


Mid-Autumn Festival Holiday Schedule!

JLPCB will have 3 day's holiday from September 13 to 15. Orders can be placed online during the holidays and will be processed on Sep 16. Please kindly schedule your orders ahead of time.

Ship to  SPAIN / EUR Need Help? (<https://support.jlpcb.com/>)



[HOME \(/\)](#) [CAPABILITIES \(/capabilities/Capabilities\)](/capabilities/Capabilities) [ABOUT US \(https://support.jlpcb.com/category/15-get-started\)](https://support.jlpcb.com/category/15-get-started)

[SUPPORT \(https://support.jlpcb.com\)](https://support.jlpcb.com) [RESOURCES ▼](#)


[CALCULATE \(/quote\)](/quote)

[CART¹ \(/client/index.html#/cart\)](/client/index.html#/cart)

[ACCOUNT \(/client/index.html#/order\)](/client/index.html#/order)

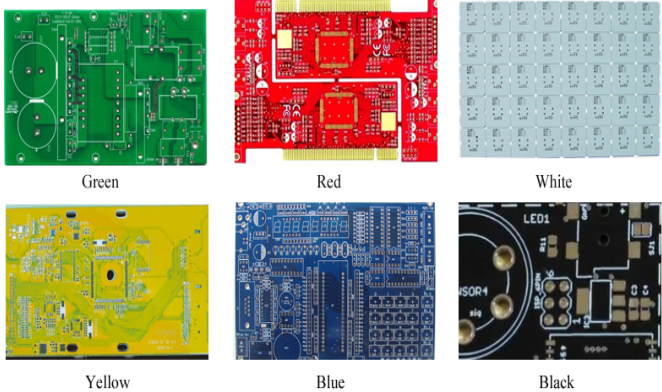
rndmnkiii | [Logout](#)

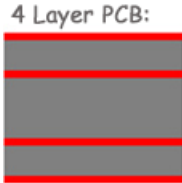

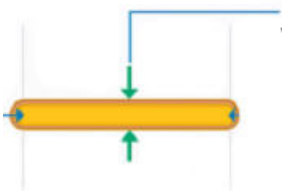
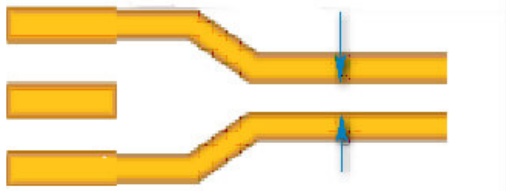
[Home \(/\)](#) >> Capabilities

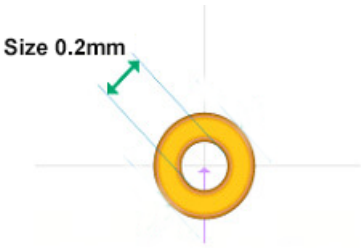
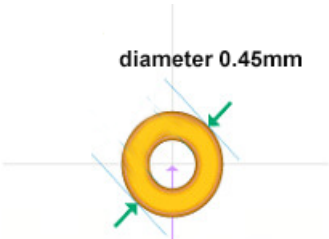

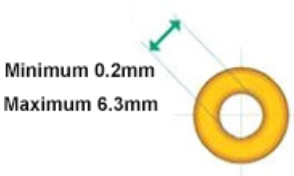
Item	Capability	Details	
Layer	1-6 layer	1-6 copper layers PCB prototypes (Don't support Blind/Buried Vias)	
Controlled Impedance PCB	4/6 layer	No extra charge and only accept our default layer stack-up	Controlled Impedance PCB Layer Stackup (https://jlpcb.com/client/index.html#/impedance) JLPCB Impedance Calculator (https://jlpcb.com/client/index.html#/impedanceCalculation)
Material	FR-4	FR-4 board material	




We use cookies to offer you a better experience. Detailed information on the use of cookies on this website is provided in our Privacy Policy (<https://support.jlpcb.com/article/59-privacy-policy>). By using this site, you consent to the use of our cookies.

