen free material (High Tg)		Shengyi: S1165, PP:S0165
8	Halogen free material (normal Tg)		Shengyi: S1155, PP:S0155
9	PCB Thickness	mm	0.13-7.0 (if the board thickness≤0.5mm, the panel size must be ≤ 18 in
10	FR-4 PREPREGS		Shengyi 7628 2116 1080 3313 106
11	Copper Foil	um	12, 18, 35, 70
12	Outer Layer with differenct copper foil on both sides		18/35, 35/70
13	Finished copper (18um based copper)	um	≥35 (normal is 52 um, namely 1.5 oz)
14	Finished copper (35um based copper)	um	≥ 55
15	Finished copper (70um based copper)	um	≥90
16	solder mask color		Green, Yellow, Black, Blue, Red, White, Matte Green
17	Legend color		White, Black, Yellow
18	Surface finishing		HAL, HAL Lead free, Flash Gold, Immersion Gold, Immersion Silver, Immersion Tin, OSP, Hard Gold
19	selective surface finishing		ENIG+OSP, ENIG+GOLD Finger, Flash Gold+HASL, Flash Gold+Gold Finger, Immersion Silver+ Gold Finger, Immersion Tin + Gold Finger
20	Peelable solder mask	mm	0.2 - 0.5
21	Countersink size and angle		Angle 82, 90, 120 ϕ≤10mm
22	min core thickness	mm	0.05
23	Max drilling	mm	6.3
24	Mix laminating		Rogers/Taconic/Arlon mixing with FR4
25	Board Layer	Layer Count	2-40
26	Max board size for 2 Layers	inch	23*35 (Length above 30 inch needs to be evaluated
27	Max board size for 4 Layers	inch	22.5*33.5 (Length above 30 inch needs to be evaluated
28	Max board size (≥6 Layers)	inch	22.5*26.5 (Length above 22.5 inch needs to be evalu
29	Min Board Size	mm	10*10
30	Max panel size for RF material (Rogers, Arlon, Tyconi cseries)	inch	16*18
31	Tolerance for outline routing	mil	±4 (Complex routing and slot, needs to be evaluated)
32	Min Internal angle radius	mm	0.4
33	Tolerance of depth control slot or blind hole (NPTH)	mm	±0.10
34	Mechanical blind & burried vias with times laminating		laminating less than 3 times
35	Gold thickness (Immersion Gold)	um	0.025 - 0.10
36	Nickel thickness (Immersion Gold)	um	3/5
37	Silver thickness (Immersion Silver)	um	0.1 - 0.3
38	Min Tin thickness (HAL lead free)	um	0.4 (Large bare copper area for HAL
39	Tin thickness (Immersion Tin)	um	> 0.8
40	Gold thickness (Hard gold plating)	um	0.15 - 3
41	Gold thickness (gold finger/ connector)	um	0.25 - 1.3 (Min thickness point)
42	Nickel thickness (gold finger/ connector	um	3/5
43	Gold thickness (Flash gold)	um	0.025 - 0.10
44	Nickel thickness (Flash gold)	um	3/5
45	Min copper thickness in the hole	um	Average 25um, Min thickness point above 20um
46	Min thickness for isolation layer	mm	0.075 (only for half oz base copper)
47	Min pad for BGA	mil	10 (For flash gold pcb can be 7 mil)
48	Min pad	mil	12 (min 0.10mm only for laser drilling
49	Min solermask thickness	um	10
50	Min width for the legend which soldermask layer	mil	8
51	Min soldermask bridge	mil	4 (Green Soldermask), 5 (all other soldermask colours) (if base copper≤ 1oz can be 5 mil) (if base copper is 2-4 oz it can be 6 mil)
52	Min. width of soldermask cover line (single side)	mil	2.5 (pcb partial should be 2 mil)
53	Min soldermask clearance (single side)	mil	2 mil (10% area of flash gold pcb can be 1.5 mil, 10% area of another surface treatment should be 1 mil
54	Max via damision for soldermask via plugging	mm	0.65
55	Soldermask thickness cover Via	um	5/8
56	Tolerance for the press hole without soldering	mil	± 2

