



# *Regular Custom PCB*

*Specification*

*&&*

*FAQ*

### Order-online

Regular Custom PCB Service: <https://www.elecrow.com/pcb-manufacturing.html>

Stencil: <https://www.elecrow.com/laser-cut-pcb-stencil-p-450.html>

To get the quotation via [service@elecrow.com](mailto:service@elecrow.com)

Premium PCB Service: <https://www.elecrow.com/premium-pcb-service-p-1061.html>

PCB Assembly: <https://www.elecrow.com/pcb-assembly.html>

### Contact us

[pcb@elecrow.com](mailto:pcb@elecrow.com)   [order@elecrow.com](mailto:order@elecrow.com)   [techsupport@elecrow.com](mailto:techsupport@elecrow.com)  
[service@elecrow.com](mailto:service@elecrow.com)   [qc@elecrow.com](mailto:qc@elecrow.com)

### Regular Custom PCB VS Premium PCB

This part will show you parameter comparison between regular custom PCB and Premium PCB. So you could have a better choice for your board.

### 4-layer Regular Custom PCB stack-up

If you have puzzle with the thickness, dielectric constant and copper weight of each layer for 4-layer PCB, you may find out the information you need in this part.

### Elecrow Regular Custom PCB Specification

**The PCB will be fully test.** PCB manufacturing ability could be found in this part, such as minimum holes size, minimum trace width and so on.

### FAQ of Regular Custom PCB

Some other frequently asked question will be list in this part. You could click on each question to find out the answer easily. More detail on next page.  
Please also read the file “**Reading Me before Ordering** ” which is in our ordering page to know more detail about our service.

### Panelizing Problem

If you want to know more about panelizing, please go this page.

<https://www.elecrow.com/download/How-to-Reduce-Cost-with-Panelizing-Service.pdf>

## FAQ of Regular Custom PCB

Please click on the each question to find out the answer easily.

- [How to place a new order?](#)
- [Is it possible to repeat a previous order? What should I do?](#)
- [I just place an order for PCB and I forget to choose the stencil, can I add a stencil into this order?](#)
- [Could you make the board with .brd file, .pcb file, .lay6 file or .pdf file?](#)
- [I just have the schematic diagram, could you make gerber according to this?](#)
- [Which format of gerber you could accept? Could you please make the PCB without outline in the gerber?](#)
- [Is “custom shape” allowed for PCB outline?](#)
- [Boards are outline milled, right?](#)
- [How to name the inner layer for 4-layer PCB?](#)
- [My PCB just have 3-layer and one inner layer is empty, could I choose 4-layer PCB for it?](#)
- [What is the inner copper weight of 1oz and 2oz 4-layer PCB, 6-layer PCB and 8-layer PCB?](#)
- [Can you make the holes and characters which are in Paste Layer in the stencil?](#)
- [Will you make the GTP and GBP both in one stencil if the area is enough? Or make them separately into two stencils?](#)
- [How can I order this board? PCB without any copper with mask both sides \(and silkscreen\)?](#)
- [I want to make my board as 1 layer with mask both sides, is it possible? Or it could just be with mask one side?](#)

## FAQ of Regular Custom PCB

Please click on the each question to find out the answer easily.

- [I would like to order the PCBs without any surface finish at all, just the bare board/pure copper. Is it possible?](#)
- [There is no need to make the board with solder mask, just exposed copper for the whole panel, what should I do?](#)
- [I don't want to make silkscreen for my PCB, could I leave the silkscreen empty?](#)
- [Could you make yellow board with white silkscreen, not black?](#)
- [I want to make PCB with colorful silkscreen, like red, blue and so on. Could you do that?](#)
- [Could you please don't put any ID numbers in the board?](#)
- [Is there a way to provide the information regarding non-plated through holes to you in future designs?](#)
- [Could you make the PCB without NC drill hole layer?](#)
- [When will you ship my order with 24H/48H/72H/96H rush service?](#)
- [If I place an order for 48H rush on Thursday, will you ship my order on Sunday?](#)
- [Could you make rectangle hole or oval hole for PCB?](#)
- [What is the depth of v-groove/v-cut?](#)
- [What is the minimum width of Technological Edge that you can make?](#)
- [What is castellated hole/half hole?](#)
- [I want to make my board with golden finger with beveled edge, but there is no option for this, how to do that?](#)

## Regular Custom PCB VS Premium PCB

Parameters	Regular Custom PCB(online-ordering)	Premium PCB service(quote via <a href="mailto:service@elecrow.com">service@elecrow.com</a> )
Material	FR-4 TG130	FR-4/FPC/Rigid-Flex/Rogers/Aluminum Substrate/Copper Substrate/TG150/ TG170, etc
Layer	1-2-4-6-8	1-28
Thickness	0.6mm/0.8mm/1.0mm/1.2mm/1.6mm/2.0mm	0.4mm-6.0mm
Solder Mask Color	green/red/yellow/blue/white/black/purple/matte black	
Silkscreen Color	Not allowed to specify	Could be specified
Copper Thickness	1oz, 2oz	0.5oz-12oz
Surface Finish	Hasl/Hasl(lead free)/Immersion Gold	Hasl/Hasl(lead free)/Immersion Gold/Carbon /Hard Gold(Plated Gold)/Immersion Silver
Min trace /trace spacing	0.15mm/6mil	3mil
Min drilling hole	0.3mm(diameter)	0.2mm(diameter)
Min BGA size	0.35mm(diameter)	0.2mm(diameter)
Castellated hole	Support	Support
Beveled edge	Support	Support
Golden finger	Support	Support
Buried/Blind hole	Not support	Support
Resign Plug hole	Not support	Support
Specified stack-up	Not support	Support