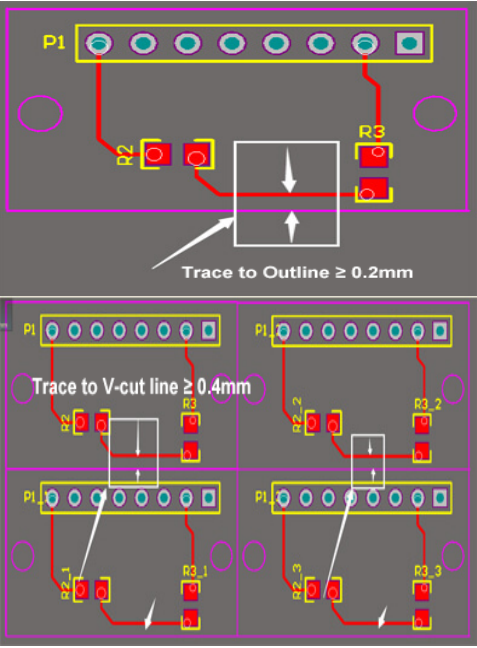
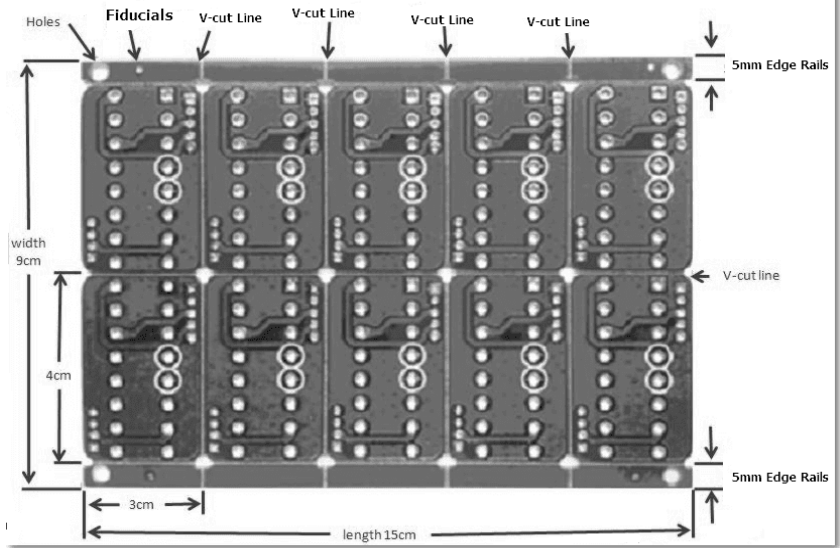
Accept

