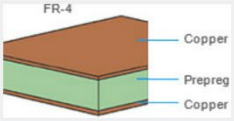
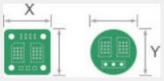
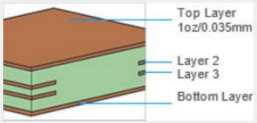
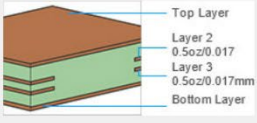
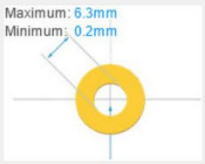
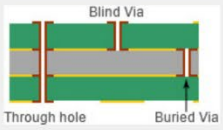
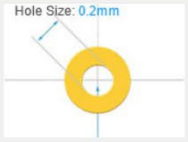
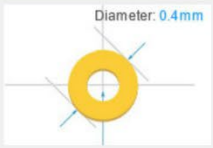
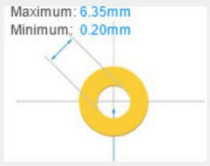
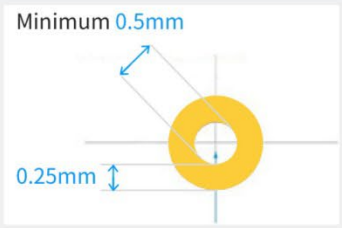
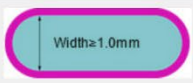
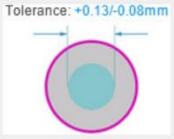
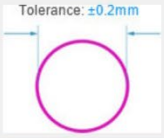
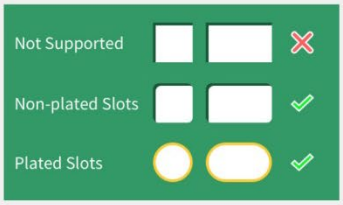
+ PCB Specifications
+ Drill/Hole Size
+ Minimun Annular Ring
+ Minimun Clearance
+ Minimum trace width and spacing
+ BGA
+ Solder Mask
+ Legend
+ Board Outlines
+ Panelization

− PCB Specifications

Features	Capability	Notes	Patterns
count	1,2,4,6 layers	The number of copper layers in the board.	
Material	FR-4	FR-4 Standard Tg 130-140/ Tg 155	
Dielectric constant	4.5(double-sided PCB)	7628 Prepreg 4.6 3313 Prepreg 4.05 2116 Prepreg 4.25	
Max. Dimension	400x500mm	The maximum dimension PCB can accept	
Dimension Tolerance	±0.2mm	±0.2mm for CNC routing, and ±0.4mm for V-scoring	
Board Thickness	0.4/0.6/0.8/1.0/1.2/1.6/2.0mm	The thickness of finished board.	
Thickness Tolerance (Thickness≥1.0mm)	± 10%	e.g. For the 1.6mm board thickness, the finished board thickness ranges from 1.44mm(T-1.6×10%) to 1.76mm(T+1.6×10%)	
Thickness Tolerance (Thickness<1.0mm)	± 0.1mm	e.g. For the 0.8mm board thickness, the finished board thickness ranges from 0.7mm(T-0.1) to 0.9mm(T+0.1).	
Finished Outer Layer Copper	1 oz/【28 g】2 oz (35um/70um)	Finished copper weight of outer layer is 1oz【28 g】or 2oz【57 g】.	
Finished Inner Layer Copper	0.5 oz (17.5um)	Finished copper weight of inner layer is 0.5oz【14 g】by default. 2 oz【57 g】inner copper weight is available for 4-layer PCBs with 1.6mm thickness/pcb3313 stackup/2oz【57 g】outer copper weight.	

