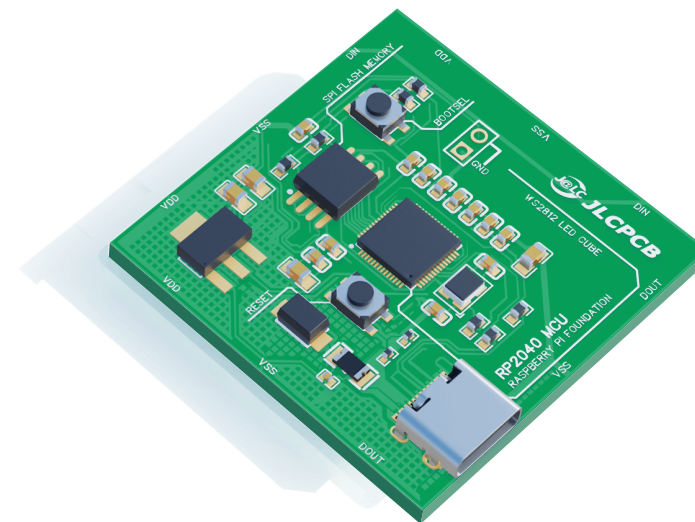


[Why JLCPCB?](#)[Capabilities](#)[Support](#)[Resources](#)[Order now](#)[My file](#)[Sign in](#)

# PCB Manufacturing & Assembly Capabilities

Know JLCPCB's Capabilities & Get your PCBs Built Fast

[Green](#)[Purple](#)[Red](#)[Yellow](#)[Blue](#)[White](#)[Black](#)

## Rigid PCB

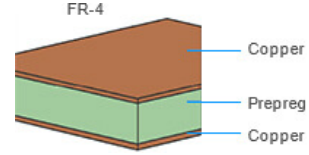
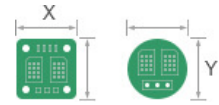

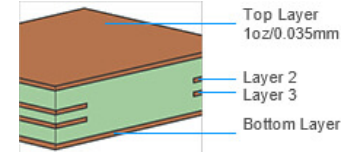
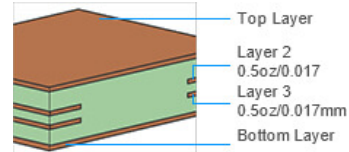
## Flex PCB

## PCB Assembly

### PCB Specifications



Features	Capability	Notes	Patterns
Layer count	1-20 Layers	The number of copper layers in the board.	
Controlled Impedance	4/6 layer, default layer stack-up	<a href="#">Controlled Impedance PCB Layer Stackup</a> <a href="#">JLCPCB Impedance Calculator</a>	

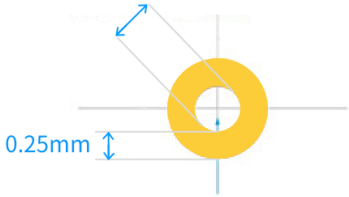
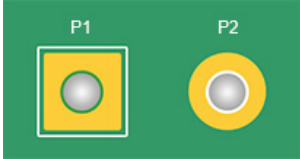


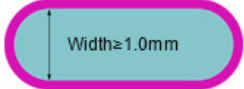
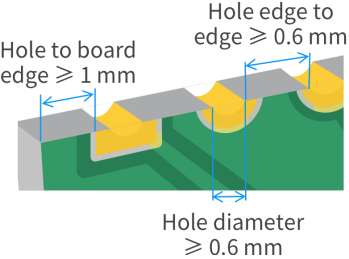
Material	FR-4 Aluminum Copper core Rogers / PTFE Teflon	FR-4: Tg 135 / Tg140 / Tg155 / Tg170 Aluminum thermal conductivity: 1W/m.K Copper core thermal conductivity: 380W/m.K	
Dielectric constant	4.5(double-sided PCB)	7628 Prepreg 4.6 3313 Prepreg 4.05 2116 Prepreg 4.25	
Max. Dimension	400x500mm	The maximum dimension JLCPCB can accept	
Dimension Tolerance	±0.1mm	±0.1mm(Precision) and ±0.2mm(Regular) for CNC routing, and ±0.4mm for V-scoring	
Board Thickness	0.4 - 2.5 mm	Thickness for FR4 are: 0.4/0.6/0.8/1.0/1.2/2.0 mm (2.5 mm only available with 12 layers or more. )	
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 / 2 oz (35um / 70um)	Finished copper weight of outer layer is 1oz or 2oz.	
Finished Inner Layer Copper	0.5 oz / 1 oz / 2 oz (17.5um / 35um / 70um)	Finished copper weight of inner layer is 0.5oz by default.	
Surface Finish	HASL (leadless / lead-free), ENIG, OSP (copper core boards only)	FR4 has all three finishes available, 6+ layers and RF boards only have ENIG.	

