

# PCB Capability - PCBA allinone Solution

PCBAallinone provides 2-64 layered PCBs for fabrication, including Rigid , flex boards ,Rigid-Flex, HDI. Quantities range from prototype to mass production.

Items	Manufacturing Capabilities	Remarks
Number of Layers	1-64 layers	For orders above 10 layers, please contact our sales rep.
Material	FR-4,Aluminum, High-TG 、 Rogers、 Nelco、 RCC、 PTFE,Au Base,,Halogen- Free,ARLON、 Mixed Laminate,Rigid-Flex.	Except for FR-4 and Alu Base in quoting system,Please send mail inquiry.
Max Board Size (Dimension)	610x1100 mm	Any sizes beyond this dimension, please contact us.
Board Size Tolerance(Outline)	±0.2mm/±0.5mm	±0.2mm for CNC routing, and ±0.5mm for V-scoring.
Board Thickness	0.2-2.4mm	0.2, 0.4, 0.6, 0.8, 1.0, 1.2, 1.6, 2.0, 2.4mm. Please contact us if your board exceeds these.
(T≥ 1.0mm)/Tolerance For Finished Board Thickness (BoardThickness≥ 1.0mm)	±10%	T=1.6mm(1.44mm): (1.6-1.6× 10%)~1.76mm(1.6+1.6×10%)  For example,the required board thickness is 1.6mm,the actual thickness range 1.44(1.6- 1.6×10%)mm to1.76(1.6+1.6×10%)mm .  T=0.8mm, (0.7mm) : (T- 0.1)~0.9mm(T+0.1)
Board Thickness Tolerance(t<1.0mm)	±0.1mm	For example,the required board thickness is 0.8mm,the actual thickness range 0.7mm(0.8- 0.1) to 0.9mm(0.8+0.1).

Items	Manufacturing Capabilities	Remarks
Min Trace Width/Space	$\geq 3/3\text{mil}(0.076\text{mm})$	4/4mil(1 OZ) , 5/5mil(2 OZ) , 8/8mil(3 OZ) , 4/4mil (1OZ finish copper)
Outer Layer Copper Thickness	1oz/2oz/3oz(35 $\mu\text{m}$ /70 $\mu\text{m}$ /105 $\mu\text{m}$ )	Also known as copper weight. 35 $\mu\text{m}$ =1oz, 70 $\mu\text{m}$ =2oz, 105 $\mu\text{m}$ =3oz. Please contact us if you need copper weight greater than 3oz.
Min Copper Griding Width/Gap	$\geq 6\text{mil}/8\text{mil}(0.15/0.20\text{mm})$	6/8mil (1OZ finish copper) ,8/10mil(2OZ finish copper),10/12mil(3OZ finish copper)
Min Each Text Font Width	$\geq 8\text{mil}(0.20\text{mm})$	8mil (1OZ finish copper) , 10mil(2OZ finish copper), 12mil(3OZ finish copper)
BGA/Min BGA	$\geq 8\text{mil}(0.2\text{mm})$	Please send mail inquiry
Finish Outer Copper Thickness	1-12oz	>3OZ, Please send mail inquiry
Min Spacing from board edge to Conductiv	$\geq 10\text{mil}(0.25\text{mm})$	Spacing $\geq 0.25\text{mm}$ for Routing,>0.35mm for scoring. Otherwise accept copper exposed on board edge.
Min Width of the Trace Connected Copper Ground	4mil	
Max Aspect Ratiofor,Board thickness vs drill bit size	<12	>8,please send mail inquiry