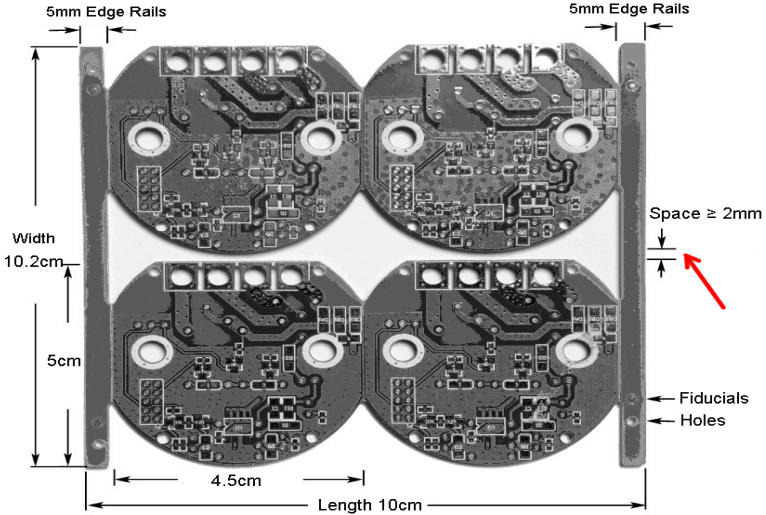
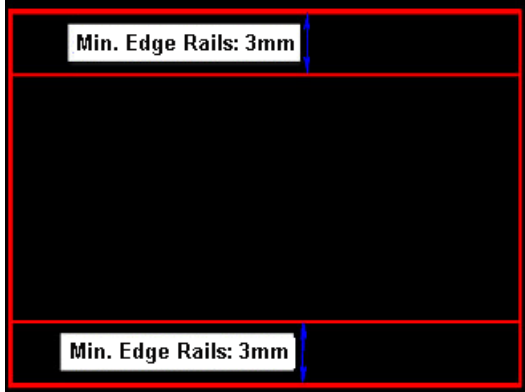
Max. Dimension	400x500mm	For Single&Double Layer, the maximum dimension is 400x500mm; For Multi Layer PCB, the maximum dimension is 300x300mm.	
Dimension Tolerance (Outline)	±0.2mm	±0.2mm for CNC routing, and ±0.2mm for V-scoring	
Solder Mask	LPI	Liquid Photo-Imageable Solder Mask is the most common mask type. No extra fee for different color of solder mask.	 <p>Green Red White</p> <p>Yellow Blue Black</p>
Thickness	0.4--2.0mm	0.4/0.6/0.8/1.0/1.2/1.6/2.0mm	
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/75um)	Finished outer layer copper weight is 1oz or 2oz	 <p>4 Layer PCB: Top Layer : 10Z/0.035mm Layer 2 Layer 3 Bottom Lauer : 10Z/0.035mm</p>
Finished Inner Layer Copper	0.5 oz (17um)	Finished inner layer copper weight is 0.5 oz only	 <p>4 Layer PCB: Top Layer Layer 2 : 0.5oz/0.017mm Layer 3 : 0.5oz/0.017mm Bottom Layer</p>
Min. Trace	3.5mil	For Single&Double Layer PCB, the minimum trace width is 5 mil; For Multi Layer PCB, the minimum trace width is 3.5mil.	 <p>Width 3.5mil</p>
Min. Spacing	3.5mil	For Single&Double Layer PCB, the minimum spacing is 5 mil; For Multi Layer PCB, the minimum spacing is 3.5mil.	 <p>Minimum spacing 3.5mil</p>

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	0.45mm	For Single&Double Layer PCB, the minimum Via diameter is 0.6mm; For Multi Layer PCB, the minimum via diameter is 0.45mm.	
Via To Trace	≥5mil	Minimum distance between via(plated holes) and trace is 5mil.	
Drill Hole Size	0.2--6.3mm	Min. drill size is 0.2mm, Max. drill size is 6.3mm.	
Hole Size Tolerance	±0.08mm	e.g. For the 0.6mm hole, the finished hole size between 0.52mm to 0.68mm is acceptable.	

Annular Ring	≥3mil	Annular ring surrounded by traces should be equal to or larger than 3mil	 <p>The diagram shows a yellow annular ring. Two horizontal purple lines are positioned above and below the ring. Blue arrows point from these lines towards the ring. The space between the lines is labeled '3mil'. Below the lines, the text 'Min. Annular Ring' is written.</p>
Min. Character Width	≥6mil	Characters width less than 6mil(0.153mm) will be unidentifiable.	 <p>The diagram shows the characters 'C' and '1' in a bold, olive-green font. A vertical blue dimension line to the right of the '1' indicates a height of '32mil'. A horizontal blue dimension line below the '1' indicates a width of '6mil'.</p>
Min. Character Height	≥32mil	Characters height less than 32mil will be unidentifiable.	 <p>The diagram shows the characters 'C' and '1' in a bold, olive-green font. A vertical blue dimension line to the right of the '1' indicates a height of '32mil'. A horizontal blue dimension line below the '1' indicates a width of '6mil'.</p>

