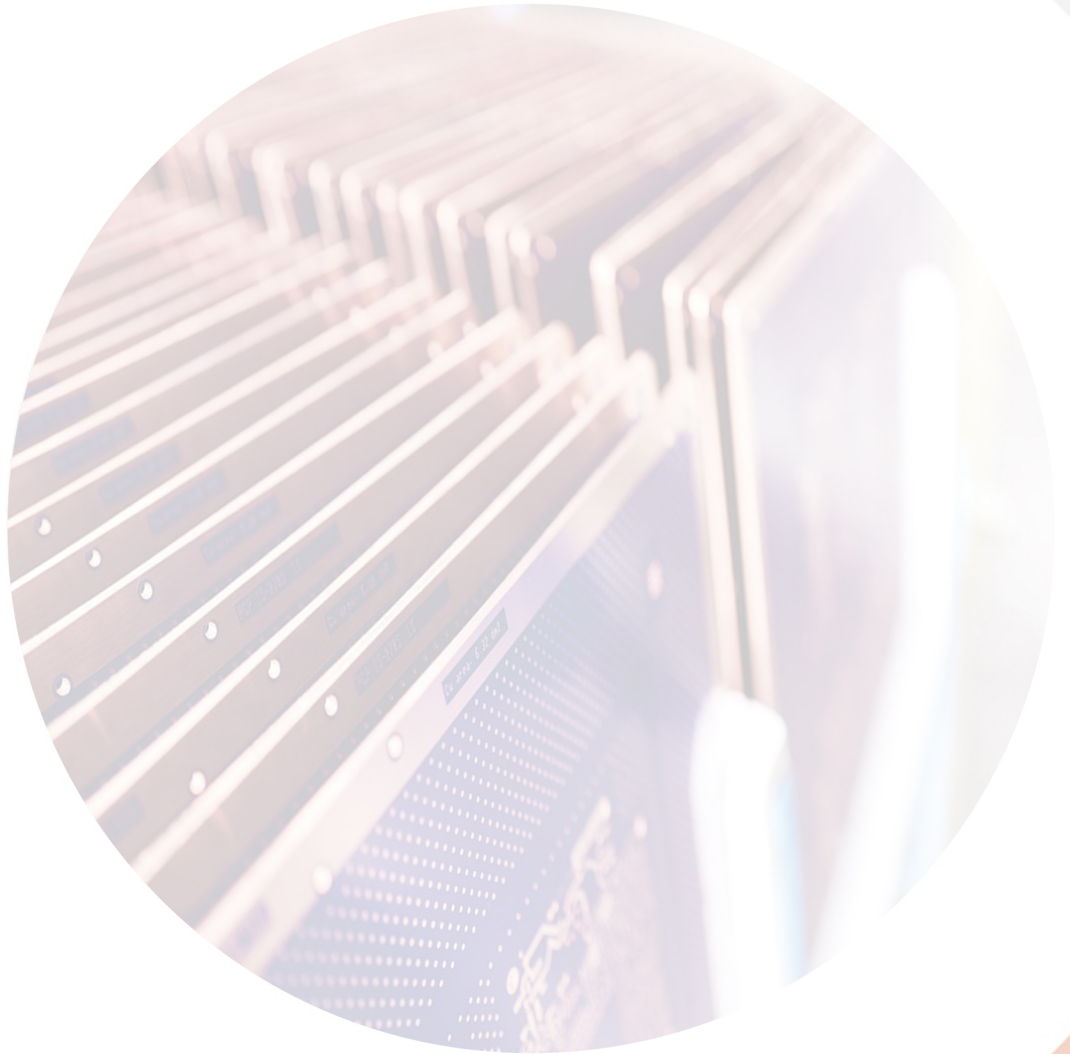




Dear customer,

This document aims to inform pcb designers about PCB design keypoints to achieve best results from PCB production processes. Values mentioned in this document can be improved upon special requests if possible.

Contents

1	Laminate types.....	3
2	Panel sizes.....	3
2.1	Production panel.....	3
2.2	Customer panel.....	3
3	Pcb thickness.....	4
4	Copper thickness.....	4
5	Plated holes and base copper.....	5
6	Solder mask specifications.....	6
6.1	Solder mask specifications.....	6
6.2	Other colours and solder mask thickness.....	6
6.3	Via types.....	7
6.3.1	Tented via.....	7
6.3.2	Plugged via.....	7
7	Surface treatment.....	8
8	Legend print specifications.....	8
9	Mechanical capabilities.....	9
9.1	Drilling (holes and slots).....	9
9.2	Aspect ratio.....	9
9.3	Milling.....	9
9.4	V-cut.....	10
9.5	Countersink.....	10
9.6	Blind and buried	11
10	Multilayer production.....	12
10.1	Standard stack up.....	12
10.2	Impedance control.....	13
11	Carbon ink printing.....	14
12	Peelable soldermask.....	14
13	Design rules.....	15
14	General tolerances.....	16

1.Laminate types

Laminate types	Standard	Advanced
Materials	FR2, FR4, CEM1, CEM3, IMS, PI	Rogers, Taconic, Arlon, Teflon, Ceramic

