Manufacturing Standards

The following manufacturing standards apply to JETPCB.com PCB manufacture

1. Specifications

Items	General specifications	Other specifications	REMARK
Material	FR-4（TG-140）	FR-4（TG-170） CEM-3	CEM-3 only for single/double layer
Board thickness	1.6mm	0.5~3.5mm	Aspect ratio should below 1:8 when the thickness of board is over 1.6mm
Layer counts	2~6 layer	8~14 layer	
Surface Finishing	HASL Immersion gold (ENIG) Gold finger plating	Gold plating OSP Immersion Silver Immersion Tin SMOBC	Finished dimension should larger than 90*55mm when the surface finishing is OSP or Immersion Silver or Immersion Tin
Thickness of HASL	100~1000μ”		
Thickness of immersion gold	Au：2~5μ” Ni：100μ”~150μ”		
Thickness of gold-plating	Au：3~5μ” Ni：100μ”~200μ”	Au：6μ”~30μ” Ni：100μ”~200μ”	The min. trace gap should greater than 5 mil on outer layer if overall gold plating is applied
OSP specifications	F2 LX Thickness 8~20μ”		Finished dimension must be larger than 90*55mm.
Thickness of immersion silver	4~12μ”		Finished dimension must be larger than 90*55mm.
Thickness of Immersion tin	20~30μ”	40μ”	Finished dimension must be larger than 150*100mm
Solder mask colors	Green	Yellow, red, blue, black, white	
Solder mask thickness	More than 0.4mil	More than 0.7mil	

Legend color	White	Yellow, black	
Maximum finished dimension	500mm*400mm		
Board accuracy	±0.25mm	±0.1mm	
Minimum hole diameter	±0.3mm（12mil）	±0.2mm（8mil）	The aspect ratio is below 1：8.
Minimum PAD diameter	0.5mm（20mil）	±0.4mm（16mil）	
Minimum trace width/space	trace width：5mil trace space：5mil	trace width：4mil trace space：4mil	4/4mil : only applied when finished copper thickness below 35μm.
BGA diameter	More than 14mil	8~12mil	8~12mil : only applied when finished copper thickness below 35μm.
SMD solder dam	More than 3mil		
Inner-layer copper thickness	0.5 oz：18μm 1.0 oz：35μm	2 oz：70μm	
Outer-layer copper thickness	1.0 oz：35μm	2 oz：70μm	

2. Notes

Here are things you need to be cautious of when using or storing printed circuit boards:

A. Avoid water if the boards are to be placed on the ground.
B. Warranty is one month after shipping date.
C. Avoid the place exposure to strong sunlight, high temperature and moisture, etc.
D. The best temperature to store the boards is below 30 degrees C and the moisture below 60%.
E. If the boards are stored for a longer period of time, and then to be used at 120 degrees C, please dehumidify for one hour first.
F. When handling, be sure to wear gloves and prevent damaging the peripheral corners.
G. When designing the width of trace, please allow the flowing current value at 0.2A and control the temperature within an upwards of 10 degrees.
H. Do not wash the boards with acid solvent to avoid soldering damage.

I. If the boards are to be scrapped, please handle in accordance to the industrial waste disposal law.

3.Manufacturing Standards

3.1 Material

(1) The boards

- Heatproof epoxy resin. (FR-4)
- The inner layer is glass fiber.
- No material factory is appointed for the boards.

(2) The copper foil

- The purity of the copper foil is more than 99.5%.
- No material factory is appointed for the boards.

3.2 Copper plating

- The average thickness is more than 20μm, the minimum copper-plating thickness is more than 15μm for via, trace and gold finger.
- The quality and purity of copper is 99.5%.

3.3 The surface treatment

- No copper is allowed to expose at the bottom.
- As for HASL and solder, the color of copper seen on part of the board means that the solder-resistant ink is sprayed a little bit thinner.
- Lead free surface treatment includes HASL lead free, immersion gold, OSP, gold plating, immersion silver, and immersion tin, etc.