Trace to Outline	$\geq 0.2\text{mm}$	<p>Ship as individual board(Routing): Trace to Outline $\geq 0.2\text{mm}$;</p> <p>Ship as panel with V-scoring: Trace to V-cut line $\geq 0.4\text{mm}$</p>	
Panelization without space	0mm		

Panelization with space	≥2mm	Make sure the space between boards should be ≥2mm, otherwise it will be hard to process for routing.	
Min. Edge Rails	3mm		
Copper Hatching with Pads	Hatch	We will apply Copper Hatching if your PCBs designed with Pads.	
Slot Drawing with Pads	Outline	Please use Outline to design if there are many non-plated (NPTH) holes.	

Protel/dxp Solder Layer	Solder Layer	Do not mistake the Paste Layer as Solder Layer.	
Protel/dxp Outline Layer	Keepout Layer/Mechanical Layer	Only choose one from Keepout Layer or Mechanical Layer as outline.	
Min. Half Hole Diameter	0.6mm	Half hole is a special technology, so half hole diameter should be greater than 0.6mm.	
Soldermask Bridge/ Blind and buried vias		Soldermask Bridge/Blind and buried vias are not available now	

PRODUCTS/SERVICES

Instant PCB Quote (/quote)
How do I place an order
(<https://support.jlcpb.com/article/21-how-do-i-place-an-order>)
How to order a stencil
(<https://support.jlcpb.com/article/25-how-to-order-a-stencil>)
Suggested Naming Patterns
(<https://support.jlcpb.com/article/29-suggested-naming-patterns>)
Free PCB Editor (<https://easyeda.com/>)
Online gerber viewer (<http://gerber-viewer.easyeda.com/>)
Altium PCB to gerber files
(<https://support.jlcpb.com/article/42-how-to-export-altium-pcb-to-gerber-files>)
Eagle PCB to gerber files
(<https://support.jlcpb.com/article/43-how-to-export-eagle-pcb-to-gerber-files>)
Kicad PCB to gerber files
(<https://support.jlcpb.com/article/44-how-to-export-kicad-pcb-to-gerber-files>)
Diptrace PCB to gerber files
(<https://support.jlcpb.com/article/46-how-to-export-diptrace-pcb-to-gerber-files>)

COMPANY

About Us
(<https://support.jlcpb.com/category/15-get-started>)
Our Team
(</client/index.html#ourTeam>)
Capabilities
(</capabilities/Capabilities>)
Shipping & Delivery
(<https://support.jlcpb.com/article/27-shipping-delivery>)
Payment Method
(<https://support.jlcpb.com/category/20-payment-invoice>)
Certifications
(<https://support.jlcpb.com/article/64-certifications>)
Privacy Policy
(<https://support.jlcpb.com/article/59-privacy-policy>)

SUPPORT

Support Center
(<https://support.jlcpb.com/>)
Contact Us
(<https://support.jlcpb.com/collection/16-contact-us>)

GROUP COMPANY

EasyEDA (<https://easyeda.com/>)
LCSC components
(<https://www.lcsc.com/?href=jlc-footer&source=jlc>)



JiaLiChuang (HongKong) Co., Limited, is the largest PCB prototype enterprise in China and a high-tech manufacturer specializing in quick PCB prototype and small-batch PCB production. With Over 10 years of PCB manufacturing experience, we're able to provide cost effective solutions that deliver the highest reliability PCB products to customers faster.

PROTOTYPE AND SMALL SERIES Reliable PCB Services

© 2019 JLCPCB.COM All Rights Reserved.

We use cookies to offer you a better experience. Detailed information on the use of cookies on this website is provided in our Privacy Policy (<https://support.jlcpb.com/article/59-privacy-policy>). By using this site, you consent to the use of our cookies.

Accept