



# Fusion PCB Specification

← Fusion Specification

New and returning users may [sign in](#)

Seeed Fusion Service offers one-stop prototyping service for PCB(printed circuit board) service, PCBA (PCB Assembly) service and other electronic and mechanical customized service (like CNC milling,3D printing, PCB layout service)

So easy to get PCBs online!

[Order Now](#)

## 1. PCB Gerber

The Gerber format is an open 2D binary vector image file format. It is the standard file used by printed circuit board (PCB) industry software to describe the printed circuit board images: copper layers, solder mask, legend, etc.

Gerber files should be inside a .rar or.zip archive with standard file extensions:

Extension	Layer
■ pcbname.GTL	Top Copper
■ pcbname.GTS	Top Soldermask
■ pcbname.GTO	Top Silkscreen
■ pcbname.GBL	Bottom copper
■ pcbname.GBS	Bottom Soldermask:
■ pcbname.GBO	Bottom Silkscreen:
■ pcbname.TXT	Drills
■ pcbname.GML/GKO	Board Outline
■ pcbname.GL2	Inner Layer2( for ≥4 layer)
■ pcbname.GL3	Inner Layer3( for ≥4 layer)

**Notes:**1. Gerber file must be the RS-274x format.

2. Drill file(pcbname.TXT) should be Excellon format.

3. Gerber file and Drill file(pcb name.TXT) must be put in the same file.

4. Board outline is must

## 2. PCB Specification for FR4-TG130

### Fusion Specification

Fusion PCB/PCBA Order Submission Guidelines

### Fusion PCB Specification

Fusion PCBA Specification

Fusion PCB Stencil Specification

Fusion Aluminum Board

Fusion FPC board

Fusion Advanced PCB Specification

### Contact Support

### Give feedback

### Knowledge Base

Assurance 2

Fusion Ordering FAQ 36

**Fusion Specification** 7

Fusion technical support 38

General 2

Ordering 8

Products' FAQ 82

Shipping 5

Tech Support 4

よくある質問(Fusion向け) 32

技術ガイド(Fusion向け) 27

標準規格書(Fusion向け) 8

All articles

**seedstudio**

Items	Description	Specs
		Unit: mm

seeedstudio · Feedback &amp; Ideas for seeed · Terms of Service &amp; Privacy Policy

## Board Dimension

## Min size

10mm\*10mm

Powered By UserVoice · Product Management Platform

If your board width is smaller than this size, you can make a bigger panel and use slots to separate the board.

## Max size

500\*500mm

## Available Board Layers

1-16 layers

## Available Board Qty

Min : 5pcs

Max: 8000pcs

## Dielectric Constant

4.2-4.7

## Dielectric Separation thickness

0.075 - 5.0

## Available Board Thickness

1-2 layers

0.6, 0.8, 1, 1.2, 1.6, 2, 2.5, 3

4 layers

0.8, 1, 1.2, 1.6, 2, 2.5, 3

6-8 layers

1, 1.2, 1.6, 2, 2.5, 3

10 layers

1.2, 1.6, 2, 2.5, 3

12 layers

1.6, 2, 2.5, 3

14 layers

2, 2.5, 3

16 layers

2.5,3

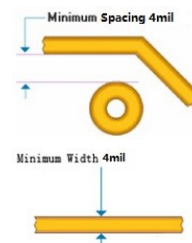
## Available Board Copper Weigh

1oz. 2oz. 3oz.

## Board Thickness Tolerance

± 10%

## Minimum trace spacing / width

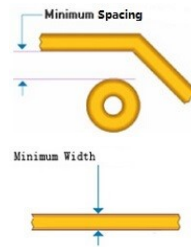


For 1oz, 4/4mil, 5/5mil,  
6/6mil

For 2oz, 10/10mil

For 3oz, 15/15mil

Minimum trace width in inner layers  
(for 4 layers board)



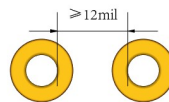
≥6mil

Minimum distance between trace and copper pour



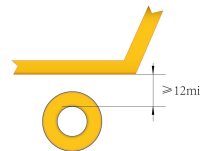
For 1oz ≥8mil  
For 2oz ≥12mil  
For 3oz ≥15mi

