Notice: JLCPCB factories will be closed on Oct. 1–3 and Oct. 6 for holidays. Orders can still be placed and customer service will remain available. The delivery date will be postponed accordingly.   Learn more>

JLC Other Services: JLC3DP - 3D Printing | JLCCNC - CNC Machining | JLCMC - Mechatronic Parts   ⬇ Download   💲 Coupons   ⓘ Help   🇺🇸 USD ⌄

J@LC JLCPCB   Why JLCPCB?   Capabilities   Support ⌄   Resources ⌄   🔍   Order now   My file   Sign in   🛒 0

PCB Manufacturing & Assembly Capabilities

Know JLCPCB's Capabilities & Get your PCBs Built Fast

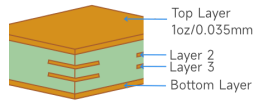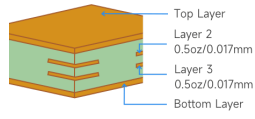🟢 Green   🟣 Purple   🔴 Red   🟡 Yellow   🔵 Blue   ⚪ White   ⚫ Black
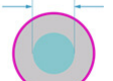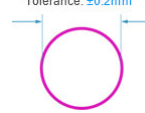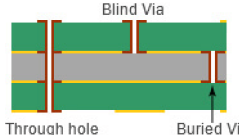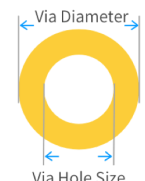
👋 Welcome back help you?
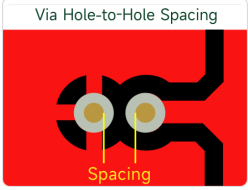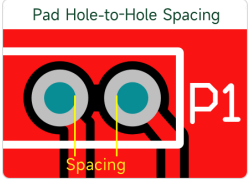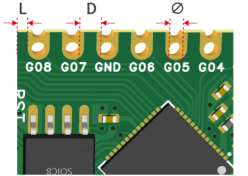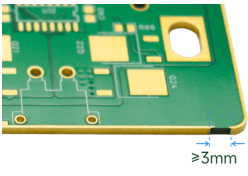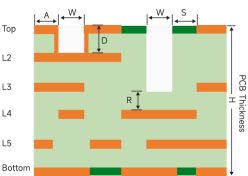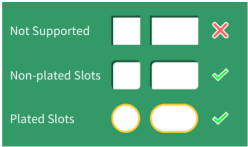
Chat Now

Rigid PCB	Flex PCB	PCB Assembly

PCB Specifications ⌄

Features	Capability	Description	Patterns
Layer count	1-32 Layers	The number of copper layers in the PCB	
Controlled Impedance	4/6/8/10/12/14/16/18/20/.../32 layers	User Guide to the JLCPCB Impedance Calculator JLCPCB Impedance Calculator	
Impedance Tolerance	±10%		
Material	FR-4	Grade A laminates from suppliers including Nan Ya, KB, Shengyi and etc.	
	Aluminum-Core	1-layer Aluminum-core PCBs	
	Copper-Core	1-layer copper-core PCBs with direct heatsink contacts to core (≥ 1 × 1 mm)	
	RF PCB	1 oz copper, 2-layer RF PCBs with Rogers and PTFE cores	
FR-4 Dielectric Constants	4.5 (2-Layer PCB)	7628 Prepreg 4.4 3313 Perpreg 4.1 2116 Perpreg 4.16	
Maximum Dimensions	FR4 PCB: 670 × 600 mm Rogers / PTFE Teflon PCB: 590 × 438 mm Aluminum PCB: 602 × 506 mm Copper PCB: 480 × 286 mm	These limits apply to PCBs with thickness ≥ 0.8 mm. The thinner FR4 PCBs are 500 × 600 mm maximum. 2-layer FR4 PCBs can reach a maximum size of 1020 × 600 mm.	
Minimum Dimensions	Regular: 3 × 3 mm. Castellated / Plated Edges: 10 × 10 mm.	These limits apply to PCBs with thickness ≥ 0.6 mm. Manual review required for thinner PCBs. Panelization is recommended for small-sized boards.	
Dimension Tolerance	±0.1mm	±0.1mm(Precision) and ±0.2mm(Regular) for CNC routing, and ±0.4mm for V-scoring	
Thickness	0.4 – 4.5 mm	Thickness for FR4 are: 0.4/0.6/0.8/1.0/1.2/1.6/2.0 mm (2.5 mm and above are for 12+ layer PCBs only)	