3.4 The finished dimension

(1) The tolerance of the finished dimension

- The general tolerance is 0.25mm.
- The minimum tolerance is 0.1mm.

(2) Routing shape

- The finished board is routed by routing machine.
- If the route is not a right angle of 90º, and the R angle is 0.5~1.0mm,and if there is a circuit within 1mm, the R angle cannot be fabricated.
- The distance must be at least 2mm when cut.
- Some special shaping may not be fabricated.

3.5 Stackup

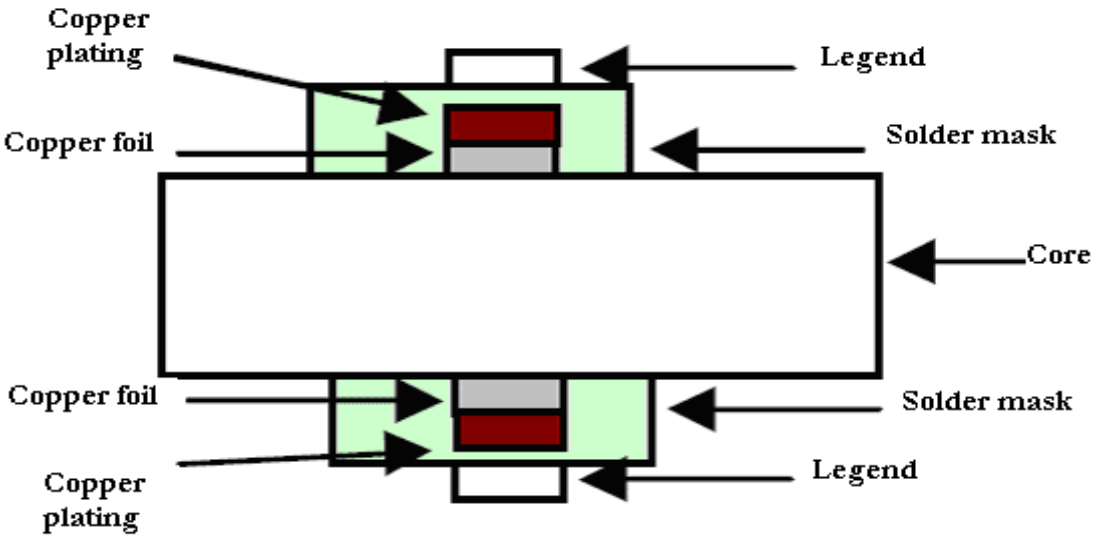
(1) The Tolerance of Board Thickness

T < 1.0mm：±15%

1.0<T<1.6mm：±0.15mm

T>1.6mm：±10%

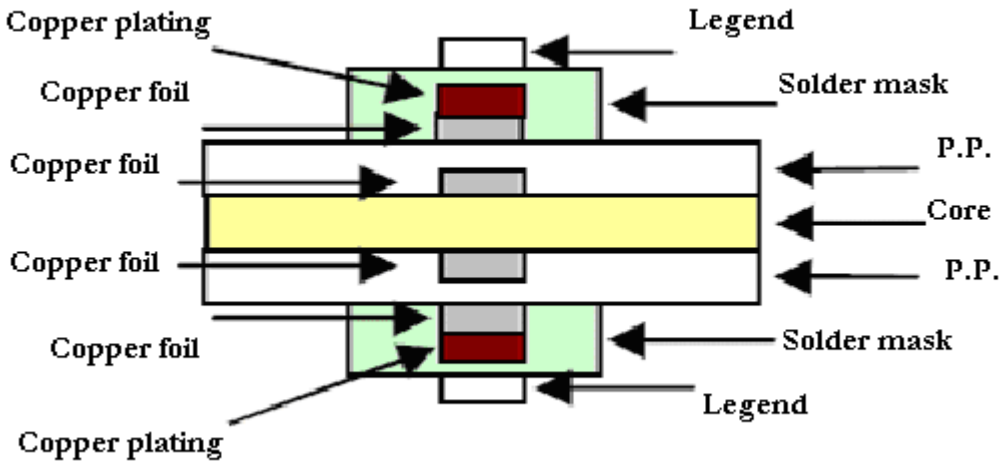
(2) Stackup of 2-layer board



Board thickness	1.6	1.2	1.0	0.8
Solder mask	0.010			
Copper plating	0.025			
Copper foil（L1）	0.018			
Core	1.5	1.07	0.9	0.7
Copper foil（L2）	0.018			
Copper plating	0.025			
Solder mask	0.010			