4-layer Regular Custom PCB stack-up

<p>Finish Thickness: <b>0.6mm±0.10mm</b></p> <div><div>GTL</div><div>Prepreg</div><div>Core (with two inner copper layer)</div><div>Prepreg</div><div>GBL</div></div> <div><div>1oz/0.035mm</div><div>2116*1</div><div>G1 0.5oz/0.0175mm 0.3mm(with copper)</div><div>G2 0.5oz/0.0175mm</div><div>2116*1</div><div>1oz/0.035mm</div></div>	<p>Finish Thickness: <b>1.2mm±0.10mm</b></p> <div><div>GTL</div><div>Prepreg</div><div>Core (with two inner copper layer)</div><div>Prepreg</div><div>GBL</div></div> <div><div>1oz/0.035mm</div><div>7628*1</div><div>G1 0.5oz/0.0175mm 0.8mm(with copper)</div><div>G2 0.5oz/0.0175mm</div><div>7628*1</div><div>1oz/0.035mm</div></div>
<p>Finish Thickness: <b>0.8mm±0.10mm</b></p> <div><div>GTL</div><div>Prepreg</div><div>Core (with two inner copper layer)</div><div>Prepreg</div><div>GBL</div></div> <div><div>1oz/0.035mm</div><div>2116*1</div><div>G1 0.5oz/0.0175mm 0.5mm(with copper)</div><div>G2 0.5oz/0.0175mm</div><div>2116*1</div><div>1oz/0.035mm</div></div>	<p>Finish Thickness: <b>1.6mm±0.10mm</b></p> <div><div>GTL</div><div>Prepreg</div><div>Core (with two inner copper layer)</div><div>Prepreg</div><div>GBL</div></div> <div><div>1oz/0.035mm</div><div>7628*1</div><div>G1 0.5oz/0.0175mm 1.2mm(with copper)</div><div>G2 0.5oz/0.0175mm</div><div>7628*1</div><div>1oz/0.035mm</div></div>
<p>Finish Thickness: <b>1.0mm±0.10mm</b></p> <div><div>GTL</div><div>Prepreg</div><div>Core (with two inner copper layer)</div><div>Prepreg</div><div>GBL</div></div> <div><div>1oz/0.035mm</div><div>7628*1</div><div>G1 0.5oz/0.0175mm 0.6mm(with copper)</div><div>G2 0.5oz/0.0175mm</div><div>7628*1</div><div>1oz/0.035mm</div></div>	<p>Finish Thickness: <b>2.0mm±0.10mm</b></p> <div><div>GTL</div><div>Prepreg</div><div>Core (with two inner copper layer)</div><div>Prepreg</div><div>GBL</div></div> <div><div>1oz/0.035mm</div><div>7628*1</div><div>G1 0.5oz/0.0175mm 1.5mm(with copper)</div><div>G2 0.5oz/0.0175mm</div><div>7628*1</div><div>1oz/0.035mm</div></div>

	Thickness	Dielectric constant
FR4		4.5
2116 Prepreg	0.10mm-0.12mm	4.0
7628 Prepreg	0.17mm-0.19mm	4.2

**Elecrow Regular Custom PCB Specification**

Description	Model	Size
Minimum trace spacing/width		$\geq 0.15\text{mm}$ for 1oz; $\geq 0.25\text{mm}$ for 2oz
Minimum pad spacing		$\geq 0.15\text{mm}$
Minimum distance between PCB outline and trace		$\geq 0.7\text{mm}$ (PCB thickness from 0.6mm to 1.6mm); $\geq 1\text{mm}$ (2mm thickness PCB)
Minimum spacing between trace and pad		$\geq 0.15\text{mm}$
Minimum via/through hole		$\geq 0.25\text{mm}$ for vias; $\geq 0.3\text{mm}$ for through holes
Spacing between holes		$\geq 0.35\text{mm}$
Annular ring		$\geq 0.15\text{mm}$ for 2-layer PCB; $\geq 0.2\text{mm}$ for 4-layer PCB.