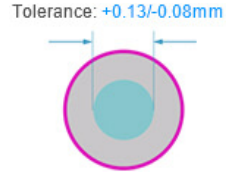
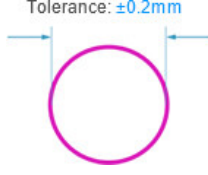
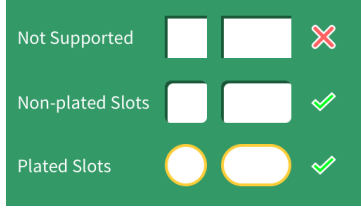
Aluminium core boards only have HASL. Copper core boards only have OSP.

## Drill/Hole Size



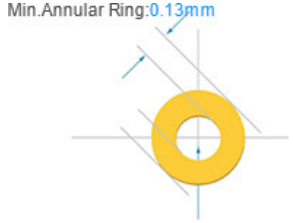
Features	Capability	Notes	Patterns
Drill Hole Size	0.15mm - 6.30mm	1 & 2 Layer PCB: 0.3 - 6.3mm Multi-Layer PCB: 0.15 - 6.3mm (0.15mm more costly)	<p>Maximum: 6.3mm Minimum: 0.2mm</p>
Drill Hole Size Tolerance	+0.13/-0.08mm	e.g. for the 0.6mm hole size, the finished hole size between 0.52mm to 0.73mm is acceptable.	
Blind/Buried Vias	Don't support	Currently we don't support Blind/Buried Vias, only make through holes.	<p>Blind Via Through hole Buried Via</p>
Min. Via hole size/diameter	0.15mm / 0.25mm	<p>- 1 &amp; 2 Layer PCB: 0.3mm(hole size) / 0.5mm(diameter)</p> <p>- Multi-Layer PCB: 0.15mm(Via hole size) / 0.25mm(Via diameter)</p> <p>① Via diameter should be 0.1mm(0.15mm preferred) larger than Via hole size</p> <p>② Preferred Min. Via hole size: 0.2mm</p>	
PTH hole Size	0.20mm - 6.35mm	The annular ring size will be enlarged to 0.15mm in production.	<p>Maximum: 6.35mm Minimum: 0.20mm</p>

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.	<p>Minimum 0.5mm</p> 
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 portion 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	<p>A castellated pad includes a plated half-hole on the edge of a board, usually used on daughter PCB modules to solder to carrier boards.</p> <ol style="list-style-type: none"> <li>① Hole diameter <math>\geq 0.6</math> mm</li> <li>② Hole to board edge <math>\geq 1</math> mm</li> <li>③ Min. board size 10 × 10 mm</li> </ol>	

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.	 <p>Tolerance: +0.13/-0.08mm</p>
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.	 <p>Tolerance: ±0.2mm</p>
Rectangle Hole/Slot	Don't support	<p>Rectangle/Square Slots, we don't make rectangular or square plated holes, only make oval or round plated slots.</p> <p>For non-plated slots, rounded corner-rectangular or square slots are supported. The recommended minimum size is 3x3mm.</p>	 <p>Not Supported</p> <p>Non-plated Slots</p> <p>Plated Slots</p>

## Minimum Annular Ring

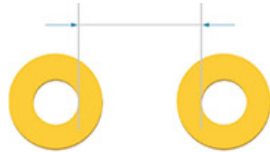
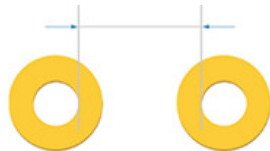
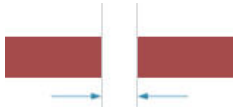
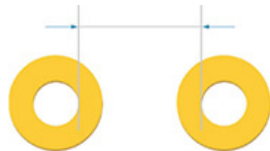





	Minimum annular ring	PTH	Patterns
1oz Copper	0.13mm	0.3mm	 <p>Min. Annular Ring: 0.13mm</p>
2oz Copper	0.2mm	0.3mm	

## Minimum 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	
--------------	-------	--

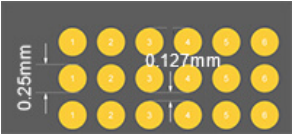
Minimum trace width and spacing

▼

	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 Copper weight	8mil (0.2mm)	8mil (0.2mm)	

BGA

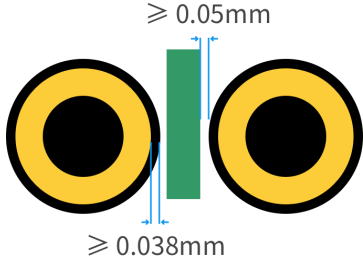
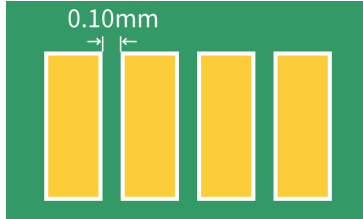
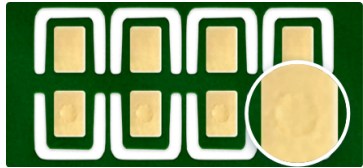
▼

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

Soldermask Expansion	0.038mm	2 layer: Expansion $\geq 0.038$ mm each side; Edge of opening to adjacent traces $\geq 0.05$ mm. Multilayer: No expansion required	
Min. Solder bridge	0.10mm	2 layers: 0.10 mm regular, 0.08 mm minimum (0.13 mm with black or white soldermask). Multilayer: 0.08 mm (0.13 mm with black or white soldermask). Soldermask webs are possible between pads at least 4 mil apart.	
Via Covering	Epoxy Filled & Capped Copper paste Filled&Capped	Via hole size: 0.2 to 0.5 mm Annular ring: 0.05 mm minimum, 0.075 mm preferred The via-in-pad must be placed more than 1.0 mm from regular PTHs or NPTHs <a href="#">Learn more&gt;</a>	
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 ink 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.	