Unit: mm

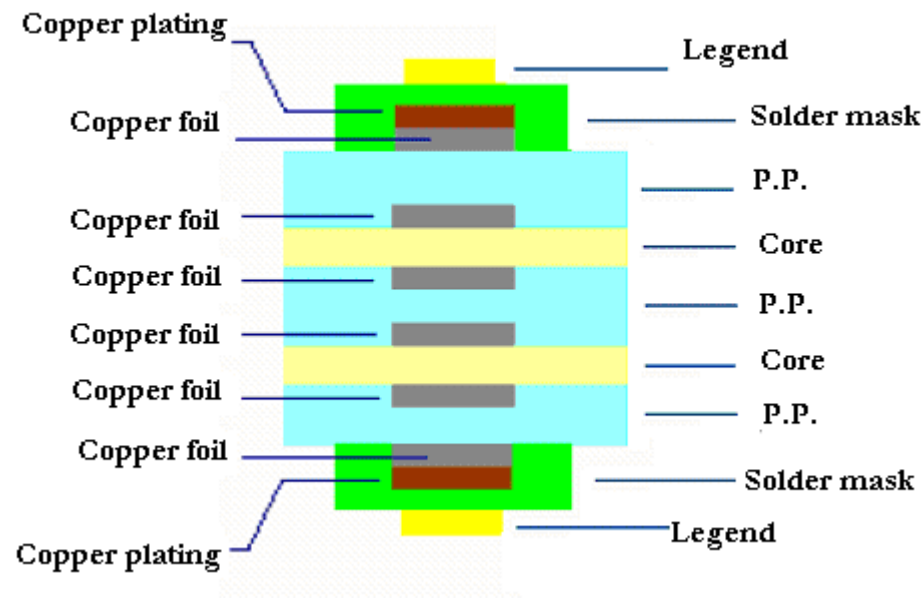
(3) Stackup of 4-layer board



Board thickness	1.6	1.2	1.0	0.8
Solder mask	0.010			
Copper plating	0.025			
Copper foil（L1）	0.018			
P.P.	0.23	0.15	0.23	0.13
Copper foil（L2）	0.035			
Core	0.93	0.7	0.38	0.38
Copper foil（L3）	0.035			
P.P.	0.23	0.15	0.23	0.13
Copper foil（L4）	0.018			
Copper plating	0.025			
Solder mask	0.010			