− Drill/Hole Size

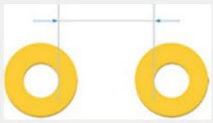
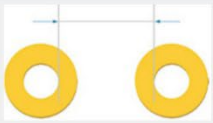
Features	Capability	Notes	Patterns
Drill Hole Size	0.20mm- 6.30mm	Min. drill size is 0.20mm. Max. drill size is 6.30mm.	
Drill Hole Size Tolerance	+0.13/-0.08mm	e.g. for the 0.6mm hole size, the finished hole size between 0.52mm to 0.73mm is acceptable.	
Blind/Buried Vias	Don't support	Currently we don't support Blind/Buried Vias, only make through holes.	
Min. Via hole size	0.2mm	For Single&Double Layer PCB, the minimum via hole size is 0.3mm;For Multi Layer PCB, the minimum via hole size is 0.2mm	
Min. Via diameter	0.4mm	For Single&Double Layer PCB, the minimum Via diameter is 0.5mm; For Multi Layer PCB, the minimum via diameter is 0.45mm(Limitation 0.4mm).	
PTH hole Size	0.20mm – 6.35mm	The annular ring size will be enlarged to 0.15mm in production.	
Pad Size	Minimum 1.0mm	The pad size will be enlarged by 0.5mm than the hole size. The minimum size of annular ring around plated through hole pads is 0.25mm. If the recommended sizes are not respected then the pad will not be produced properly.	
Min. Non-plated holes	0.50mm	The minimum NPTH dimension is 0.50mm, Please add the NPTH in the mechanical layer or keep out layer.	
NPTH	0.2mm	We make NPTH via dry sealing film process, if customer would like a NPTH but around with pad/copper, our engineer will dig out around pad/copper about 0.2mm-0.25mm, otherwise the metal potion will be flowed into the hole and it becomes a PTH. (there will be no copper dig out optimization for single board).	
Min. Plated Slots	0.5mm	The minimum plated slot width is 0.5mm, which is drawn with a pad.	
Min. Non-Plated Slots	1.0mm	The minimum Non-Plated Slot Width is 1.0mm, please draw the slot outline in the mechanical layer(GML or GKO)	
Min. Castellated Holes	0.60mm	The minimum diameter of castellated holes is 0.60mm.	

Hole size Tolerance (Plated)	+0.13mm/-0.08mm	e.g. for the 1.00mm Plated hole, the finished hole size between 0.92mm to 1.13mm is acceptable.	
Hole size Tolerance (Non-Plated)	±0.2mm	e.g. for the 1.00mm Non-Plated hole, the finished hole size between 0.80mm to 1.20mm is acceptable.	
Rectangle Hole/Slot	Don't support	Rectangle/Square Slots, we don't make rectangular or square plated holes,only make oval or round plated slots. For non-plated slots, rounded corner-rectangular or square slots are supported. The recommended minimum size is 3x3mm.	

− Minimun Annular Ring

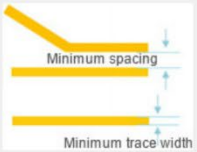
	Minimum annular ring	PTH	Patterns
1oz【28 g】 Copper	0.13mm	0.3mm	
2oz【57 g】 Copper	0.2mm	0.3mm	

− Minimun Clearance

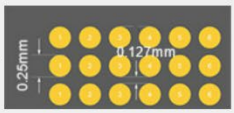
Features	Capability	Patterns
Hole to hole clearance(Different nets)	0.5mm	
Via to Via clearance(Same nets)	0.254mm	
Pad to Pad clearance(Pad without hole, Different nets)	0.127mm	
Pad to Pad clearance(Pad with hole, Different nets)	0.5mm	
Via to Track	0.254mm	
PTH to Track	0.33mm	

NPTH to Track	0.254mm	
Pad to Track	0.2mm	

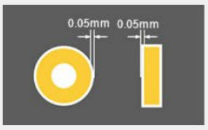
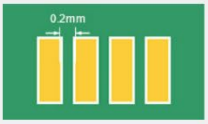
− Minimum trace width and spacing

	Min. Trace width	Min. Spacing	Patterns
1-2 Layers	5mil (0.127mm)	5mil (0.127mm)	
4-6 Layers	3.5mil (0.09mm)	3.5mil (0.09mm)	
2oz【57 g】 Copper weight	8mil (0.2mm)	8mil (0.2mm)	

