



MULTECH PCB TECHNOLOGIES CO., LIMITED 艾尔特线路科技(HK)有限公司

Multech Capability

1.0 板类型(Type of board)

Metal base PCB, Hi-Tg heavy copper foil PCB, Flat winding PCB, High frequency PCB, Mixed dielectric base PCB, back-plate circuits, Flex and Flex-rigid.

2.0 物料类型 (Laminates type)

FR-4,CEM-1/CEM-3,ISOLA,Teflon,PTFE,Rogers,Nelco,

Terquist,Thermagon,Aluminium, Tg130°C(150°C,170°C,180°C,210°C), Alon. Polyclad

3.0 板层 (Layer coun)

1~40layers

4.0 板厚 (Board thickness)

4.1 Min: 2-Layer: 0.40mm (0.016")

4.2 4-Layer: 0.40mm (0.016")

4.3 6-Layer: 0.55mm (0.022")

4.4 8-Layer: 0.60mm (0.024")

4.5 10-Layer: 1.22mm (0.048")

4.6 Max: 8mm (0.3150")

5.0 完成铜厚 (Base copper thickness)

5.1 1/3oz to 13 oz

6.0 HDI/Micro-via :yes

7.0 表面处理 (Surface finish)

HASL / Lead Free HASL (Sn100c,thickness :7-15um),

Immersion Gold (1-3u",4-6u")/ Silver(78.7u") / Tin (\geq 0.8um),

Flash Gold(max:5u"),

Hard Gold plating (max: 50u"),

Selective thick Gold Plating,

Gold Finger (max. :50u"),

Plating Silver (Silver through Hole, Carbon ink, Peelable mark, OSP

ENEPIG

OSP

8.0 阻抗控制 (Impedance Control)

Single trace or differential controlled, 50, 60, 70, 100, 110, 120, 130 $\pm 5\%$

9.0 钻孔 (Drilling)

9.1. 孔径公差(Hole Tolerance(after drilling)):

$\phi \leq 3.175\text{mm}$ 为 $\pm 3\text{mil}$; $\phi \geq 3.20\text{mm}$ 为 $\pm 4\text{mil}$

$\phi \leq 3.175\text{mm}$ is $\pm 3\text{mil}$; $\phi \geq 3.20\text{mm}$ is $\pm 4\text{mil}$

9.2. 最大孔径:6.5mm

Max dia: 6.5mm

9.3. 最小孔径:0.1mm

Min dia: 0.1mm

9.4. 叠板高度:

Stack height:

$N \times X \div D \leq 11$

N: Board thickness/板厚

X: Stack number/叠板数

D: Smallest drill size/最小钻孔直径

9.5. 背钻(Backdrilling)

10.0 沉铜 (PTH)

10.1. 最小可生产孔径:0.15mm

Min hole dia:0.15mm

10.2. 纵横比:12:1

Aspect ratio:12:1

11.0 沉金

11.1 纵横比:5:1

Aspect ratio:5:1

12.0 干菲林

Dry Film

12.1. 封孔最小 RING:7mil(A/W)

Tenting min ring:7mil(A/W)

12.2. 最小线粗/线隙:3mil/3mil(A/W)

Min line width/line space:3mil/3mil(A/W)

13.0 图形电镀

Pattern Plating

13.1. 最小孔径:0.15mm

Min hole dia:0.15mm by mechanical

14.0 蚀板

Etching

14.1. 最大尺寸: 710mm×1100mm

Max size: 710×1100mm

14.2. 板厚: 0.2~8mm

Board thickness: 0.2~8mm

15.0 绿油、碳油、白字、丝印蓝胶

W/F、Carbon coating、C/M、Silk screen Peelable Solder Mask

15.1 最大尺寸:710mm×1100mm

Max size: 710×1100mm

16.0. 绿油桥

W/F dam

16.1 SMT 最小间隙(指黑菲林): 7.0mil(绿油板)、8.0mil(黑油)

Minimum spacing between SMT PAD and SMT PAD (A/W): 7.0mil (Green ink)、8.0mil(Black ink)

16.2 绿油桥最小宽度(绿油):3.5mil、(黑油)6mil

Minimum solder mask dam width:(Green ink)3.5mil、(black ink)6mil

16.3 绿油厚度(最小), 丝印一次, 线角:0.2mil, 线面:0.5mil; 丝印二次, 线角:0.4mil, 线面:1.2mil。

Solder mask thickness (min), single coating, trace corner:0.2mil, trace surface:0.5mil; double coating, trace corner:0.4mil, trace surface:1.2mil.

