GUH CIRCUIT INDUSTRY (PG) SDN BHD (199201000239 (231743-T))

Single Sided PCB Production & Technical Capabilities

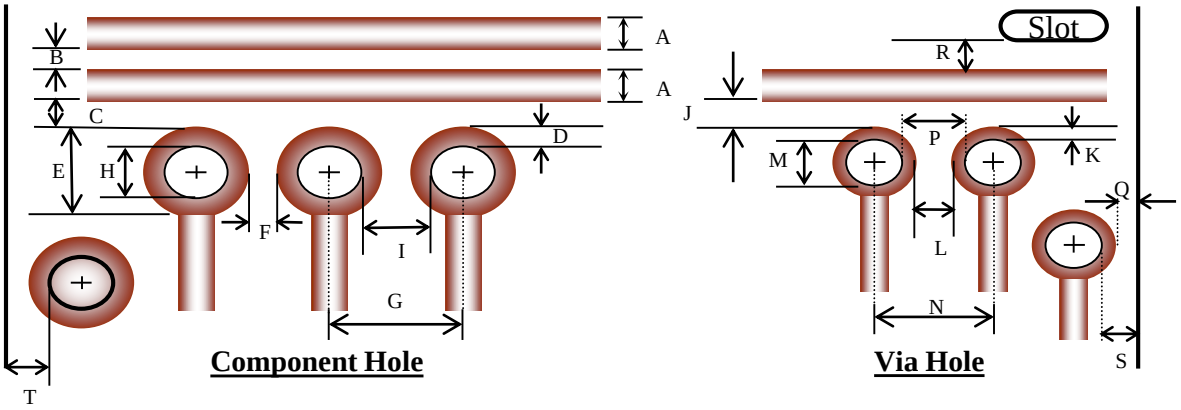
No.	Design Items	Specification / Tolerance
1	Minimum copper track width / conductor width (by screen printing)	0.18 mm
2	Minimum gap between copper track / copper pad (by screen printing)	0.18 mm
3	Minimum copper ring	0.20 mm
4	Printed copper pad and hole location accuracy	+/- 0.120 mm
5	Copper pad to pad location accuracy	+/- 0.120 mm
6	Solder resist opening registration accuracy	+/- 0.125 mm
7	Minimum punching hole diameter :- FR-4	ø 2.00 mm
	Minimum punching hole diameter :- CEM-1	ø 0.70 mm
	Minimum punching hole diameter :- CEM-3	ø 0.75 mm
	Minimum punching hole diameter :- XPC , FR-1 , FR-2 (Paper Phenolic)	ø 0.60 mm
	Minimum punching hole diameter tolerance	+ 0.10 mm / - 0 mm
8	Minimum punching slot size :- FR-4	1.0 x 2.00 mm
	Minimum punching slot size :- CEM-1 / CEM-3	1.0 x 1.50 mm
	Minimum punching slot size :- XPC , FR-1 , FR-2 (Paper Phenolic)	0.8 x 1.00 mm
9	Minimum CNC hole drilling diameter	0.35 mm
	Minimum CNC slot drill diameter	0.80 X 1.70 mm
	CNC hole drilling diameter tolerance	+/- 0.05 mm
10	Minimum workable CCL base material thickness with UL approval	0.80 mm
11	Hole edge to board edge distance	≥ 1.20 mm
12	Hole edge to V-cut line distance	≥ 1.60 mm
13	Hole edge to hole edge distance	≥ 1.00 mm
14	Minimum solder resist opening clearance from copper pad / land	≥ 0.15 mm
15	Minimum symbol opening clearance from copper pad / land	≥ 0.15 mm
16	Nickel plating thickness (electroless / electrolytic)	2 µm ~ 5 µm
17	Gold plating thickness (electroless / electrolytic)	0.025 µm ~ 0.05 µm
18	Electrical test SMD pad pitch distance	> 0.40 mm
19	Maximum numbers of electrical test point	6,144 points