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	2-layer: 1 oz / 2 oz / 2.5 oz / 3.5 oz / 4.5oz Multi-layer: 1 oz / 2 oz		
Finished Inner Layer Copper	0.5 oz / 1 oz / 2 oz	Finished copper weight of inner layer is 0.5oz by default.	
Soldermask	Green, Purple, Red, Yellow, Blue, White, and Black.	We use LPI (Liquid Photo Imageable) solder mask. This is the most common type of mask used today. Heat-cured ink soldermask is usually found on low-cost, single-sided PCBs.	
Surface Finish	HASL (leaded / lead-free), ENIG, OSP (copper core boards only)	FR4 has all three finishes available, 6+ layers and RF boards only have ENIG. Aluminium core boards only have HASL. Copper core boards only have OSP.	
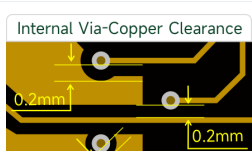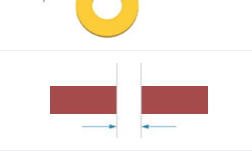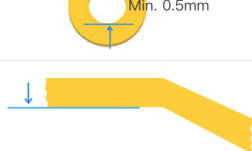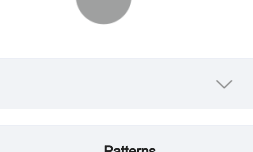
Drilling ⌄

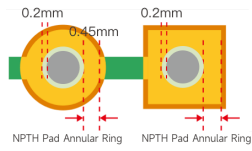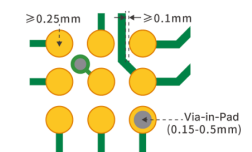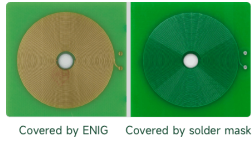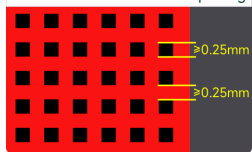| Features	Capability	Description	Patterns
Drill Diameter	1-layer: 0.3 – 6.3 mm 2-layer: 0.15 – 6.3 mm Multilayer: 0.15 – 6.3 mm	Holes with diameter ≥ 6.3 mm are CNC routed from a smaller drilled hole. Min. drill diameter for 2- or more-layer PCBs is 0.15 mm (more costly!) Min. drill diameter for aluminum-core PCBs is 0.65 mm Min. drill diameter for copper-core PCBs is 1.0 mm	Maximum: 6.3mm Minimum: 0.15mm
Hole size Tolerance (Plated)	Through-holes：+0.13 / -0.08 mm Press-fit holes： ±0.05 mm (multilayer ENIG boards only – mention the specific holes in PCB Remark)	e.g. for the 0.6mm hole size, the finished hole size between 0.52mm to 0.73mm is acceptable.	Tolerance: +0.13/-0.08mm
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.	Tolerance: ±0.2mm
Average Hole Plating Thickness	18μm		
Blind/Buried Vias	Not supported	Currently we don't support Blind/Buried Vias, only make through holes.	
Min. Via hole size/diameter	0.15mm / 0.25mm	1-layer (NPTH only): 0.3 mm hole size / 0.5 mm via diameter 2-layer: 0.15mm hole size / 0.25mm via diameter Multilayer: 0.15 mm hole size / 0.25 mm via diameter ① Via diameter should be 0.1mm(0.15mm preferred) larger than Via hole size. ② Preferred Min. Via hole size: 0.2mm	
Min. Non-plated holes	0.50mm	Please draw NPTHs in the mechanical layer or keep out layer.	
Min. Plated Slots	0.5mm	The minimum plated slot width is 0.5mm, which is drawn with a pad. The length of the slot should be at least 2 times of the width.	
Min. Non-Plated Slots	1.0mm	The minimum Non-Plated Slot Width is 1.0mm, please draw the slot outline in the mechanical layer(GM1 or GKO)	