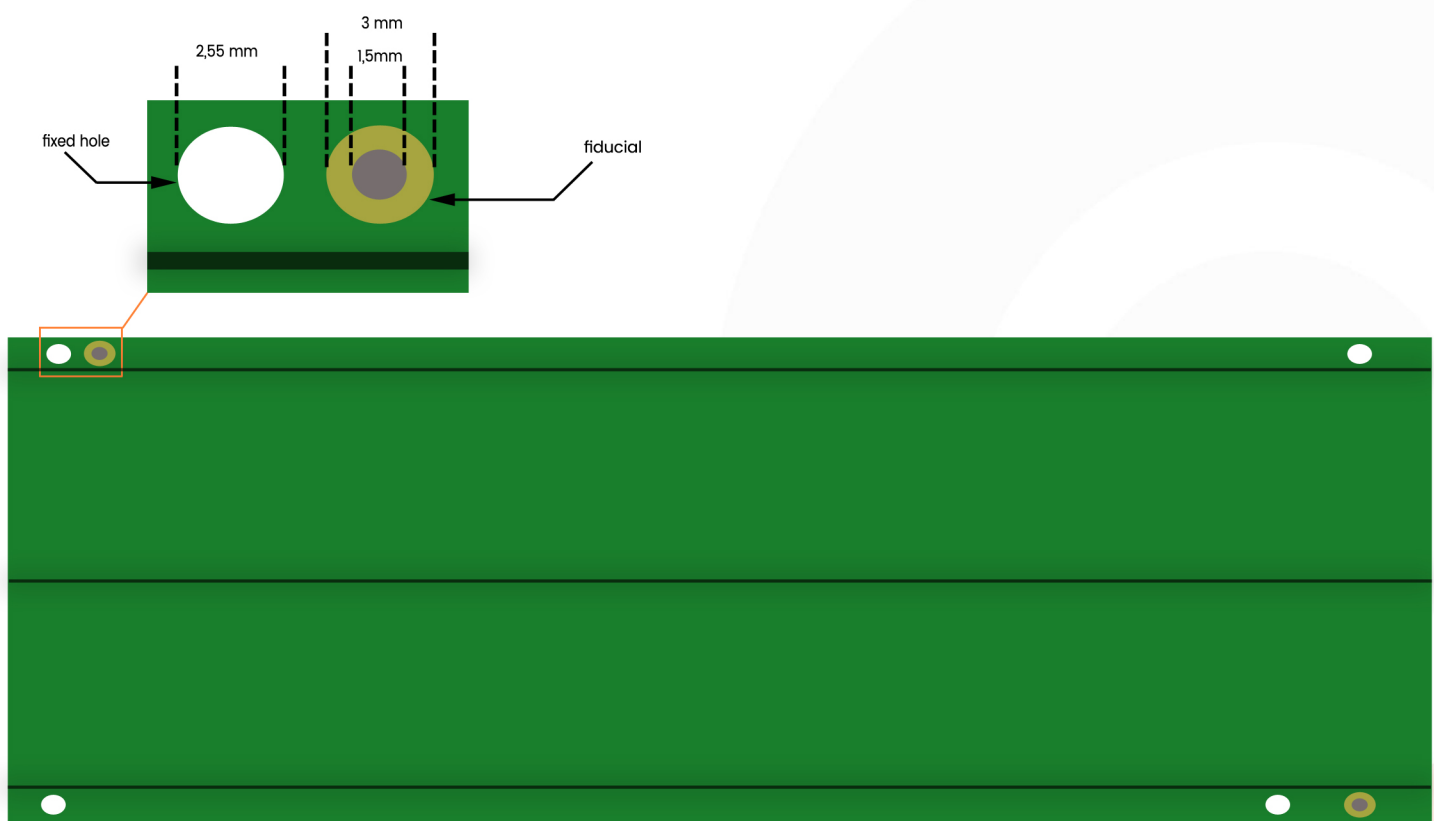
2.Panel sizes

2.1 Production Panel

PCB type	Standard	Advanced
Single side	500 mm x700 mm	600 mm x 1000 mm
Double side	500 mm x600 mm	600 mm x 1000 mm
Multilayer	450 mm x 550 mm	500 mm x 600 mm

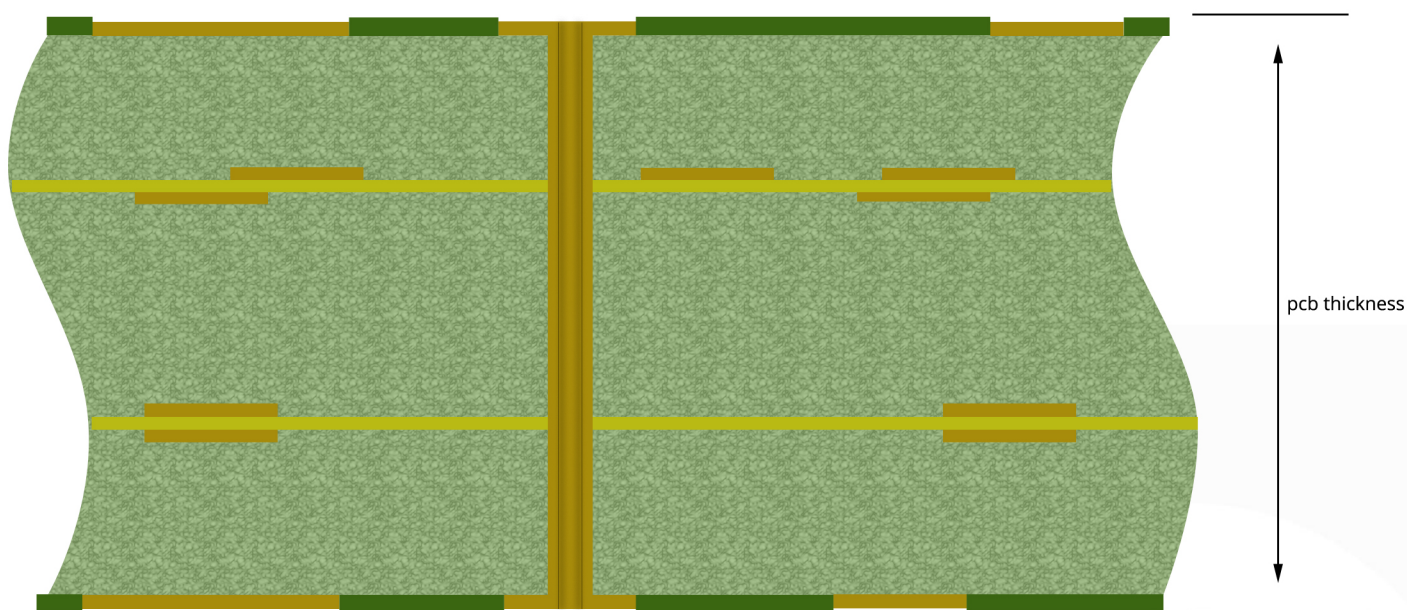
2.2 Customer panel

We generate a customer panel depending whether the pcb has a SMD component or not unless it is specified from you that pcb should be singular pcb shipment. Four fixing holes for outline profiling (2.55mm diameter) and four fiducial points (1.5mm copper and 3mm soldermask aperture)are added to technology borders.



3.PCB Thickness

PCB type	Standard	Advanced
Single side	0.8 mm to 2.4 mm	< 0.8 mm and > 2.4 mm
Single side (IMS)	1 mm to 1.6 mm	< 1 mm and > 1.6 mm
Double side	0.8 mm to 2.4 mm	< 0.8 mm and > 2.4 mm
Multilayer	0.8 mm to 2 mm	< 0.8 mm and > 2 mm