Single Sided PCB Production & Technical Capabilities

No.	Design Items	Specification / Tolerance
20	Minimum surface copper thickness	17 µm
21	Maximum surface copper thickness	70 µm
22	Minimum pitch distance for punched hole diameter (1.6mm PCB thickness)	Pitch
	Diameter ø 0.60 mm ~ ø 0.90 mm	1.78 mm
	Diameter ø 1.00 mm ~ ø 1.10 mm	2.00 mm
	Diameter ø 1.20 mm ~ ø 1.30 mm	2.30 mm
	Diameter ø 1.40 mm ~ ø 1.50 mm	2.60 mm
	Diameter ø 1.60 mm ~ ø 1.70 mm	2.90 mm
	Diameter ø 1.80 mm ~ ø 1.90 mm	3.20 mm
	Diameter ø 2.00 mm ~ ø 2.10 mm	3.45 mm
	Diameter ø 2.20 mm ~ ø 2.30 mm	3.70 mm
	Diameter ø 2.40 mm ~ ø 2.50 mm	3.95 mm
23	V-cut depth	
	1.60 mm (XPC , FR-1 , FR-2 , CEM-1 , CEM-3) – each side V-cutting	0.35 ~ 0.45 mm
	1.20 mm (XPC , FR-1 , FR-2 , CEM-1 , CEM-3) – each side V-cutting	0.25 ~ 0.35 mm
	1.00 mm (XPC , FR-1 , FR-2 , CEM-1 , CEM-3) – each side V-cutting	0.20 ~ 0.30 mm
	0.80 mm (XPC , FR-1 , FR-2 , CEM-1 , CEM-3) – each side V-cutting	0.15 ~ 0.25 mm
	1.60 mm (FR-4) – each side V-cutting	0.50 ~ 0.70 mm
	1.20 mm (FR-4) – each side V-cutting	0.40 ~ 0.50 mm
	1.00 mm (FR-4) – each side V-cutting	0.30 ~ 0.40 mm
	0.80 mm (FR-4) – each side V-cutting	0.20 ~ 0.30 mm

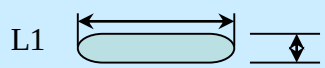
GUH CIRCUIT INDUSTRY (PG) SDN BHD (199201000239 (231743-T))

Double Sided and Multilayer PCB Production Capabilities



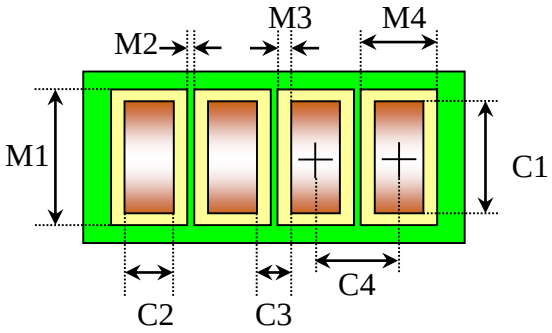
Location	Design (min)	Tolerance	Description
A	0.075 mm	+/- 0.025 mm	Minimum conductor width
B	0.075 mm	+/- 0.025 mm	Minimum gap between copper conductor
C	0.125 mm	+/- 0.05 mm	Minimum gap between copper pad ring & conductor
D	0.25 mm	+/- 0.05 mm	Minimum copper ring (component hole)
E	1.20 mm	+/- 0.08 mm	Minimum copper pad diameter (component hole)
F	0.20 mm	+/- 0.05 mm	Minimum gap between copper pad (component hole)
G	1.40 mm	+/- 0.08 mm	Minimum pitch distance for component hole
H	0.70 mm	+/- 0.05 mm	Minimum diameter for component hole
I	0.70 mm	+/- 0.05 mm	Minimum distance from hole edge to hole edge
J	0.10 mm	+/- 0.03 mm	Minimum gap between copper pad ring & conductor
K	0.10 mm	+/- 0.03 mm	Minimum copper ring (via hole)
L	0.20 mm	+/- 0.03 mm	Minimum gap between copper pad (via hole)
M	0.25 mm	+/- 0.05 mm	Minimum diameter for via hole
N	0.65 mm	+/- 0.05 mm	Minimum pitch distance for via hole
* P	0.40 mm	+/- 0.05 mm	Minimum distance from via hole edge to via hole edge
Q	0.50 mm	+/- 0.10 mm	Minimum gap between copper pad ring & board edge
R	0.50 mm	+/- 0.08 mm	Minimum gap between conductor & CNC slot edge
S	≥ 0.60 mm	+/- 0.10 mm	CNC drilling hole edge to outline board edge distance
T	≥ 1.50 mm	+/- 0.10 mm	Tooling punch hole edge to outline board edge distance
W1	0.80 mm	+/- 0.08 mm	Minimum slot width (by CNC)
L1	≥ 2 W1 + 0.1mm	+/- 0.08 mm	Minimum slot length (by CNC)
W1	≥ 1.00 mm	+/- 0.10 mm	Minimum slot width (by tooling punch)
L1	≥ 2.30 mm	+/- 0.10 mm	Minimum slot length (by tooling punch)

