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

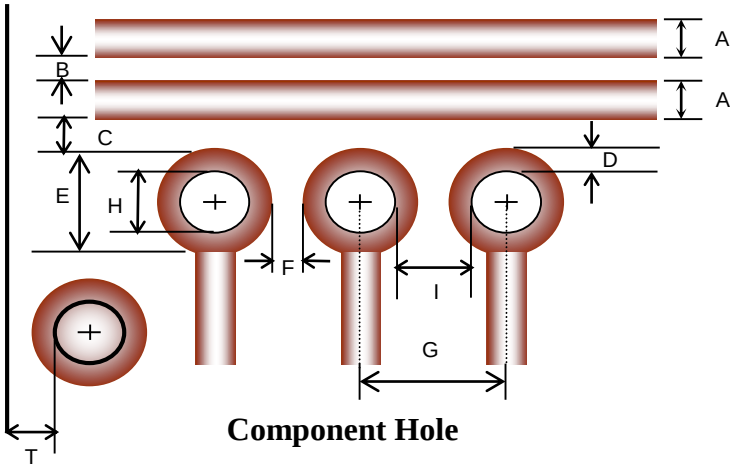
Single Sided PCB Production & Technical Capabilities

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	$\geq$ 1.20 mm
12	Hole edge to V-cut line distance	$\geq$ 1.60 mm
13	Hole edge to hole edge distance	$\geq$ 1.00 mm
14	Minimum solder resist opening clearance from copper pad / land	$\geq$ 0.15 mm
15	Minimum symbol opening clearance from copper pad / land	$\geq$ 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

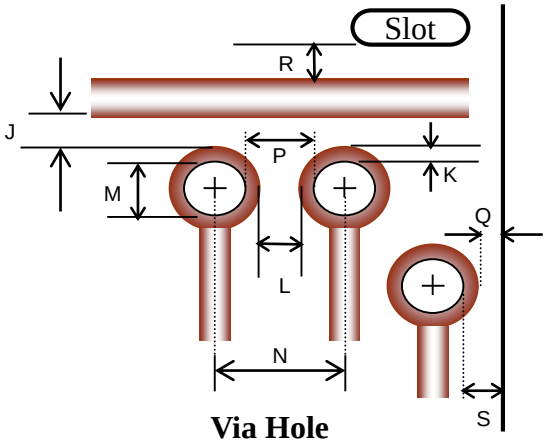
Single Sided PCB Production & Technical Capabilities

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

Double Sided and Multilayer PCB Production Capabilities

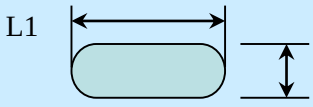


Component Hole



Via Hole

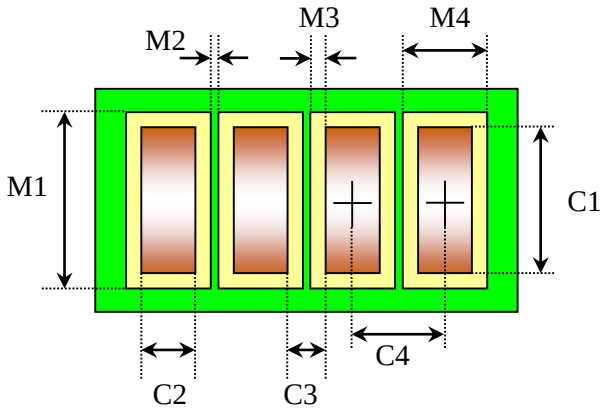
Location	Design (min)	Tolerance	Description
A	0.075 mm	+/- 0.025 mm	Minimum Conductor Width
B	0.075 mm	+/- 0.025 mm	Minimum Gap Between Copper Conductor
C	0.125 mm	+/- 0.05 mm	Minimum Gap Between Copper Pad Ring & Conductor
D	0.25 mm	+/- 0.05 mm	Minimum Copper Ring (Component Hole)
E	1.20 mm	+/- 0.08 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	0.10 mm	+/- 0.03 mm	Minimum Gap Between Copper Pad (Via Hole)
M	0.15 mm	+/- 0.05 mm	Minimum Diameter for Via Hole
N	0.55 mm	+/- 0.05 mm	Minimum pitch distance For via hole
P *	0.30 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)



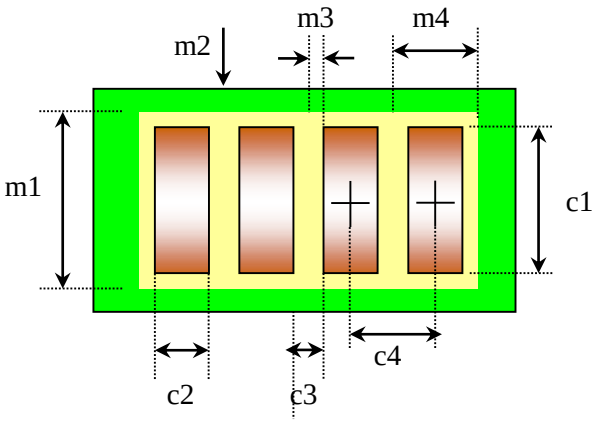
Slot Hole (FR-4 Material)