Unit: mm

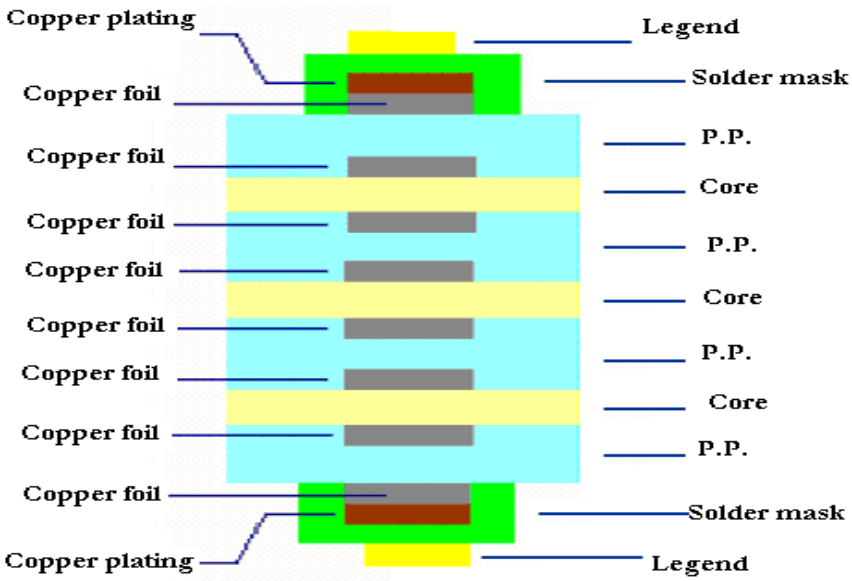
(4) Stackup of 6-layer board



Board thickness	1.6	1.2	1.0	0.8
Solder mask	0.010			
Copper plating	0.025			
Copper foil（L1）	0.018			
P.P.	0.23	0.13	0.1	0.1
Copper foil（L2）	0.035			
Core	0.38	0.25	0.2	0.13
Copper foil（L3）	0.035			
P.P.	0.15	0.2	0.15	0.13
Copper foil（L4）	0.035			
Core	0.38	0.25	0.2	0.13
Copper foil (L5）	0.035			
P.P.	0.23	0.13	0.1	0.1
Copper foil（L6）	0.018			
Copper plating	0.025			
Solder mask	0.010			

Unit：mm

(5) Stackup of 8-layer board

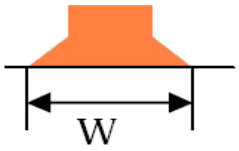


Board thickness	1.6	1.2	1.0	0.8
Solder mask	0.010			
Copper plating	0.025			
Copper foil（L1）	0.018			
P.P.	0.15	0.1	0.1	0.05
Copper foil（L2）	0.035			
Core	0.25	0.15	0.1	0.08
Copper foil（L3）	0.035			
P.P.	0.15	0.13	0.1	0.1
Copper foil（L4）	0.035			
Core	0.25	0.15	0.1	0.08
Copper foil (L5）	0.035			
P.P.	0.15	0.13	0.1	0.1
Copper foil（L6）	0.035			
Core	0.25	0.15	0.1	0.08
Copper foil（L7）	0.035			
Core	0.15	0.1	0.1	0.05
Copper foil（L8）	0.018			
Copper plating	0.025			
Solder mask	0.010			

unit：mm

3.6 Trace Width

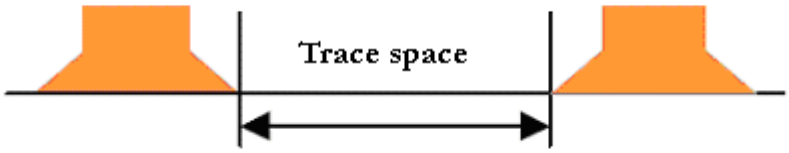
- The minimum trace width：5mil（0.127mm）
- The minimum trace width calculation as shown below.
- The tolerance of trace width.



3.7 Trace Space

(1) The minimum conductor space（including the distance between trace and trace, the distance between the space and PAD, and the distance between PAD and PAD）

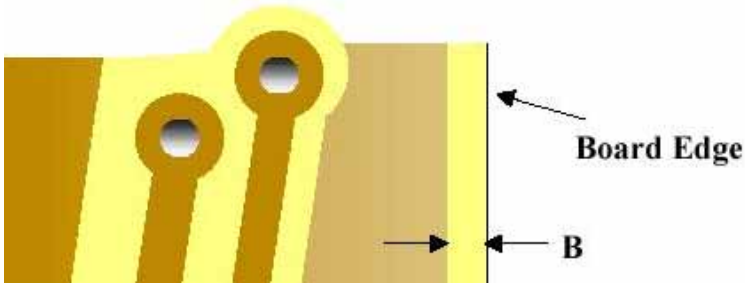
The minimum space：5mil（0.127mm）



(2) The Distance Between Conductor and the Board Edge

The distance between conductor and the finished board edge:

B≧20mil（0.5mm）

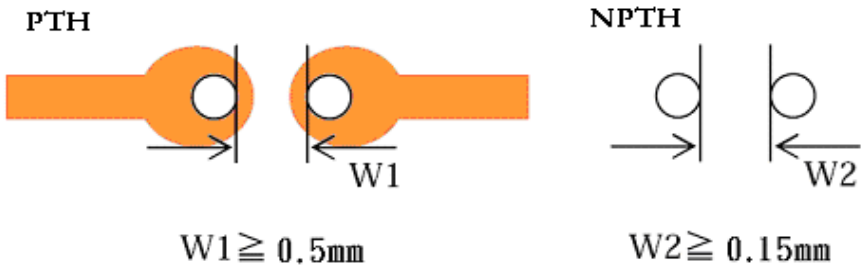


3.8 The Specifications of Hole Diameter and PAD

(1) Tolerance of Hole

Hole Diameter	0.2~0.4mm (via)	0.4~6.0mm	More than 6.0mm
Tolerance of NPTH		±0.05mm	±0.15mm
Tolerance of PTH	＋0.05mm －0.15mm	±0.08mm	±0.1mm
PAD size	Hole Diameter＋ 0.25mm	Hole Diameter＋ 0.3mm	Hole Diameter＋ 0.5mm

(2) The Space Between the Holes



3.9 The Accuracy of Drilling

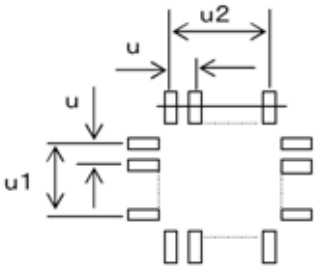
Items	True position tolerance between the holes	Central position tolerance between the hole and the conductor
First-time hole processing	±0.08mm	±0.1mm
Second-time hole processing	±0.1mm	±0.15mm

Second-time hole processing is limited to the NPTH hole, and applied when the special production procedure is required.

3.10 SMD Solder Padding

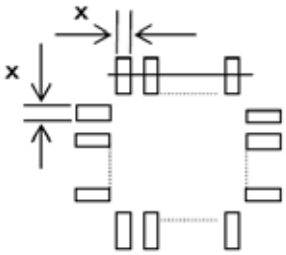
(1) SMD Pitch

SMD Pitch	Minimum distance	Tolerance
U	0.254mm	±0.03mm
U1、U2		±0.05mm



(2) Conductor Width of SMD

SMD Width (X)	Tolerance
0.127~0.35mm	±0.03mm
X＞0.35mm	±0.05mm



3.11 Solder mask Specifications

(1) Color Categorization

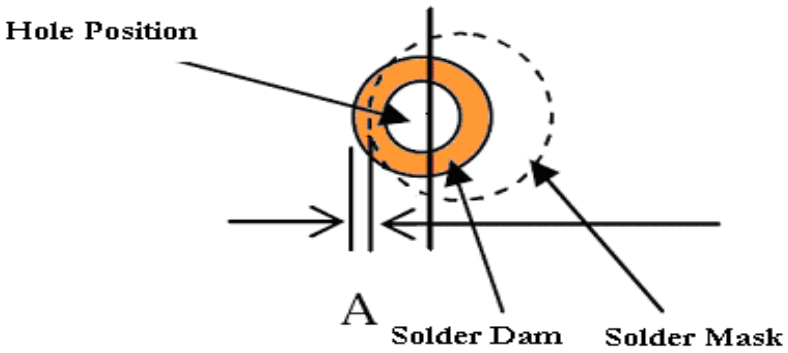
* The standard color is green. The printing side is manufactured according to the client's instructions.

(2) Basic Conditions

* The Solder mask ink shall not be applied unevenly, or come peeling. And, no bubble shall squeeze into the space between the conductors.