Slot Hole (FR-4 Material)

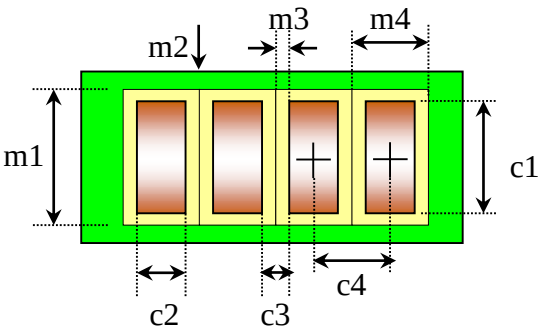


* Subject to application of material selection

Double Sided and Multilayer PCB Production Capabilities



SMD Pad Pitch > 0.60 mm



SMD Pad Pitch = 0.50 mm

SMD Pad Pitch > 0.60 mm

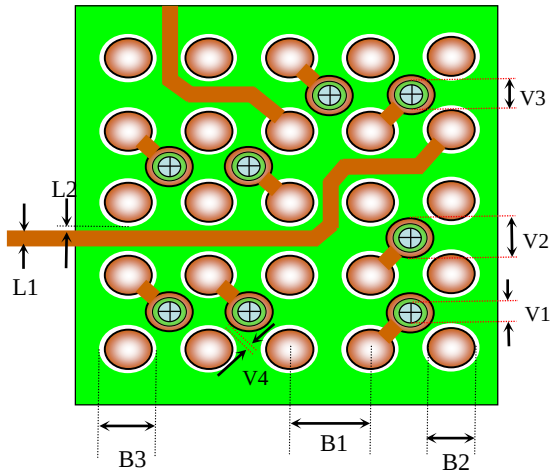
Location	Design	Tolerance	Description
M1	0.80 mm	+/- 0.05 mm	Minimum length of solder resist opening
M2	0.10 mm	+/- 0.05 mm	Minimum solder resist width between openings
M3	0.10 mm	+/- 0.05 mm	Minimum solder resist opening clearance from copper land
M4	0.50 mm	+/- 0.05 mm	Minimum solder resist opening width
C1	0.60 mm	+/- 0.05 mm	Minimum SMD pad length
C2	0.30 mm	+/- 0.05 mm	Minimum SMD pad width
C3	0.30 mm	+/- 0.05 mm	Minimum gap in between SMD pads
C4	0.60 mm	+/- 0.05 mm	Minimum SMD pitch

SMD Pad Pitch = 0.50 mm

m1	0.80 mm	+/- 0.05 mm	Minimum length of solder resist opening
m2	NIL	NIL	No solder resist in between
m3	0.10 mm	+/- 0.05 mm	Minimum solder resist opening clearance from copper land
m4	0.50 mm	+/- 0.05 mm	Minimum solder resist opening width
c1	0.60 mm	+/- 0.05 mm	Minimum SMD pad length
c2	0.30 mm	+/- 0.05 mm	Minimum SMD pad width
c3	0.20 mm	+/- 0.05 mm	Minimum gap in between SMD pads
c4	0.50 mm	+/- 0.05 mm	Minimum SMD pitch

The above production capabilities serves as a general guideline and is subject to further confirmation.