***** Subjected to the application of material selection

Double Sided and Multilayer PCB : Solder Mask Process Capabilities



SMD Pad Pitch > 0.60 mm



SMD Pad Pitch = 0.50 mm

SMD Pad Pitch > 0.60 mm

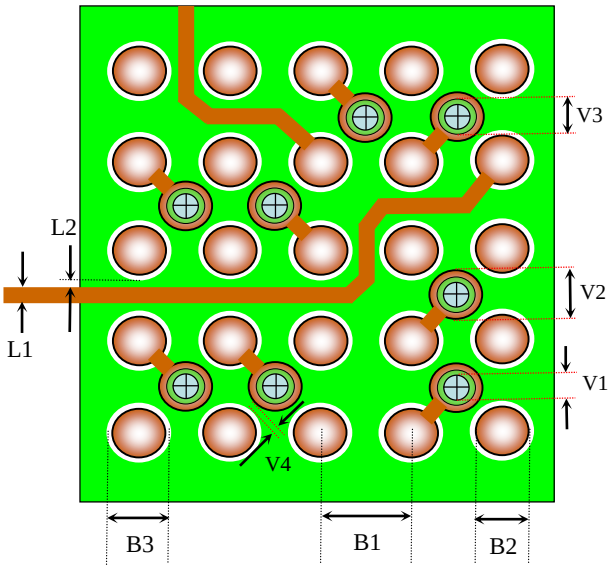
Location	Design	Tolerance	Description
M1	0.80 mm	+/- 0.05 mm	Minimum length of solder resist opening
M2	0.080 mm	+/- 0.05 mm	Minimum solder resist width between openings
M3	0.050 mm	+/- 0.05 mm	Minimum solder resist opening clearance from copper land
M4	0.50 mm	+/- 0.05 mm	Minimum solder resist opening 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

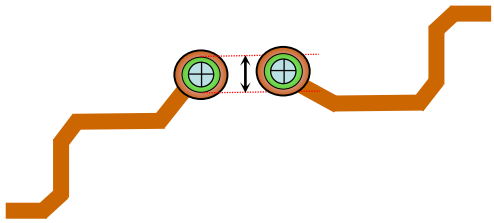
m1	0.80 mm	+/- 0.05 mm	Minimum length of solder resist opening
m2	NIL	NIL	No Solder Resist In Between
m3	0.050 mm	+/- 0.05 mm	Minimum solder resist opening clearance from copper land
m4	0.50 mm	+/- 0.05 mm	Minimum solder resist opening width
c1	0.60 mm	+/- 0.05 mm	Minimum SMD pad length
c2	0.30 mm	+/- 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 serve as general guidelines only, and subject to further negotiation and confirmation.

Double Sided and Multilayer PCB Production Capabilities :



BGA Pad Design

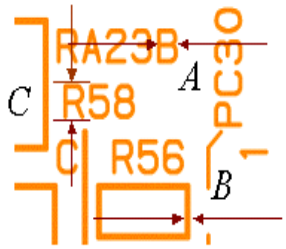


Via Hole Design

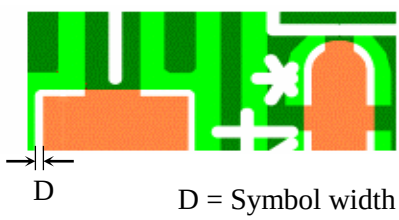
BGA Pad

Location	BGA	BGA	BGA	Tolerance	Description
B1	0.65mm Pitch	0.70mm Pitch	0.80mm Pitch	+/- 0.05 mm	Pitch Between BGA & BGA Pad
B2	0.30 mm	0.30 mm	0.40 mm	+/- 0.05 mm	BGA pad diameter
B3	0.40 mm	0.40 mm	0.50 mm	+/- 0.05 mm	BGA solder resist opening diameter
V1	0.20 mm	0.20 mm	0.25 mm	+/- 0.05 mm	Minimum Diameter for Via Hole
V2	0.40 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	0.06 mm	0.095 mm	0.090 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	0.075 mm	0.10 mm	0.10 mm	+/- 0.05 mm	Minimum Gap between conductor & BGA resist opening

Symbol Printing Capability :-



Minimum A ≥ 0.60 mm
Minimum B ≥ 0.15 mm
Minimum C ≥ 0.80 mm
Minimum D ≥ 0.15 mm

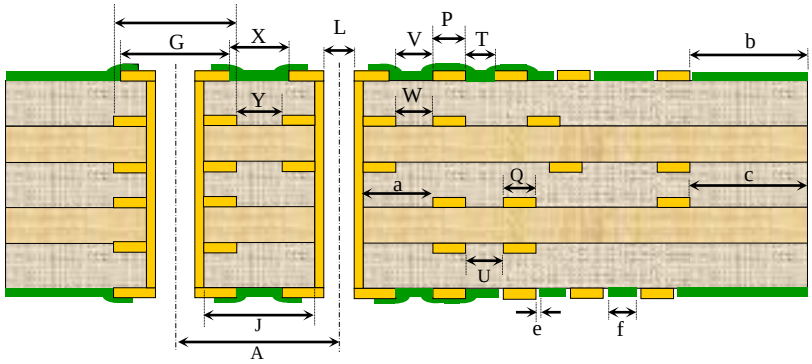


D = Symbol width

The above production capabilities serve as general guidelines only, and subject to further negotiation and 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	0.45 mm
	K		Inner layer land / pad	0.45 mm
	L	Through hole diameter	Through hole diameter after copper plated	0.15 mm
Pattern Design	**P**	Pattern track width	Outer layer pattern	0.075 mm
	Q		Inner layer pattern	0.075 mm
	T	Spacing between pattern track	Outer layer track gap	0.075 mm
	U		Inner layer track gap	0.075 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	0.08 mm
		Solder resist misregisteration tolerance		+/- 0.05 mm

* Subjected to the application of material selection

The above production capabilities serve as general guidelines only, and subject to further negotiation and confirmation.