Minimum distance between vias (plated holes)



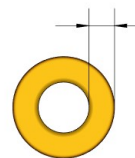
≥12mil  
Aim to prevent Ion migration

Minimum distance between via(plated holes) and trace



≥12mil  
Aim to prevent Ion migration

Annular Rings



≥0.152mm/6mil

Outer Layer Copper Thickness



0.035-0.07(1oz-2oz)

Inner Layer Copper Thickness

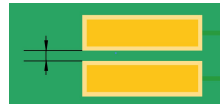


0.017(0.5oz)

Drilling Hole Diameter(Mechanical)

0.2 - 6.3mm

Width of Solder Mask Dam



Normal:

 $\geq 0.32\text{mm}$  for Green $\geq 0.35\text{mm}$  for Other colors

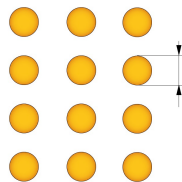
Limitation(need extra fee):

 $\geq 0.10\text{mm}$  for Green $\geq 0.13\text{mm}$  for Other colors

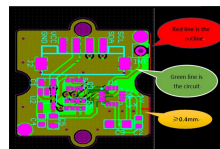
Diameter of Castellated Holes

 $\geq 0.50\text{mm}$ 

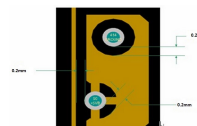
Size of BGA

For 6/6mil  $\geq 0.45\text{mm}$ For 5/5mil  $\geq 0.35\text{mm}$ For 4/4mil  $\geq 0.25\text{mm}$ 

Circuit to edge

 $\geq 0.3\text{mm}$ 

Minimum distance between inner trace and non-plated hole

 $\geq 0.2\text{mm}/7.87\text{mil}$ 

Minimum silkscreen height/trace width

height  $\geq 0.5842\text{mm}/23\text{mil}$ trace width  $\geq 0.1016\text{mm}/4\text{mil}$ 

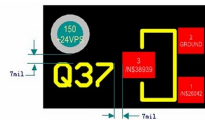
Perfect aspect ratio of silkscreen

1 : 5

Silkscreen colour

Silkscreen colour is **WHITE**  
when solder mask is Green,  
Red, Yellow, Blue, Black

Silkscreen colour is **BLACK**  
when solder mask is white



≥0.1524mm/6mil

Minimum distance between pad and silkscreen

Minimum milling slot width ≥0.8mm

Slot Tolerance(Mechanical) ±0.15mm

The minimum board dimension of V-CUT ≥70\*55mm

The maximum board dimension of V-CUT The V-cut line only is cut from edge to edge ≤380\*380mm

The minimum sub-board dimension of V-CUT ≥8\*8mm

PCB production time(build time) 3-14 working days

Aluminum Board Specification	
Layer:	1, 2
Available Board Thickness:	1mm, 1.2mm, 1.6mm, 2mm
QTY within:	min : 5, max:8000
Board Dimension:	min : 10mm*10mm
	max : 500*500mm
Available Board Colour:	White (Silkscreen is black)
	Black (Silkscreen is white )
Copper weight:	1oz
Surface Finish:	HASL, HASL lead free, ENIG, OSP, Hard Gold
Minimum Trace Width:	4mil

Minimum Trace Spacing:	4mil
Minimum Drilling Hole:	0.2mm
Production Lead Time:	5-11 business days

FPC Board Specification	
Layer:	1, 2
Available Board Thickness:	0.1mm,0.15mm
QTY within:	min:5,max:8000
Board Dimension:	max:100mm*100mm
Available Board Colour:	Yellow
Silkscreen:	White
Copper weight:	0.5oz
Surface Finish:	ENIG
Minimum Trace Width:	0.10mm
Minimum Trace Spacing:	0.10mm
Minimum Drilling Hole:	0.3mm
Production Lead Time:	5-7 business days

### 3.Panelization Rules:

Please [click here](#) for more details.

Order Now

This article was helpful · [Flag this article as inaccurate...](#)