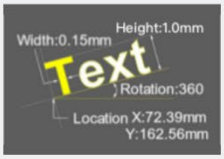
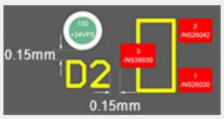
− BGA

Layer count	Min. BGA Pad Dimensions	Min. Distance Between BGA	Patterns
1/2 layers	0.25 mm	0.127mm	
4/6 layers	0.25 mm	0.127mm	

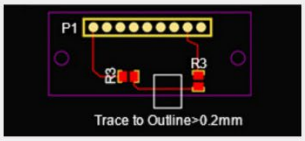
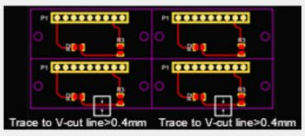
− Solder Mask

Features	Capability	Notes	Patterns
Solder mask opening/ expansion	0.05mm	The solder mask should have a minimum of a 0.05 mm “growth/mask opening” around the pad to allow for any mis-registration.	
Solder bridge	0.2mm(green) 0.254mm(other colors)	To have solder mask bridge, the spacing between copper pads edge must be 0.2mm (8mils) or more.	
Solder mask color	green, red, yellow, blue, white, and black.	We use LPI (Liquid Photo Imageable) solder mask. It is the most common type of mask used today.	
Solder mask dielectric constant	3.8		
Solder mask thickness	10-15UM		

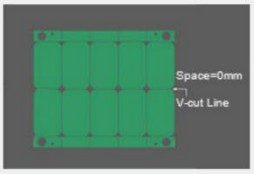
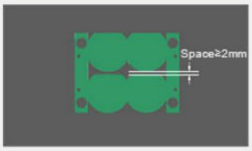
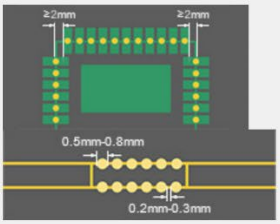
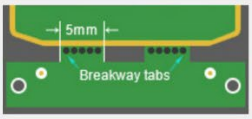
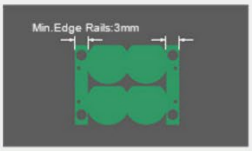
Legend

Features	Capability	Notes	Patterns
Minimum Line Width	6 mil (0.153mm)	Characters width less than 6mil(0.153mm) will be unidentifiable.	
Minimum text height	40 mil (1.0mm)	Characters height less than 40 mil(1.0mm) will be unidentifiable.	
Character width to height ratio	1:6	The preferred ratio of width to height is 1:6.	
Hollow-carved Character width to height ratio	1:6	The preferred ratio of width to height is 1:6	
Pad To Silkscreen	0.15mm	The Minimum Distance Between Pad and Silkscreen is 0.15mm.	

Board Outlines

Features	Capability	Notes	Patterns
Trace to Outline	0.2mm	Ships as individual board(Rounting):Trace to Outline≥0.2mm	
Trace to V-cut line	0.4mm	Ship as panel with V-scoring: Trace to V-cut line≥0.4mm	

− Panelization

Features	Capability	Notes	Patterns
Panelization without space	0mm	The space between boards is 0mm.	
Panelization with space	2mm	Make sure the space between boards should be ≥2mm,otherwise it will be hard to process for rounding.	
Panelized Round board	≥20mmx20mm	The single round board size should be≥20mmx20mm. Panelize with stamp holes and add tooling strips on four board edges	
Panelized castellated holes board	Panelize with stamp holes and add tooling strips on four board edges	The distance between castellated hole and board corner should be larger than 2mm. Recommended diameter of stamp hole is 0.5mm-0.8mm; Recommended distance between the two stamp holes is 0.2-0.3mm	
Min. Width of Breakaway Tab	4mm	The minimum width of breakaway tab is 4mm. For breakaway with mouse-bites, the minimum width is 5mm.	
Min. Edge Rails	3mm	If choosing panel by PCB, we will add 5mm edge rails on both sides by default.