Items	Manufacturing Capabilities	Remarks
Inner Layer Copper Thickness	1oz/1.5oz(35μm/50μm)	Inner copper weight as per customer's request for 4 and 6 layers. Please contact us if you need copper weight greater than 1.5oz.
Drill Sizes (CNC)	0.2-6.3mm	Min drill size is 0.2mm, max drill is 6.3mm. Any holes greater than 6.3mm or smaller than 0.3mm will be subject to extra charges.
Min Width of Annular Ring	0.15mm(6mil)	For pads with vias in the middle, Min width for Annular Ring is 0.15mm(6mil).
Finished Hole Diameter (CNC)	0.1mm-6.2mm/( Min Laser Hole 0.075mm)	The finished hole diameter will be smaller than size of drill bits because of copper plating in the hole barrels
Finished Hole Size Tolerance(CNC)	±0.08mm	For example, if the drill size is 0.6mm, the finished hole diameter ranges from 0.52mm to 0.68mm will be considered acceptable.
Standard Stamp Hole Size	0.5mm	Typical distance between stamp holes edge is 0.25mm,Placed in the center of outline, stamp holes count at each side $\geq 3$
Max Solder Filled Via	$\leq 0.6$ mm	Just cover the vias( $>0.6$ mm),can' t be filled
Min Annular Ring for Via	4mil	Min 6mil annular ring for component hole.Increase Annular rings help electricity flows.
Solder Mask	LPI	Liquid Photo-Imageable is the mostly adopted. Thermosetting Ink is used in the inexpensive paper-based boards.

Items	Manufacturing Capabilities	Remarks
Minimum Character Width(Legend)	0.15mm	Characters of less than 0.15mm wide will be too narrow to be identifiable.
Minimum Character Height (Legend)	0.8mm	Characters of less than 0.8mm high will be too small to be recognizable.
Character Width to Height Ratio (Legend)	1:5	In PCB silkscreen legends processing, 1:5 is the most suitable ratio
Minimum Diameter of Plated Half Holes	0.6mm	Design Half-Holes greater than 0.6mm to ensure better connection between boards.
Surface Treatment	HASL with lead HASL lead free Immersion Gold/Tin/Silver OSP、Immersion Gold + OSP Carbon, ENIG, Hard Gold,Selective Gold,Electrolytic Gold	The most popular three types of PCB surface finish. Please contact us for other finishes.
Solder Mask	Green ,Red, Yellow, Blue, White ,Black	No extra charge (Green, Red, Yellow, Blue)
Min Solder Bridge	Green Ink, 0.1mm Black/White ink,0.15mm Other Ink, 0.12mm	Require over 0.18mm space between copper pads to keep solder bridge.Thicker copper thickness, larger space required.
Min Text Height	≥30mil	If the text height < 0.8mm,the silkscreen can't be clear.
Min Text Font Width	≥5mil	If the text font width < 0.1mm,the silkscreen can't be printed.

Items	Manufacturing Capabilities	Remarks
The space between component box and S/M Opening	$\geq 0.2\text{mm}$	If the space $< 0.2\text{mm}$ , the device box is cleared by increased soldermask, the line width of device box get thinner, it would result poor screen printing.
Min aspect ratio, text width vs height	1:6	
Min drill bit for Slots	0.60mm	Finish Plated Slot Width $\geq 0.45\text{mm}$ , Finish Non-plated Slot Size Width $\geq 0.60\text{mm}$
V-CUT	V-cut length $\geq 55\text{mm}$ V-cut width $\geq 380\text{mm}$	V-cut width means the length of the side that is perpendicular to V-cut line.
Different Boards In a Panel	max. 6 different PCBs	Scoring or Rout with breakout pins.
Peel Strength	$\geq 2.0\text{N/cm}$	
Flame Retardant Rating	94V-0	
Impedance Controlled Type	Single ended, Differential, Single ended Coplanar. Differential Coplanar	Control 45~85 OHM For Single ended/Single ended Coplanar Impedance, Control 85~110 OHM For Differential/Differential Coplanar Impedance
Special Process	Resin Filled, via in pad, Mixed Laminate, PTFE, Blind & Buried Via, IC Bonding, Thick Copper	Special processes need to be reviewed.
Silkscreen	White, Black, None	No extra charge.

Items	Manufacturing Capabilities	Remarks
Panelization	V-scoring, Tab-routing, Tab-routing with Perforation (Stamp Holes)	Leave min clearance of 1.6mm between boards for break-routing. For V-score panelization, set the space between boards to be zero.
HDI	1+n+1、2+n+2、3+n+3	
Max Aspect Ratio	18:1	
Bow & Twist	≤0.5%	
Special Capability	Gold Finger Plating 、 Peelable、 Carbon ink	
Blind & Buried Via	Multi-press Cycle≤ 3 times	Please send mail inquiry