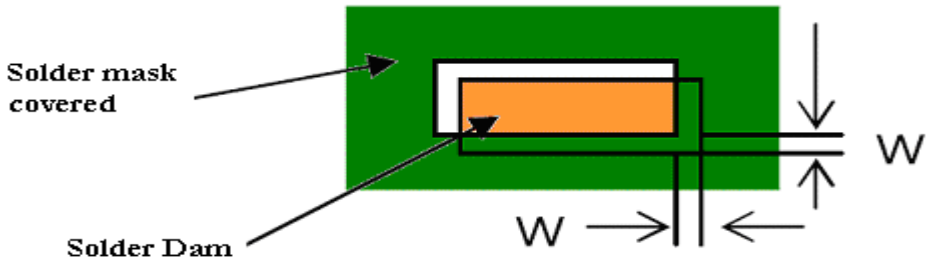
(3) Spread Criteria

A. Solder-resistance of component hole



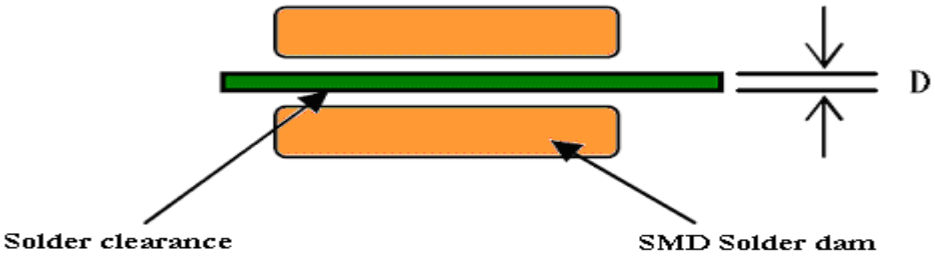
- A≦0.05mm
- The trace shall not show up.
- No solder mask shall drip into the component holes.
- Solder mask allows in the via.

B. SMD Solder Mask



- W≦0.05mm
- SMD solder dam shall denude 90% of uncovered green solder mask.

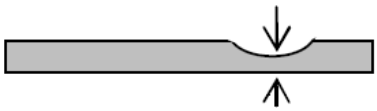
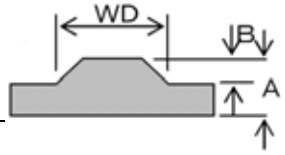
C. SMD Solder Dam

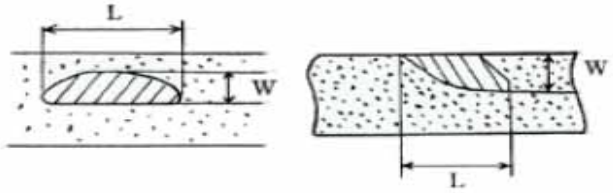


- ※ D≧0.1mm

3.12 Conductor Damaged

The permitted area of the trace damaged

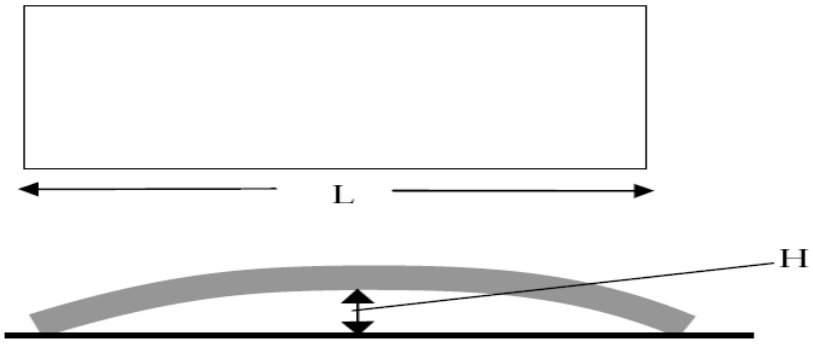
Items	Judgment Criteria
Short, Trace broken	Not allowed to happen.
Trace dent	The minimum conductor width without positioning holes and loop is more than 2/3 of design value. 
Trace protrude	When WD＞A On the principle：B ＝ 0.1 x less than A When WD＜A 

	On the principle：B ＝ 0.2 x less than A
Conductor damaged	The damaged part on the conductor below 5mm wide（deficiency, gaps, and positioning holes, etc.）must be within 1/3 of the conductor width. The damaged part, the length L, shall not exceed the width of the conductor. W≦the conductor width x 1/3 L≦the conductor width 