16.4 丝印最小网格能力:8mil×8mil(2 盎司底铜以下板);

15mil×15mil(2OZ 底铜以上板)。

Min copper mesh of silk screen capability:8mil×8mil (<2OZ base copper board)

15mil×15mil (≥2OZ base copper board)

17.0. 碳油

Carbon coating

17.1 印油 Pad 和 铜 Pad 单边差值:

carbon PAD should be bigger than copper pad each side:

- >10mil 侧面不露铜
- >10mil No exposure copper in PAD edge
- >7mil 侧面露铜
- >7mil exposure copper in PAD edge

17.2 印油 Pad 最小间距:12mil(A/W)

Distance between carbon coating PADS: $\geq 12\text{mil(A/W)}$

17.3 需印碳油之 Pad 到 Ring 径及其它 Pad 的最小距离: 10mil

Distance between carbon coating PAD and hole ring or other PADS which are not required to be printed with carbon: $\geq 10\text{mil}$

17.4 碳油阻值:

Carbon coating resistance:

- $\leq 500 \Omega$ 印一次
- $\leq 500 \Omega$ Single coating
- $\leq 300 \Omega$ 印二次
- $\leq 300 \Omega$ Double coating

test Area: 10mm \times 1mm

18.0 白字

C/M

18.1 最小线粗:5mil

Min line width: 5mil

18.2 最小线隙:5mil

Min line spacing:5mil

18.3 最小字符内径:

Min character inner dia:

8mil/HOZ 或 1OZ base Cu; 10mil/2OZ base Cu

8mil/HOZ~1OZ base copper; 10mil/2OZ base copper

18.4 白字厚度:

C/M thickness:

- 0.4~0.8mil 100T 网印一次
- 0.4~0.8mil silk screen one time with 100T
- 3~4mil 51T、100T 网各印一次
- 3~4mil silk screen twice with 51T&100T

19.0 丝印蓝胶

Silk screen Peelable solder mask

19.1 蓝胶厚度:

Peelable solder mask thickness:

- 4~8mil 51T 网印一次
- 4~8mil Coating one time with 51T
- 8~20mil 21T 网印二次
- 8~20mil Coating twice with 21T

20.0 镀金手指

G/F

20.1. 金厚:2~50 μ "

Au thickness: 2~50 μ "

20.2. 镍厚:50~300 μ "

Ni thickness: 50~300 μ "

20.3. 有效槽深:

Cell depth(useful): 8.0 "

21.0 喷锡

HAL

9.1. 锡厚:100~1000 μ " (沉铜孔) 20~2000 μ " (SMT)

Sn/Pb thickness: 100~1000 μ " (PTH hole), 20~2000 μ " (SMT)

9.2. 纵横比:

Aspect ratio:

3:1(max) 垂直喷锡机

3:1(max) Vertical machine

22.0 抗氧化

OSP

22.1. 板厚:0.4~4.0mm

Board thickness: 0.4~4.0mm

23.0 锣板

Routing

23.1. 公差:±4mil (孔到板边,板边到板边,槽和孔径公差)

Tolerance:±4mil(hole to board edge, board edge to board edge, slot and hole dia)

24.0 啤板

Punching

24.1. 公差: ±7mil (外围)

Tolerance: ±7mil(out-line)

24.2. 啤坑之间距离:≥2mm

Distance between punching slot: ≥2mm

24.3. 最小啤坑尺寸:1mm×2mm

Min punching slot size: 1mm×2mm

24.4. 最大啤板尺寸:16.5 " ×14 "

Max punching unit size: 16.5 " ×14 "

25.0 V 坑

V-cut

25.1. 板子尺寸:

Board size:

最大:650×650mm² 最小:100×100mm²

Max: 650×650mm² Min: 100×100mm²

25.2. 板厚: 0.3~4.5mm

Board thickness: 0.3~4.5mm

25.3. V 坑线平行度: ±3mil (最大)

V-cut parallel tolerance: ±3mil (Max)

25.4. 保留厚度公差: ±3mil

Residual web thickness tolerance: ±3mil

25.5. 直角精度:90±2°

The rectangular precision: 90±2°

25.6. 直角与管位孔基准线位置精度:90±2°

Position precision at rectangular to target hole base line: 90±2°

25.7. 上、下刀对准度:

Leveling tolerance range on up and down side of v-cut line: ≤4mil

25.8. 跳刀最小距离:15mm

Min jump v-cut step: 15mm

25.9. 跳刀下点及起点位置公差:2mm(单向走刀);6mm(双向走刀)

Jump v-cut down point and end point position tolerance:2mm(single direction);6mm(double direction)

26.0 板曲

Warpage

板料厚度		翘曲度(%)
mm	inch	双面及多层
0.5	(0.020)	1.0
0.9	(0.037)	
1.0	(0.039)	0.7
1.60	(0.064)	

PCB thickness		Warpage(%)
mm	inch	D/S & M/L
0.5	(0.020)	1.0
0.9	(0.037)	
1.0	(0.039)	0.7
1.60	(0.064)	