

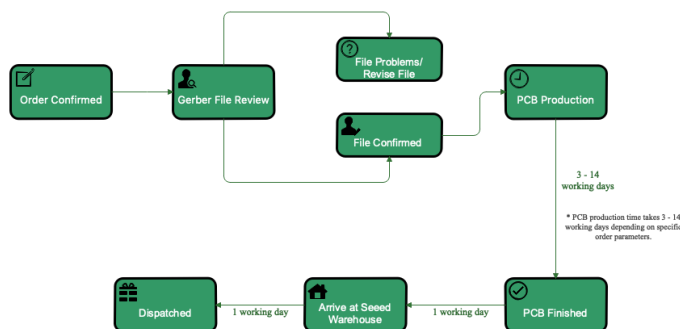


# Fusion PCB Specification

← Fusion Specification

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

## ■ PCB Order Flow Chart



## ■ 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

## Fusion Specification

### Fusion PCB Specification

Fusion PCBA Specification

Fusion PCB Stencil Specification

Fusion Aluminum PCB Specification

Fusion FPC (Flexible PCB) Specification

Fusion Advanced PCB Specification

Fusion Shipment Specification

Fusion Reference Data



**Contact Support**



**Give feedback**



**Knowledge Base**

Assurance 2

General 2

**Fusion Specification 8**

Fusion Service FAQ 17

Fusion Technical FAQ 19

Have a problem with PCB design? 7

How to generate PCB Gerber files? 11

Have a problem with Fusion orders? 11

Have a shipment problem with Fusion orders? 8

Bazaar Ordering 18

Products' FAQ 90

Shipping 5

Tech Support 4

よくある質問 (Fusion向け) 33

常见问题 2

技術ガイド (Fusion向け) 31

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

**4. Board outline is required**


標準規格書 (Fusion向け) 9

All articles



Seedstudio

▪ **PCB Specification for FR4-TG130**

Items	Description	Specs
		Unit: mm
Board Dimension	Min size	10mm*10mm Tip: 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	1-6 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: ≥0.32mm for Green ≥0.35mm for Other colors Limitation(need extra fee): ≥0.10mm for Green ≥0.13mm for Other colors
Diameter of Castellated Holes		≥0.50mm

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 $\geq 0.5842\text{mm}$ /23mil trace width $\geq 0.1016\text{mm}$ /4mil
Perfect aspect ratio of silkscreen		1 : 5
Silkscreen colour		Silkscreen colour is <b>WHITE</b> when solder mask is Green, Red, Yellow, Blue, Black  Silkscreen colour is <b>BLACK</b> when solder mask is white
Minimum distance between pad and silkscreen		$\geq 0.1524\text{mm}/6\text{mil}$
Minimum milling slot width		$\geq 0.8\text{mm}$
Slot Tolerance(Mechanical)		$\pm 0.15\text{mm}$
The minimum board dimension of V-CUT	The V-cut line only is cut from edge to edge	$\geq 70*55\text{mm}$
The maximum board dimension of V-CUT		$\leq 380*380\text{mm}$
The minimum sub-board dimension of V-CUT		$\geq 8*8\text{mm}$
		3-14 working days

PCB production time(build time)		* PCB production time takes 3 - 14 working days depending on specific order parameters.
---------------------------------	--	---

Aluminum Board Specification	
Layer:	1
Available Board Thickness:	1mm, 1.2mm, 1.6mm, 2mm
QTY	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 working days
The temperature of FPC	Low-temperature resistance: -40°C High-temperature resistance: 280°C

- **Panelization Rules:**

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

[Order Now](#)

Seeedstudio · Feedback & Ideas for seeed · Terms of Service & Privacy Policy

Powered By UserVoice · Product Management Platform