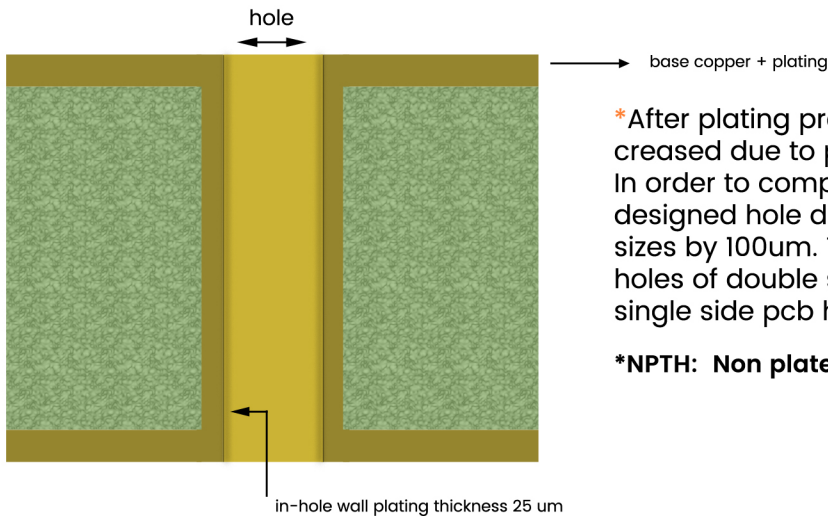


4.Copper Thickness

PCB type	Standard	Advanced
Single side	35 μ m , 70 μ m (1 oz , 2 oz)	> 70 μ m (> 2 oz)
Double side	35 μ m , 70 μ m , 105 μ m (1 oz , 2 oz , 3oz)	> 105 μ m (> 3 oz)
Multilayer	18 μ m , 35 μ m , 70 μ m (1/2 oz , 1 oz , 2 oz)	> 70 μ m and 12 μ m (> 2 oz and 1/3 oz)

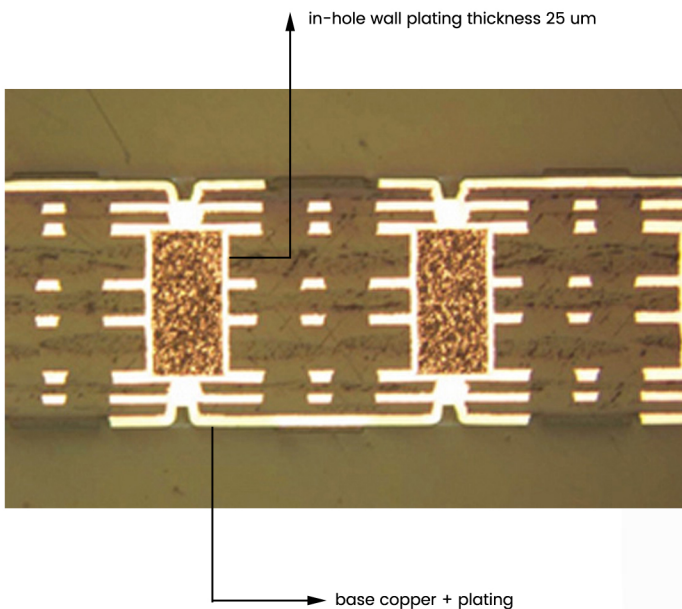
* Surface copper thickness for 2 oz. orders will be measured between 50-55 μ m and for 3 oz. orders will be between 85-90 μ m.

5.Plated through holes (PTH)



*After plating process , drilled hole diameters are decreased due to plated copper thickness inside the holes. In order to compensate this reduction and to achieve the designed hole diameter value, we increase the hole drill sizes by 100um. This application is only applied for PTH holes of double side and multilayer pcbs. NPTH holes and single side pcb hole diameters are not changed.

***NPTH: Non plated through holes**



6.Solder mask specifications

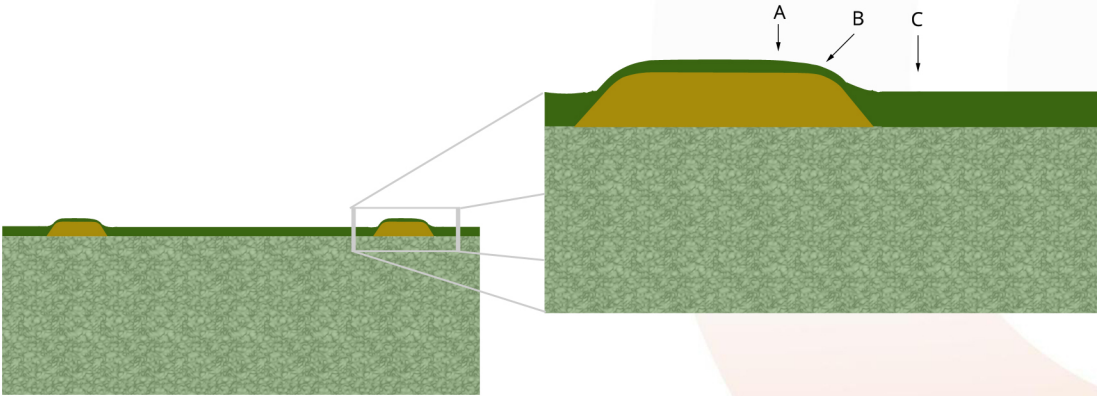
6.1 Colour Scale For White Led PCB's

2000 K6500 K

Warm white	White	Cold white
Taiyo - psr 4000 lew3	Taiyo - WT-02	KSM - S6189EWT34
KSM - S6189EWT44-SF LEW3 Alt.	KSM - S6189EWT45 WT-02 Alt.	
Goo -PSR-290EXW WS-2		

6.2 Other colours and soldermask thicknesses

Feature	Standard	On demand
Solder mask colour	green, white, black, blue, red,	yellow , brown , purple
Solder mask thickness	A over tracks between 10µm - 30µm	-
	B track edges ≥ 8µm	-
	C between tracks ≥ 8µm	-



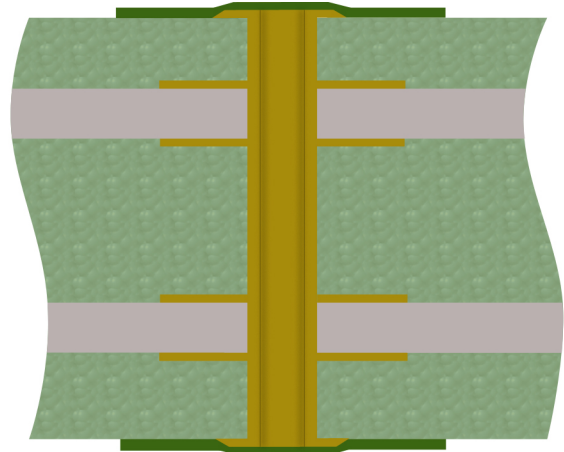
6.3 Via types

6.3.1 Tented via

The tenting via method is being applied on vias with solder resist ink, and covers the via surfaces

This method does not require additional processing or fees, but 100% results cannot be guaranteed.

Via Type I

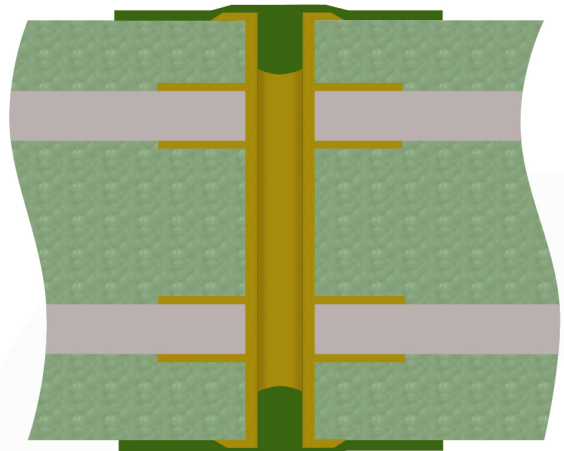


6.3.2 Plugged via

The plugged via method is being applied on vias with solder resist ink, and plugged with the solder mask ink in via holes.