57	Max finished copper	oz	5 oz (175um)
58	Min distance for inner layer between the edge (without exposed copper)	mil	10
59	Min distance for the isolation tape inner layer	mil	8
60	Min inner layer isolation annulus (singel side)	mil	8 (8 mil for less than 6 layer PCB), 10 (10 mil for above 8 layer PCB), (8 mil for PCB partial can shave pad)
61	Min width of inner layer PAD (single side)	mil	5 (for base copper 18, 35 um, PCB partial can be 4.5 mil), 6 (for base copper 70 um), 8 (for base copper 105 um)
62	Impedance tolerance	%	±5Ω (<50Ω), ± 10% (≥50Ω); ≥50Ω can be ± 5% (need to evaluate)
63	HDI		1+n+1, 1+1+n+1+1, 2+n+2
64	RCC material		copper thickness 12 mil, resin thickness 65, 80, 100 um (resin thickness after laminating will be 55, 70, 90 um
65	Min laser drill hole size	mm	0.10 (Laser drill hole Depth ≤ 55 um), 0.13 (Laser drill hole depth ≤ 100 um
66	Max PCB thickness for mechanical drilling tool dia (0.10mm)	mm	0.60 (≤ 6 Layer)
67	Max PCB thickness for mechanical drilling tool dia (0.15mm)	mm	1.20 (≤ 8 Layer)
68	Max PCB thickness for mechanical drilling tool dia (0.25mm)	mm	5
69	Min drilling diameter for PTFE Material	mm	0.45 Min drilling size 0.4mm, if above 0.4mm, needs to be evaluated or add the drilling size and use another material instead of PTFE
70	PCB thickness tolerance (PCB thickness above 1 mm)	mm	PCB thickness tolerance ±10 %
71	PCB thickness tolerance (PCB thickness ≤ 1 mm	mm	± 0.1
72	special PCB thickness tolerance (not include special layer by layer space demand)	mm	PCB thickness ≤ 2.0mm, tolerance ± 0.1; PCB thickness 2.0 - 3.0, tolerance ± 0.15; ≥ 3.0 mm, tolerance ± 0.20
73	Aspect ratio		20:1 (not include tooling size ≤ 0.2 mm, above 2:1 will need to be evaluated)
74	Min. diameter of connecting hole	mm	0.45
75	Machining		routing; V-cut' Tab connecting; stamp holes
76	Min routing tool diameter	mm	0.6
77	Min gap between hole wall to track	mil	6 (≤ 8 layer), 8 (≤ 12 layer), 9 (≤ 20 layer), 10 (≤ 28 layer)
78	Min gap between hole wall to track (Blind and buried via PCB)	mil	9 (once laminating); 10 (twice or three times laminating)
79	Min via pad annulus of out layer	mil	4 (12, 18 um) can be PCB partial 3.5, 4.5 (35 um), 6 (70um) 10 (140 um)
80	Min gap for no copper exposure after routing	mil	8
81	Min NPTH diameter tolerance	mil	± 2 (Limited tolerance + 0 /- 0.05 or + 0.05/-0
82	Min PTH slot tolerance	mm	± 0.15
83	Min diameter of routing tool diameter	mm	0.6
84	Countersink hole		PTH and NPTH, Max hole angle 130, Max hole diameter less than 6.3mm
85	hole position tolerance	mil	± 3
86	Min insulation belt width	mil	3 (Base copper 18 um), 4 (Base copper 35 um), ≥ 3 mil
87	inner layer oxidation		Brown oxidation
88	Min line gap of inner layer (105um based copper	mil	5
89	Min line gap of inner layer (140 um based copper	mil	7
90	Min line gap of inner layer (18 um based copper	mil	3
91	Min line gap of inner layer (35 um based copper	mil	3.5
92	Min line gap of inner layer (70 um based copper	mil	4
93	Min line width of inner layer (105 um based copper	mil	5
94	Min line width of inner layer (140 um based copper	mil	7
95	Min Line width of inner layer (18 um based copper	mil	3
96	Min line width of inner layer (35 um based copper	mil	3