Via Hole-to-Hole Spacing	0.2mm		
Pad Hole-to-Hole Spacing	0.45mm		
Min. Castellated Holes	0.5mm	Castellated holes are metalized half-holes on PCB edges, commonly used on daughter boards to be soldered onto carrier PCBs. ① Hole diameter (Φ): ≥ 0.5 mm ② Hole to board edge (L): ≥ 1 mm ③ Hole to hole (D): ≥ 0.5 mm ④ Min. PCB size：10 × 10 mm ⑤ Min. PCB thickness: 0.6 mm	
Plated Edges	10 x 10mm	Plated edges are copper-plated and ENIG treated. HASL is not supported. ① Min. PCB size: 10 × 10 mm ② Min. PCB thickness: 0.6 mm ③ At least 3 breaks (more for larger PCBs) in the edge plating are required for support tab connections	
Blind Slot		① Blind slot width (W): ≥1.0mm ② Blind slot depth (D): ≥0.2mm ③ Blind slot annular width (A): ≥0.3mm (The pad width of PTH blind slots) ④ Safety distance (S): ≥0.2mm (The distance from NPTH blind slots to pad/traces/copper plane) ⑤ Blind slot remaining thickness (R): ≥0.2mm (The distance from the bottom of the blind slot to the nearest inner copper layer/surface substrate) ⑥ Supports 2-32-layer FR4 boards with a thickness of ≥0.8mm	
Rectangular Holes / Slots	Not supported	Rectangular holes and slots without rounded corners are not supported.	

Traces ⌄

Features	Capability	Description	Patterns
Min. track width and spacing (1 oz)	0.10 / 0.10 mm (4 / 4 mil)	1- and 2-layer: 0.10 / 0.10 mm (4 / 4 mil) Multilayer: 0.09 / 0.09 mm (3.5 / 3.5 mil). 3 mil is acceptable in BGA fan-outs.	
Min. track width and spacing (2 oz)	0.16 / 0.16 mm (6.5 / 6.5 mil)	2-layer: 0.16 / 0.16 mm (6.5 / 6.5 mil) Multilayer: 0.16 / 0.20 mm (6.5 / 8 mil)	
Min. track width and spacing (2.5 oz)	2 layer: 0.2/0.2mm(8mil/8mil)		
Min. track width and spacing (3.5 oz)	2 layer: 0.25/0.25mm(10mil/10mil)		
Min. track width and spacing (4.5 oz)	2 layer: 0.3/0.3mm(12mil/12mil)		
Track width tolerance	±20%	e.g. For a 0.1 mm track, the finished track width ranges from 0.08 and 0.12 mm.	
PTH annular ring	≥0.20mm	2-layer: 1 oz: Recommended 0.25 mm or above; absolute minimum 0.18 mm 2 oz: 0.254 mm or above Multilayer: 1 oz: Recommended 0.20 mm or above; absolute minimum 0.15 mm 2 oz: 0.254 mm or above	

Features	Capability	Description	Patterns
NPTH pad annular ring	≥0.45mm	Recommended 0.45 mm or more. This is to allow a 0.2 mm ring of copper to be removed around the hole for the sealing film to attach. Pad sizes smaller than the recommended value can result in the annular ring being very thin or completely missing.	
BGA	0.25mm	① BGA pad diameter ≥ 0.25 mm ② BGA pad to trace clearance ≥ 0.1 mm (min. 0.09 mm for multilayer boards) ③ Vias can be placed within BGA pads using filled and plated-over vias	
Trace coils	0.15/0.15mm	Minimum trace width/clearance: 0.15/0.15mm, when traces are covered by solder mask (1oz). Minimum trace width/clearance: 0.25/0.25mm, when traces are NOT covered by solder mask (1oz). ENIG only(high risk of short circuit with HASL)	
Hatched grid width and spacing	0.25 mm		
Same-net track spacing	0.25mm		
Inner layer via hole to copper clearance	0.2mm		
Inner layer PTH pad hole to copper clearance	0.3mm		
Pad to track clearance	0.1mm	Min. 0.1 mm (stay well above if possible). Min. 0.09 mm locally for BGA pads	
SMD pad to pad clearance (different nets)	0.15mm	More details of SMD pad spacing: SMD Components Minimum Spacing	
Via hole to Track	0.2mm		
PTH to Track	0.28mm	0.35mm is recommended, minimum 0.28mm	
NPTH to Track	0.2mm		