With this method, via holes are not completely plugged.

Via Type III



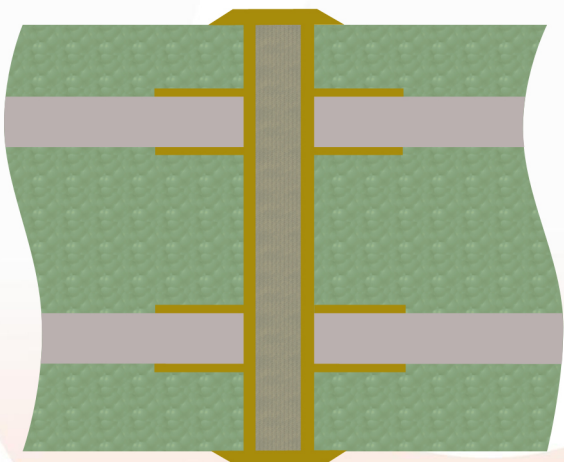
6.3.3 Filled and capped via

The filled via method is being applied in vias with resin filled.

Via Type VI

Via filled with resin, plate with copper. This method named capped.

Via type VII

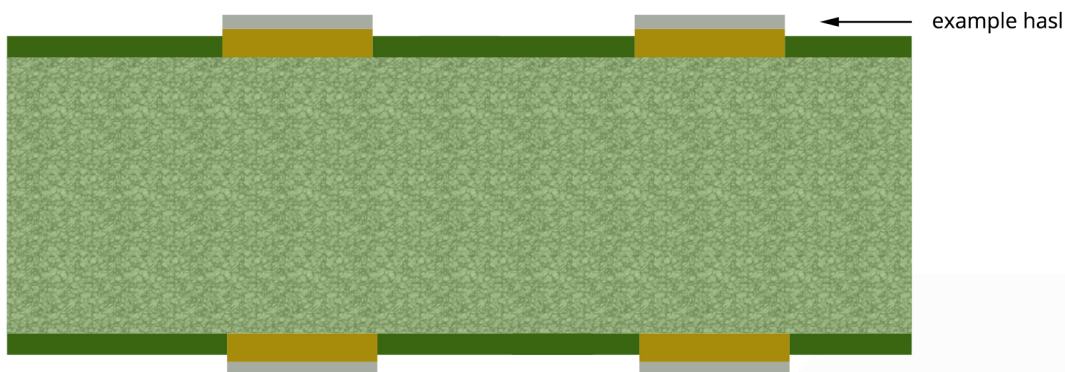


7.Surface treatment

7.1 Finishing type

PCB type	Standard	Advanced
Single side	Lead-free HASL, 63/37 HASL , OSP, ENIG	ENEPIG, Immersion Silver, Immersion Tin, Hard Gold
Double side		
Multilayer		

7.2 Finishing specifications



Solder surface	Osp	Hasl	Immersion Tin	Enig
Thickness	0.2- 0.6 μm	Covered to 40 μm	0.8 – 1.2 μm	3 – 6 μm Ni ; 0.05 – 0.13 μm Au
Shelf life	6 months	12 months	12 months	12 months
Excellent flatness	Yes	-	Yes	Yes
Cost	Low	Medium	High	High
Solderability	✓	✓✓✓	✓✓	✓✓

8.Silk screen specifications

Feature	Standard	On demand
Silk screen colour	white, black , blue	yellow , orange , grey
Silk screen thickness	5 - 15 μm	-

9. Mechanical capabilities

9.1 Drilling (Holes and Slots)

PCB type	Standard	Advanced
Single side	0.7 mm hole 0.7 mm slot	0.5 mm hole 0.5 mm slot
IMS	1.5 mm hole 2 mm slot	1.2 mm hole 1.5 mm slot
Double side	0.3 mm hole 0.7 mm slot	0.2 mm hole 0.5 mm slot
Multilayer	0.3 mm hole 0.7 mm slot	0.2 mm hole 0.5 mm slot

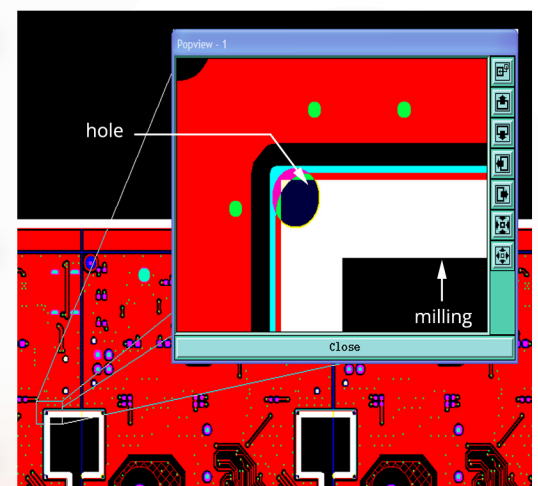
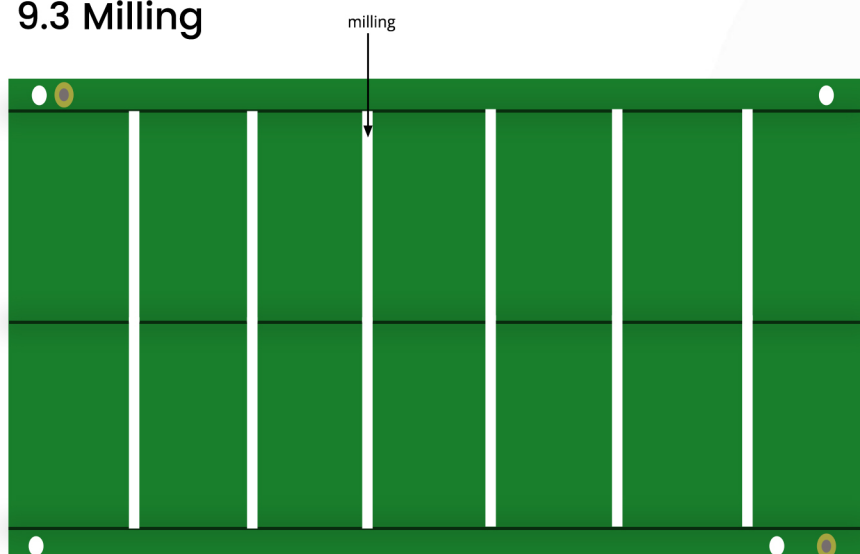
9.2 Aspect Ratio

$$\text{Aspect Ratio} = \frac{\text{Laminate thickness}}{\text{Drill diameter}} = \frac{1.6 \text{ mm}}{0.2 \text{ mm}} = 8$$

$$\text{Aspect Ratio} = \frac{\text{Laminate thickness}}{\text{Drill diameter}} = \frac{2 \text{ mm}}{0.25 \text{ mm}} = 8$$