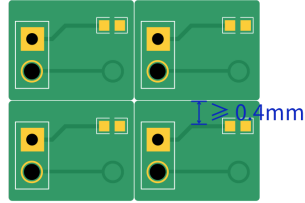
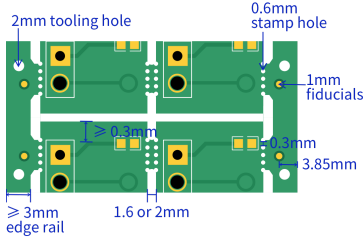


Minimum text height	40 mil (1.0mm)	Characters height less than 40 mil(1.0mm) will be unidentifiable.	
Character width to height ratio	1:6	The preferred ratio of width to height is 1:6.	
Hollow-carved Character width to height ratio	1:6	The preferred ratio of width to height is 1:6	
Pad To Silkscreen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

## Board Outlines

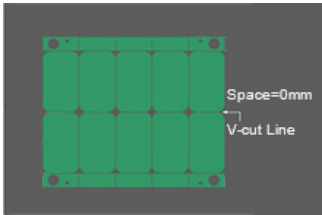
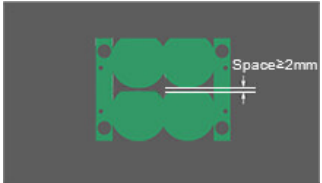


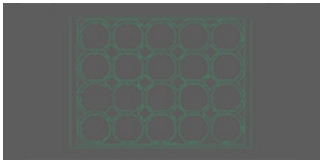
Features	Capability	Notes	Patterns
Trace to Outline(Routed)	0.3 mm	① Copper to board edge $\geq 0.3$ mm ② Copper to milled slot $\geq 0.3$ mm ③ Routed outline tolerance: $\pm 0.2$ mm (regular), $\pm 0.1$ mm (precision)	

Trace to V-cut line(V-Cut Panel)	0.4 mm	<ul style="list-style-type: none"> <li>① Copper to V-cut line <math>\geq 0.4</math> mm</li> <li>② V-cut outline tolerance <math>\pm 0.4</math> mm (board thickness <math>\geq 0.6</math> mm)</li> <li>③ Zero spacing between sub-boards by default. If using V-cuts on two opposing edges only, 1.6 or 2.0 mm spacing can be used on the other edges.</li> </ul>	 <p>V-cut</p>
Mouse Bites Panel	0.3 mm	<ul style="list-style-type: none"> <li>① Copper to routed parts of edges <math>\geq 0.3</math> mm</li> <li>② Routed outline tolerance: <math>\pm 0.2</math> mm (regular), <math>\pm 0.1</math> mm (precision)</li> <li>③ 1.6 or 2.0mm spacing between sub-boards by default.</li> <li>④ Stamp-like edges will remain after separation</li> <li>⑤ Minimum edge rail is 3mm in width. For PCB assembly, use 5mm edge rails, 1mm fiducials placed, 2mm tooling holes, 3.85mm from fiducial to board edge.</li> </ul>	

## 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 $\geq 2$ mm, otherwise it will be hard to process for rounding.	

Panelized Round board	$\geq 20\text{mm} \times 20\text{mm}$	<p>The single round board size should be <math>\geq 20\text{mm} \times 20\text{mm}</math>.</p> <p>Panelize with stamp holes and add tooling strips on four board edges</p>	
Panelized castellated holes board	Panelize with stamp holes and add tooling strips on four board edges	<p>The distance between castellated hole and board corner should be larger than 2mm.</p> <p>Recommended diameter of stamp hole is 0.5mm-0.8mm;</p> <p>Recommended distance between the two stamp holes is 0.2-0.3mm</p>	
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 JLCPCB, we will add 5mm edge rails on both sides by default.	

## READY TO GET STARTED?

[See PCB Fab FAQs](#)
[Get Instant Quote](#)

### COMPANY

[About JLCPCB](#)

### SUPPORT

[Help Center](#)

### NETWORK SITES

[EasyEDA](#)

[News](#)

[How we work](#)

[Quality Management](#)

[Security](#)

[Certifications](#)

[Shipping Info](#)

[How To Order](#)

[How To Track](#)

[Contact Us](#)

[Support Team](#)

[LCSC components](#)

[ZXHPCB\(Mass production\)](#)

CONNECT WITH US



© 2023 JLCPCB.COM All Rights Reserved. [Privacy Policy](#) [Terms & Conditions](#)