Soldermask	⌄

Features	Capability	Description	Patterns

Soldermask Expansion	1:1	LDI equipment upgraded in June 2025. Pad size:soldermask opening can be 1:1(Previous production file will be followed in Reorder). Keep at least 0.09 mm clearance between soldermask openings and neighboring traces.	
Soldermask bridge	0.10mm	1oz: Min. pad spacing: 0.10 mm (green, red, yellow, blue, purple) Min. pad spacing: 0.13 mm (black, white) 2oz: Min. pad spacing: 0.20 mm (any color)	
Plugged vias	Filled with soldermask	Vias are filled with soldermask for an opaque finish. Click for detailed explanation ① Filled vias must not have soldermask openings on either side ② Filled vias should have ≥ 0.35 mm clearance from other soldermask openings (e.g. pads) ③ Filled vias must be no wider than 0.5 mm diameter	
JLCPCB Via-in-Pad Process	Epoxy Filled & Capped Copper paste Filled&Capped	Vias are filled with epoxy resin or copper paste and then plated over to achieve an opaque and smooth finish. Click for detailed explanation ① Vias are filled and plated over. Choose copper paste filling for applications requiring high thermal conductivity. ② This process is the default for 6-layer and above multilayer boards. ③ Compatible with via diameters from 0.15 to 0.5 mm.	
Solder mask dielectric constant	3.8		
Solder mask ink thickness	≥ 10μm		

Legend	⌄

Features	Capability	Description	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.	

Outline	⌄

Features	Capability	Description	Patterns

Routed	0.2mm	① Copper clearance from routed board edges: ≥0.2 mm ② Copper clearance from routed slots: ≥0.2 mm ③ Dimension tolerance for routed board edges: ±0.2 mm (regular precision); ±0.1 mm (high precision) ④ Minimum dimension 50*50mm for high precision, and at least 3 tooling holes with minimum 1.5mm diameter on different corners. ⑤ Minimum slot width for aluminum/copper core PCB: 1.6mm.	
V-Cut	0.4mm	① Copper clearance from V-cut board edges: ≥0.4 mm ② Dimension tolerance for V-cut board edges: ±0.4 mm. PCB thickness ≥ 0.6 mm ③ Zero panel board spacing by default. Alternatively, V-cut along one direction with no spacing and route along the other direction with 1.6 or 2 mm board spacing. ④ Min. panel dimensions: 70 × 70 mm; max. panel dimensions: 475 × 475 mm ⑤ V-cut groove angle: 25°	
Mouse bites Panel	0.2mm	① Copper clearance from non-mouse-bite board edges: ≥0.2 mm ② Dimension tolerance for non-mouse-bite board edges: ±0.2 mm (regular precision); ±0.1 mm (high precision) ③ Panel board spacing: 1.6 or 2 mm ④ Serrated edges will remain after depanelization ⑤ Minimum tooling edge width: 3 mm. For SMT assembly at JLCPCB, use 5 mm tooling edges, 2 mm tooling holes, and 1 mm fiducials centered at 3.85 mm from the panel edges. ⑥ Recommended diameter of mouse bite is 0.5mm-0.8mm; Recommended distance between the two mouse-bites is 0.2-0.3mm. The minimum width of breakaway tab is 4mm. For breakaway with mouse-bites, the minimum width is 5mm.	
Panelization with space	2mm	The spacing between boards should be ≥ 2 mm, as narrow spacing results in difficulties for routing and V-cut.	
Panel of Circular PCBs	≥20mmx20mm	The single round board size should be≥20mmx20mm when choose panel by JLCPCB. Panelize with stamp holes and add tooling strips on four board edges	

READY TO GET STARTED?

See PCB Fab FAQs

Get Instant Quote

COMPANY

About JLCPCB

News

Blog

How we work

Quality Management

Security

Certifications

SUPPORT

Help Center

Shipping Info

How To Order

How To Track

Contact Us

NETWORK SITES

EasyEDA - PCB Design Tool

JLC3DP - 3D Printing&CNC Machining

JLCMC - Mechatronic Parts

OSHWLAB - Open Source Hardware

JLCPCB

CONNECT WITH US