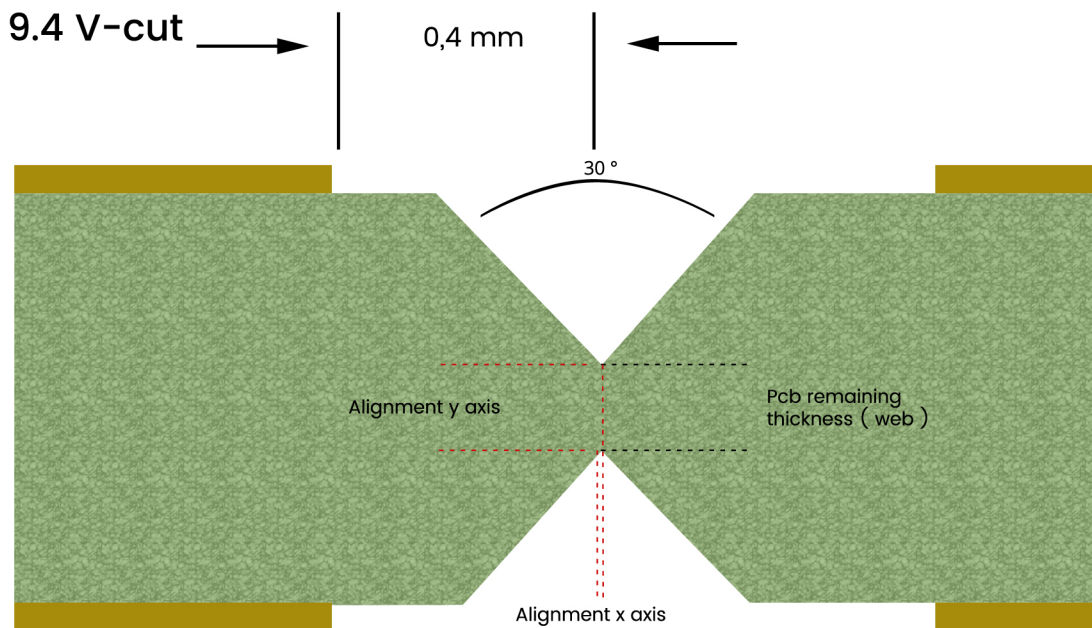
* Aspect ratio shows the ratio between minimum holes size and maximum laminate thickness that can be plated.

9.3 Milling



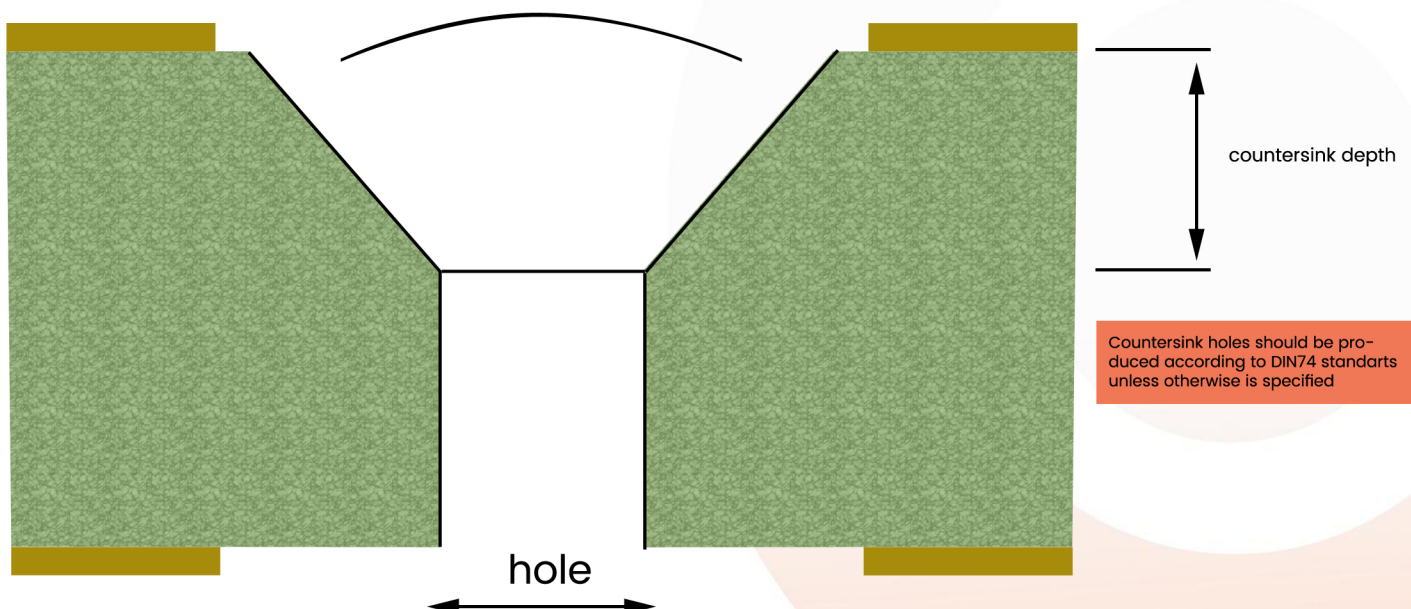
Feature	Standard	Advanced
Milling tools	1.5 mm to 2 mm	1 mm to 1.5 mm

* We add small holes to corners to achieve the best results for the cuttings that are 90 degrees and can not be achieved by standard routing bit diameters.



Feature	Standard		Advanced
Board thickness	0.6 mm to 2.4 mm		> 2.4 mm
V-cut angle	30 ° ± 5 °		40 ° ± 5 °
Alignment (x axis)	± 0.1 mm		
Alignment (y axis)	± 0.1 mm		
Min distance to copper	> 0.4 mm		
Jump v-cut	Yes		
Pcb remaining thickness (web)	CEM1	0.6 mm +/- 0.1 mm	
	FR4	0.4 mm +/- 0.1 mm	
	IMS	0.3 mm +/- 0.1 mm	