3.13 Bow and twist

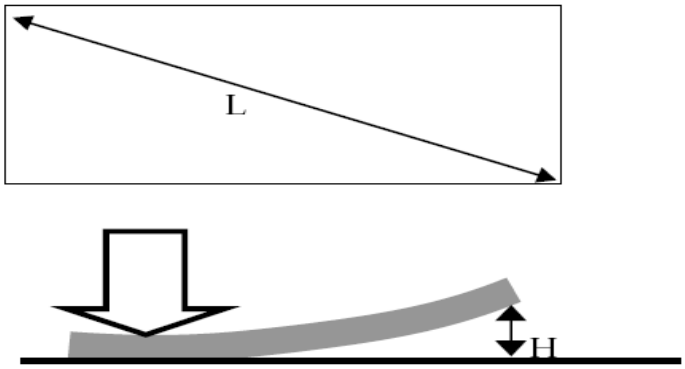
- If the board is a slender shape with a big difference of aspect ratio, or if the board is multi-layered, and has a great one-sided area of copper foil. The following specifications shall not be applied.
- The convex side of the board is placed upwards on the routing machine, the distance between the machine and the board is H. The longer direction of the board is L. The following standards must be met.

	FR-4	CEM-3
0<L<300	H/L≦1.0%	H/L≦1.5%
L≧300	H/L≦1.5%	H/L≦2.0%



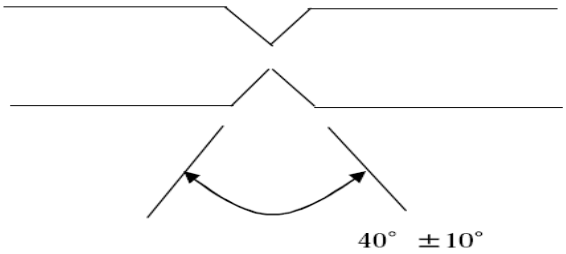
- When one side of the board is placed on the machine, or the length of the point-to-point distance measuring the diagonal line is L, the following standards must be met.

	FR-4	CEM-3
0<L<300	H/L≦1.0%	H/L≦1.5%
L≧300	H/L≦1.5%	H/L≦2.0%

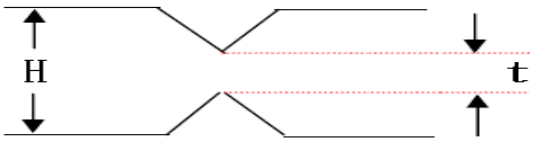


3.14 V-CUT

(1) Angle



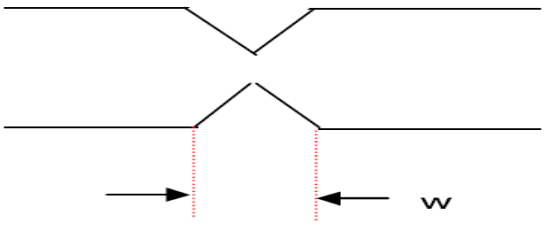
(2) Depth



Board thickness（H）	Web thickness specifications（t）
Below 0.8mm	T=Hx1/2
0.8~1.6mm	T=Hx1/3