97	Min Line width of inner layer (70 um based copper	mil	4
98	Min line gap of out layer (105 um based copper	mil	6
99	Min line gap of out layer (12, 18 um based copper	mil	3.0 (for 18 um copper), 2.5 (for 12 um copper)
100	Min Line gap of out layer (140 um based copper	mil	7
101	Min line gap of out layer (35 um based copper	mil	3.5
102	Min line gap of out layer (70 um based copper	mil	5
103	Min line width of out layer (105 um based copper	mil	8
104	Min line width of out layer (12, 18 um based copper	mil	4 (for 18 um copper), 3 (for 12 um copper)
105	Min line width of out layer (140 um based copper	mil	9
106	Min Line width of out layer (35 um based copper	mil	4.5
107	Min line width of out layer (70 um based copper	mil	6
108	Min gap between line to pad of out layer, min gap between pad to pad or out layer	mil	3 (for 12, 18 um copper), 3.5 (for 35 um copper), 5 (for 70 um copper), 6 (for 105, 140 um copper)
109	Min warpage	%	0.1 (≤ 0.3 needs to be evaluated)
110	Maz size of dry film covering slot		5 mm *3.0mm; Min dry film clearance (single side) above 15 mil

111	Max width of dry film covering hole (single side)	mil	10
112	Max hole diameter of dry film covering	mm	4.5
113	Angle tolerance of Gold Finger chamfer		± 5°
114	Rest thickness tolerance of Gold Finger chamfer	mil	± 5
115	Max length of Gold Finger	inch	2
116	Min gap between Gold Finger	mil	6
117	different length Gold Finger surface treatment		flash gold/ immersion gold plating; electrical hard gold plating
118	V-Cut angle model		20°, 30°, 45°, 60°
119	V-Cut no copper exposure, distance between V-cut line to Circuit (1.0 < PCB thickness ≤ 1.6 mm)	mm	0.36 (20°), 0.4 (30°), 0.5 (45°), 0.6 (60°)
120	V-Cut no copper exposure, distance between V-cut line to Circuit (1.6 < PCB thickness ≤ 2.4 mm)	mm	0.42 (20°), 0.51 (30°), 0.64 (45°), 0.8 (60°)
121	V-Cut no copper exposure, distance between V-cut line to Circuit (2.5<PCB thickness≤3.0mm)	mm	0.47 (20°), 0.59 (30°), 0.77 (45°), 0.97 (60°)
122	V-Cut no copper exposure, distance between V-cut line to Circuit (PCB thickness ≤ 1.0mm)	mm	0.3 (20°), 0.33 (30°), 0.37 (45°), 0.42 (60°)
123	V-Cut symmetrical tolerance	mil	± 4
124	V-Cut angle tolerance		± 5°
125	V-Cut rest thickness tolerance	mil	± 4
126	Max diameter peelable soldermask cover	mm	2
127	Distance between Peelable soldermask and pad	mil	12
128	Min gap of copper net for ground	mil	5 (for 12, 18, 35 um copper), 8 (for 70 um copper)
129	Min width of copper net for ground	mil	5 (for 12,18, 35 um copper), 10 (for 70 um copper)
130	(for 12, 18 um based copper) Min Legend width and gap		Legend width 4 mil; legend height 25 mil
131	(for 35 um based copper) Min Legend width and gap		Legend width 5 mil; Legend height 30 mil
132	(for 70 um based copper) Min Legend width and gap		Legend width 6 mil; legend height 45 mil
133	Min gap between Legend and Pad	mil	6
134	Min connect resitance	Ω	10
135	Min gap of test point to PCB edge	mm	0.5
136	Normal Max Test Current	mA	200
137	Normal Max Test Voltage	V	250
138	Min Gap of test Pad to Pad	mil	3.9
139	Min size of Test Pad	mil	3.9
140	Max insulative resistance	MΩ	100
141	Ionic clearness test	ug/cm2	≤ 1
142	Circuit peelable strength	Newton/cm	7.8
143	Hardness of soldermask	H	6
144	Burning resisting		94 V-0