9.5 Countersink

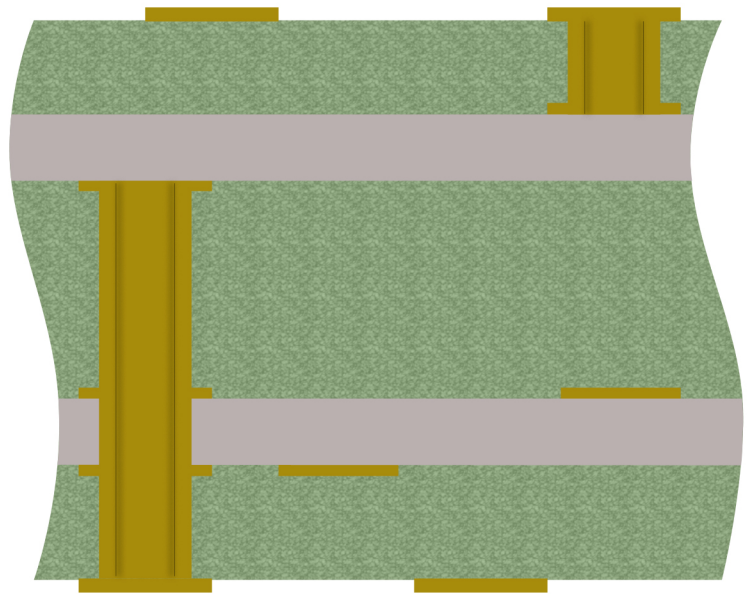


9.6 Blind and Buried vias

Blind and Buried Vias add considerably to the cost of a PCB. They should only be used when absolutely necessary.

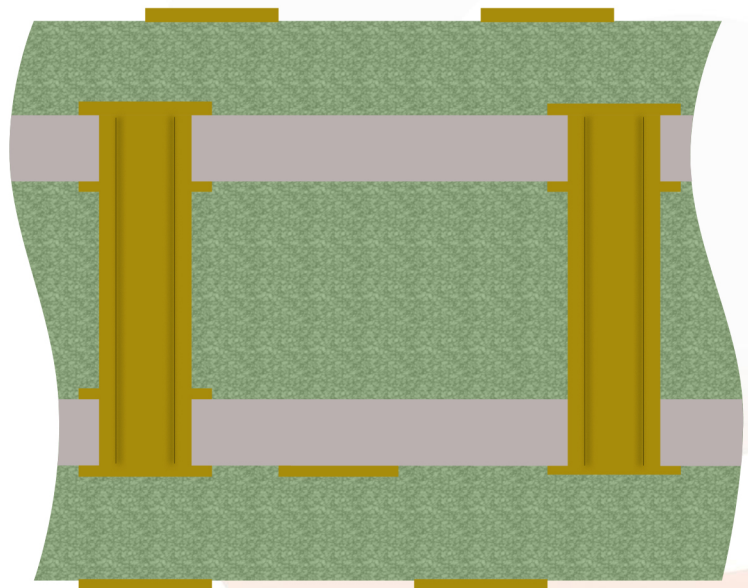
9.6.1 Blind via

Blind vias are used to connect one of the outer layer with at least the one inner layer. It should be within aspect ratio.



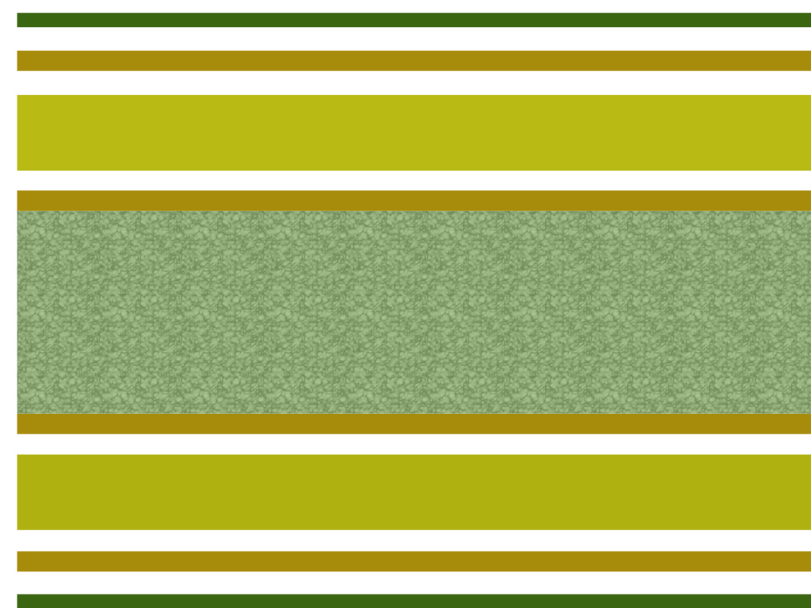
9.6.2 Buried via

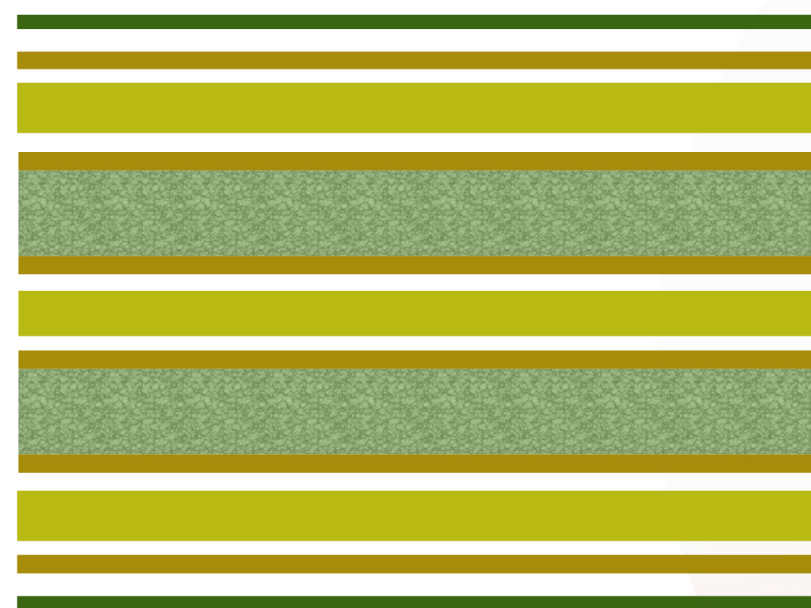
Buried vias are used to connect the inner layers, which have no contact with the outer layers. It should be within aspect ratio.