Double Sided and Multilayer PCB Production Capabilities

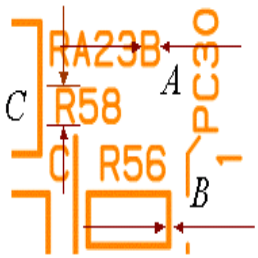


BGA Pad

BGA Pad

Location	BGA	BGA	BGA	Tolerance	Description
B1	0.60mm Pitch	0.70mm Pitch	0.80mm Pitch	+/- 0.05 mm	Pitch between BGA & BGA pad
B2	0.30 mm	0.35 mm	0.40 mm	+/- 0.05 mm	BGA pad diameter
B3	0.40 mm	0.45 mm	0.50 mm	+/- 0.05 mm	BGA solder resist opening diameter
V1	0.25 mm	0.25 mm	0.25 mm	+/- 0.05 mm	Minimum diameter for via hole
V2	0.35 mm	0.40 mm	0.45 mm	+/- 0.03 mm	Via hole copper pad diameter
V3	0.30 mm	0.30 mm	0.30 mm	+/- 0.03 mm	Via hole solder resist opening
V4	0.075 mm	0.075 mm	0.075 mm	+/- 0.03 mm	Minimum gap between BGA resist opening & via copper land
L1	0.10 mm	0.10 mm	0.10 mm	+/- 0.05 mm	Minimum conductor width
L2	0.05 mm	0.075 mm	0.10 mm	+/- 0.05 mm	Minimum gap between conductor & BGA resist opening

Symbol Printing Capability



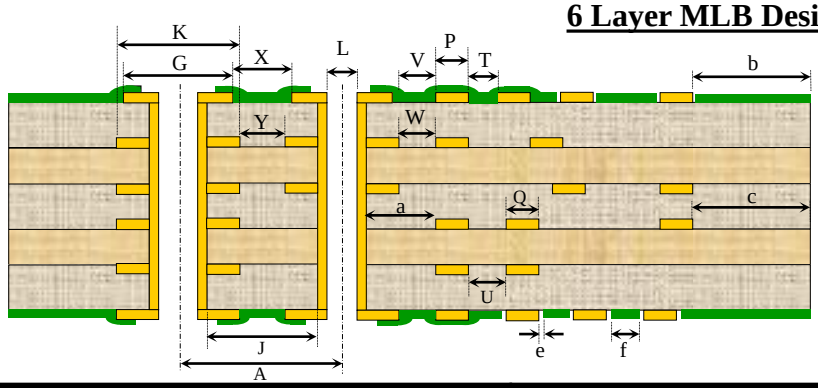
Minimum A ≥ 0.80 mm
Minimum B ≥ 0.15 mm
Minimum C ≥ 1.00 mm
Minimum D ≥ 0.15 mm

D= Symbol width

The above production capabilities serves as a general guideline and is subject to further confirmation.

Multilayer PCB Technical Capabilities

6 Layer MLB Design Standard Explanatory



	Mark	Description	Location Points	Minimum Design Value (mm)
TH Through Hole	**A**	Distance from centre of TH to centre of TH	Between TH centre & TH centre	0.70 mm
	J	Distance from wall to wall of TH *	Spaces between TH walls	0.45 mm
	G	Through hole land / pad diameter	Outer layer land / pad	0.55 mm
	K		Inner layer land / pad	0.60 mm
	L	Through hole diameter	Through hole diameter after copper plated	0.25 mm
Pattern Design	**P**	Pattern track width	Outer layer pattern	0.075 mm
	Q		Inner layer pattern	0.10 mm
	T	Spacing between pattern track	Outer layer track gap	0.075 mm
	U		Inner layer track gap	0.10 mm
	V	Spacing between pattern track and pad	Outer layer pattern track & pad gap	0.10 mm
	W		Inner layer pattern track & pad gap	0.10 mm
	X	Spacing between pad & pad	Outer layer copper pad gap	0.15 m m
	Y		Inner layer copper pad gap	0.10 mm
	a	Spacing between inner layer pattern track & TH wall	Inner Layer	0.35 mm
Outline	**b**	Spacing between pattern track & PCB outline	Outer layer pattern track	0.50 mm
	c		Inner layer	0.50 mm
	d	Spacing between TH wall & PCB outline	TH (Outline by CNC routing)	0.65 mm
			TH (Outline punch by tooling)	1.60 mm
Solder Resist	**e**	Solder resist clearance (one side)		0.05 mm
	f	Minimum solder resist width	Solder mask slit	0.10 mm
		Solder resist misregisteration tolerance		+/- 0.05 mm

* Subject to application of material selection

The above production capabilities serves as a general guideline and is subject to further confirmation.