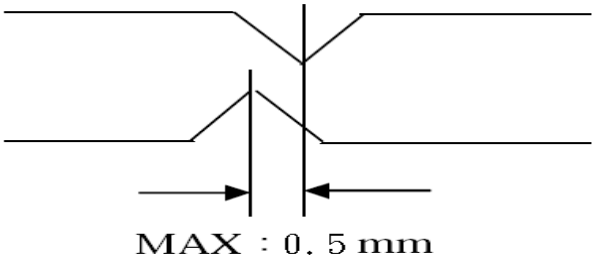
H＞1.6mm	T=0.4~0.6mm

(3) Width

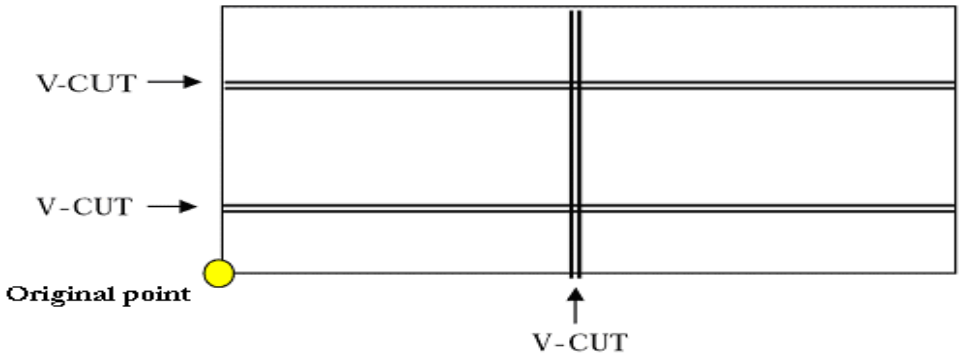


- The board thickness：0.8~1.6mm
  W=0.5±0.1mm

(4) V-CUT Deviation



(5) Position Accuracy



- With the origin as a criterion
  Below 100mm→±0.2mm
  More than 100mm→an increase of 50mm for the size is an increase of 0.1mm

3.15 Silkscreen

(1) Color: Customers may choose from white, yellow, black, red, green and gray

(2) The silkscreen font is defined by the customer's GERBER design.

(3) Silkscreen is processed after soldermask process
(4) The minimum width of the legend is 0.127mm（5mil）, and the min. height is 1.0mm; Smaller legend might become illegible.
(5) The tolerance of legend position is ±0.25mm.

4.16 Final Inspections

Final inspections includes checks on the exterior and exterior size, as well as the followings:

(1) Board quality, thickness, dimension, and quantity of order… etc.
(2) No bulges on the board surface.
(3) No other conductors unless agreed upon by customers.
(4) No copper shown, swollen, peeling, etc.
(5) No breaks on the board of tab & route.
(6) No other substance sticked on the board.
(7) No silkscreen or solder mask are allowed to drip into the holes of the components.
(8) Illegible legend marks are not allowed.
(9) No slitting and cutting. Any slitting, cutting and breaking on a board thickness of 1/2 is acceptable if it doesn't affect the loop.
(10) Damage, discoloring, scratching, deformations on the PAD shall not affect the exterior appearance.
(11) No appearance shall be affected if the printing positions of the soldermask slope to cover the pad, or the damaged parts of solder mask discolor.
(12) No board discolorations or crazing are allowed to affect the appearance.
(13) Any single measling is acceptable. But, it shall not enlarge after heated. Any sequential measlings are unacceptable.
(14) The foreign inclusions within 0.5mm is acceptable if it doesn't concern the circuitry. But, no dirty substance is allowed on the board to affect the appearance.

4. In-Process Inspections

(1) Films
- A visual inspection has to be done on all developed films.

- To find out if there are any open traces, shorts, trace thinness, symmetry, and scratches on the films.
- After inspection, the films must be protected in a plastic bag.

(2) Inspection on inner layers after exposure（not applied to a double-layer board）

- Visual Inspection is to be carried out after the dry-ink exposure process.
- To find out if there are any open traces, shorts, foreign inclusions, symmetry, and damage.

(3) Inspection on inner layers after the inner-layer etching. (Not applied to double-layer board)

- Visual Inspection is to be carried out after the dry-ink exposure process.
- To find out if there are any open traces, shorts, scratches, discolorations, bulges, foreign inclusions, peeling, residual copper, and cutting.

(4) Inspection after copper-plating

- Visual Inspection is to be carried out after the copper plating.
- To find out if there are any voids, and blocked holes.

(5) Inspection on the outer layer after exposure

- Visual Inspection is to carry out after the outer-layer dry-ink exposure.
- To find out if there are any open traces, shorts, foreign inclusions, symmetry, and scratches.

(6) Inspection on the outer layers after etching

- Visual inspection is to carry out after etching
- To find out if there are any open traces, shorts, any scratches, discolorations, bulges, foreign inclusions, peeling, residual copper, and cutting.

(7) Before shipping

- All boards would be visually inspected after production.