10.Multilayer production

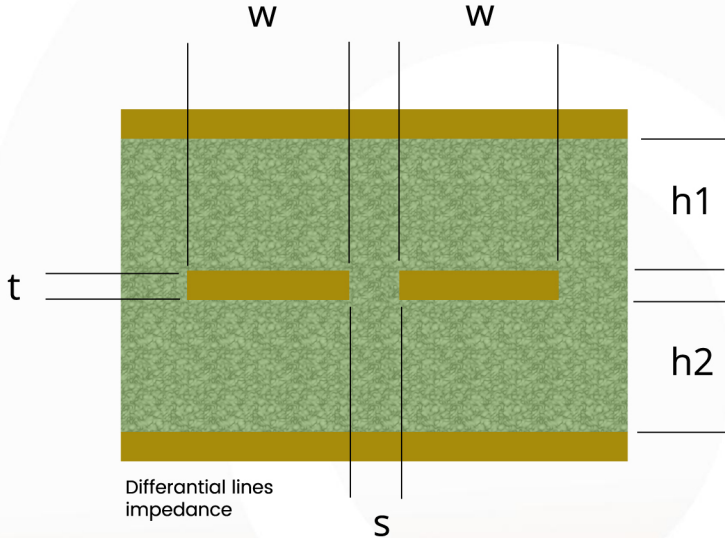
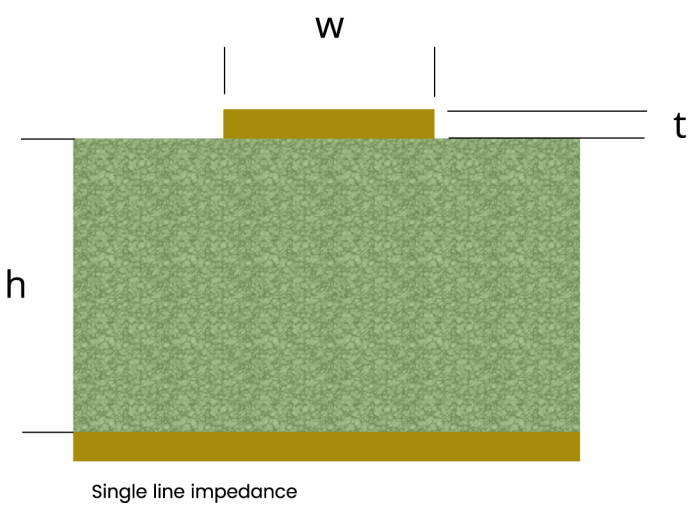
10.1 Standard stack up

	Layers	Thickness
	Top solder mask	18 µm
	Top side 18µm+plating	35 µm
	Prepreg 1x7628	180 µm
	Inner layer 1	35 µm
	Core	1,1 mm
	Inner layer 2	35 µm
	Prepreg 1x7628	180 µm
	Bot side 18µm+plating	35 µm
	Bot solder mask	18 µm
4 layer stack up	Total pcb thickness	1,6 mm

	Layers	Thickness
	Top solder mask	18 µm
	Top side 18µm+plating	35 µm
	Prepreg 1x7628	180 µm
	Inner layer 1	35 µm
	Core	0,36 mm
	Inner layer 2	35 µm
	Prepreg 1x7628	180 µm
	Inner layer 3	35 µm
	Core	0,36 mm
	Inner layer 4	35 µm
6 layer stack up	Prepreg 1x7628	180 µm
	Bot side 18µm+plating	35 µm
	Bot solder mask	18 µm
	Total pcb thickness	1,6 mm

Layers	Thickness
Top solder mask	18 µm
Top side 18um+plating	35 µm
Prepreg 2x1080	160 µm
Inner layer 1	35 µm
Core	0,25 mm
Inner layer 2	35 µm
Prepreg 2x1080	160 µm
Inner layer 3	35 µm
Core	0,25 mm
Inner layer 4	35 µm
Prepreg 2x1080	160 µm
Inner layer 5	35 µm
Core	0,25 mm
Inner layer 6	35 µm
Prepreg 2x1080	160 µm
Bot side 18um+plating	35 µm
Bot solder mask	18 µm
8 layer stack up	
Total pcb thickness :	1.60 mm

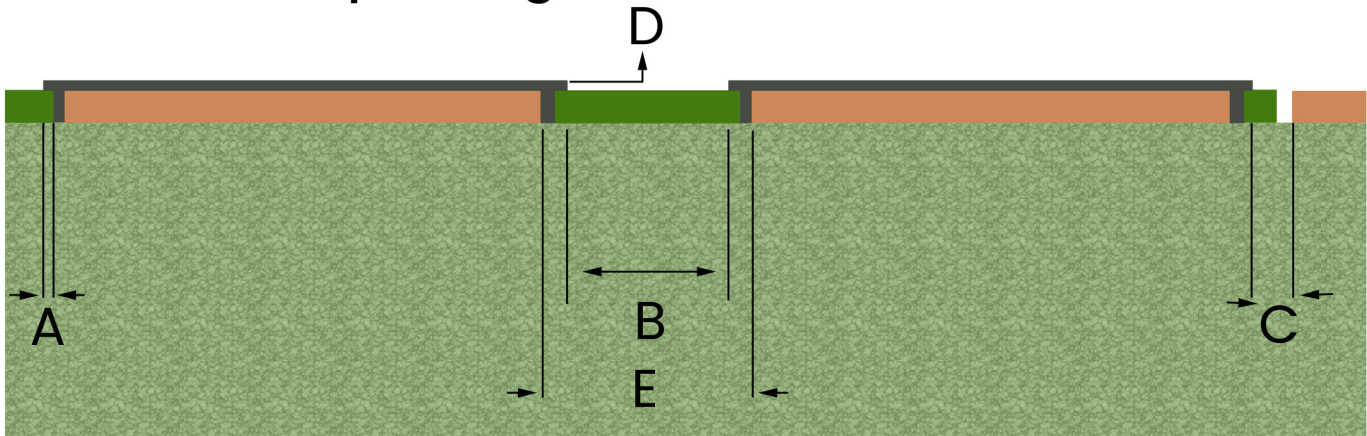
10.2 Impedance control



w: Line width
s: Space
h: Height
t: Copper thickness

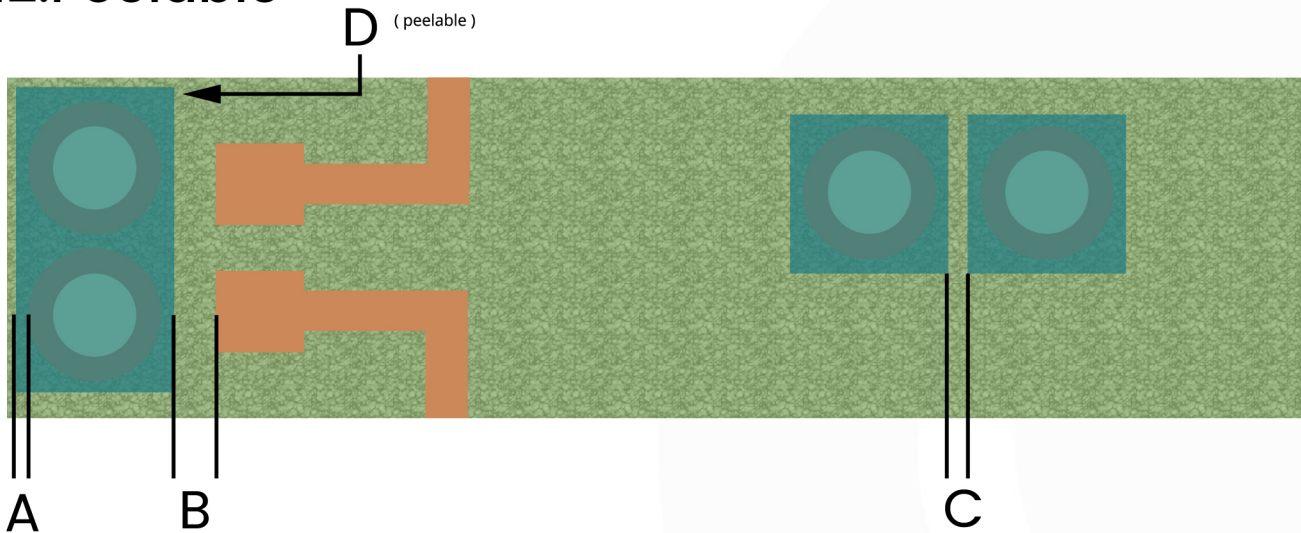
- Other factors that effect impedance control
- Pcb thickness
 - Dielectric values of dielectric materials
 - Dielectric value of solder mask

11. Carbon ink printing



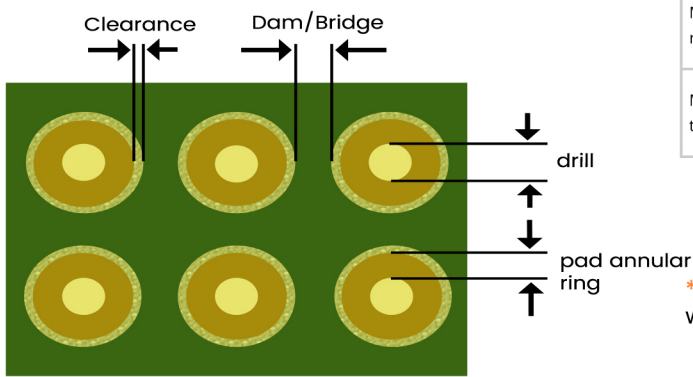
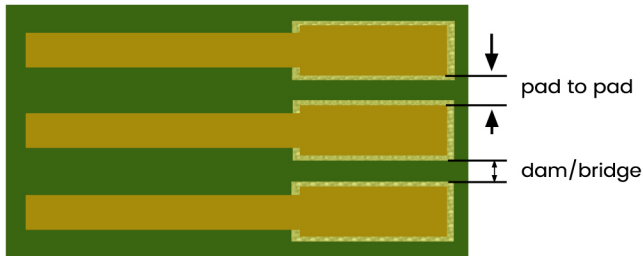
Feature	Standard	Advanced
A (carbon overflow distance)	250 μm	200 μm
B (carbon to carbon)	500 μm	400 μm
C (carbon to copper)	500 μm	400 μm
D (carbon thickness)	5 μm - 10 μm	-
E (copper to copper)	1000 μm	800 μm

12. Peelable



Feature	Standard	Advanced
A (peelable overflow distance)	400 μm	300 μm
B (peelable to pad)	500 μm	400 μm
C (peelable to peelable)	500 μm	400 μm
D (peelable thickness)	500 μm - 1000 μm	-

13.Design rules



Signal layers specifications

Specifications	Single side		Double side		Multilayer	
	Exposure	Screen printing	Standard	Advanced	Standard	Advanced
Min line width	0.2 mm (8 mil)	0.2 mm (8 mil)	0.15 mm (6 mil)	0.10 mm (4 mil)	0.15 mm (6 mil)	0.10 mm (4 mil)
Min copper to copper	0.2 mm (8 mil)	0.4 mm (16 mil)	0.15 mm (6 mil)	0.125 mm (5 mil)	0.15 mm (6 mil)	0.10 mm (5 mil)
Min annular ring	0.25 mm (10 mil)	0.25 mm (10 mil)	0.2 mm (8 mil)	0.15 mm (6 mil)	0.2 mm (8 mil)	0.10 mm (6 mil)
Min pad to pad	0.25 mm (10 mil)	0.5 mm (20 mil)	0.15 mm (6 mil)	0.125 mm (5 mil)	0.15 mm (6 mil)	0.125 mm (5 mil)

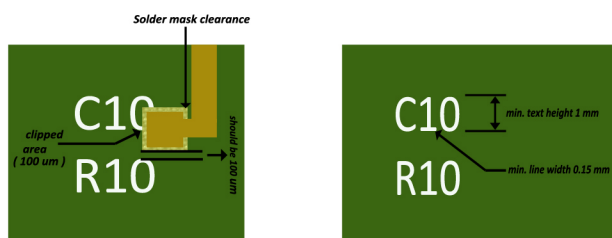
* The distances above are based on 35 um copper thickness and they will increase accordingly when the copper thickness increases.

Solder mask layers specifications

Specifications	Single side		Double side		Multilayer	
	Exposure	Screen printing	Standard	Advanced	Standard	Advanced
Min solder mask dam	100 µm (4 mil)	200 µm (8 mil)	100 µm (4 mil)	70 µm (3 mil)	100 µm (4 mil)	70 µm (3 mil)
Min solder mask clearance	100 µm (4 mil)	150 µm (6 mil)	100 µm (4 mil)	70 µm (3 mil)	100 µm (4 mil)	70 µm (3 mil)

When screen printing method is applied:

- * In order to create soldermask dam/bridge, "copper to copper" distance should be min. 0,5mm.
- * Soldermask clearance should be min 0,15mm.
- * The width of soldermask dam/bridge should be min 0,2mm.



Legend layers specifications

Specifications	Single side		Double side		Multilayer	
	Exposure	Screen printing	Standard	Advanced	Standard	Advanced
Min legend print / legend space	0.15/0.15mm (6mil)	0.15/0.15mm (6mil)	0.15/0.15mm (6mil)	0.10/0.10mm (4mil)	0.15/0.15mm (6mil)	0.10/0.10mm (4mil)

14.Tolerances

Tolerances	Values	
Plated through hole	± 0,075 mm	
Non plated through hole	± 0,05 mm	
Drill positioning	100 µm	
Cu min thickness through plating	class II 20 µm	class III 25 µm
Line width	± 20 %	
Solder resist positioning	± 0,1 mm	
Silk screen positioning	± 0,1 mm	
Registration of innerlayer to innerlayer	± 0,05 mm	
Image to image	± 0,075 mm	
Alignment x axis	± 0,1 mm	
Alignment y axis	± 0,1 mm	
Pcb remaining thickness (web)	± 0,1 mm	
V-cut depth	± 0,15 mm	
Dimension created by v-cut (after seperation per each v-cut)	FR4	± 0,2 mm
	CEM1	± 0,3 mm
	IMS	± 0,15 mm
Milling offset	± 0,1 mm	
Milling tolerance	± 0,2 mm	
Punching tolerance	± 0,2 mm	
Pcb thickness	± %10	
Impedance control	± %10	