<p>silkscreen height/width; spacing between silkscreen letters</p>		<p>Height <math>\geq 0.8\text{mm}</math>;</p> <p>Width <math>\geq 0.15\text{mm}</math>;</p> <p>Spacing between silkscreen letters <math>\geq 0.15\text{mm}</math></p>
<p>Minimum width of slot (rectangle hole/oval hole); Spacing between slots</p>		<p>Width <math>\geq 0.8\text{mm}</math>;</p> <p>Spacing between slots <math>\geq 0.8\text{mm}</math></p>
<p>Width of IC solder mask dam</p>		<p><math>\geq 0.25\text{mm}</math></p>
<p>BGA size/Minimum BGA pad spacing</p>		<p>BGA size <math>\geq 0.35\text{mm}</math>;</p> <p>BGA pad spacing <math>\geq 0.15\text{mm}</math></p>
<p>Diameter of castellated hole(half hole)</p>		<p><math>\geq 0.6\text{mm}</math></p>



## FAQ of Regular Custom PCB

- **How to place a new order?**

Please have a look at the file “Reading Me before Ordering” which is in our ordering page.

- **Is it possible to repeat a previous order? What should I do?**

Sure, there is no problem to repeat a previous order.

Please place your order in our online page with an empty file and leave us a comment about the previous order number.

- **I just place an order for PCB and I forget to choose the stencil, can I add a stencil into this order?**

Sure. We suggest you to click here <https://www.elecrow.com/laser-cut-pcb-stencil-p-450.html> to place an order for stencil and leave us a comment that ship it with your PCB order number

- **Could you make the board with .brd file, .pcb file, .lay6 file or .pdf file?**

For the file format of .brd, .pcb and .lay6, we suggest you to convert it into gerber. We need gerber file to manufacture PCB.

For .pdf file, we are sorry we could not accept this file to produce PCB.

- **I just have the schematic diagram, could you make gerber according to this?**

Please send us the schematic diagram and requirement via [service@elecrow.com](mailto:service@elecrow.com). Our engineer will help you with this problem.

- **Which format of gerber you could accept? Could you please make the PCB without outline in the gerber?**

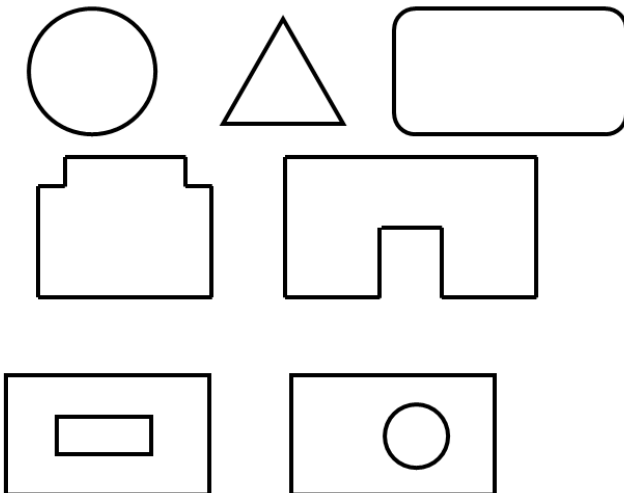
Gerber “RS-274-X” is the sole format for fabrication.

The PCB outline is necessary in production. Please make sure it is good in your gerber.

- **Is “custom shape” allowed for PCB outline?**

Of course they could be the shape like your gerber shown.

Here is the example as following.



- **Boards are outline milled, right?**

Yes, the outline and all the GML layer will be milled or add V-groove following the lines.

The V-cut line can be added on the GML/GKO layer.

The tolerance of outline is  $\pm 0.2\text{mm}$  from the middle of the outline.

- **How to name the inner layer for 4-layer PCB?**

Here is the better naming as following.

G1/Inner1-----the second layer;

G2/Inner2-----the third layer

Sometime we find the inner layer named “ly2”, “ly15” or “GND”, “Power”, that make us feel puzzle for we are not sure which is the second layer or the third layer. We suggest you to leave us a comment about stack-up when you order.

- **My PCB just has 3-layer and one inner layer is empty, could I choose 4-layer PCB for it?**

That is OK to choose 4-layer PCB for it. Please leave us a comment about which layer is empty and make sure it is a 3-layer board.

- **What is the inner copper weight of 1oz and 2oz 4-layer, 6-layer and 8-layer PCB?**

For multilayer board with 1oz outer copper, inner copper is 0.5oz.

If outer copper is 2oz, inner one is 1oz.

- **Can you make the holes and characters which are in Paste Layer in the stencil?**

We skip through-hole component pads and all drills (holes) by default except the fiducial, so you need to leave a message to us if you want to make the holes of the GTP and GBP.

Besides, we couldn't make the characters in the stencil.

- **Will you make the GTP and GBP both in one stencil if the area is enough? Or make them separately into two stencils?**

We will make the two paste layer in one stencil if the area is enough. If you want to make them separately into two stencils, please order one more stencil from here <https://www.elecrow.com/laser-cut-pcb-stencil-p-450.html> and leave us a comment in your order.

- **How can I order this board? PCB without any copper with mask single side/both sides (and silkscreen)?**

Please don't put the copper layer in your gerber or keep the copper layer empty.

Please choose “1 layer” or “2 layer” for your board. And leave us a comment about the requirement to avoid the engineering problem. Send us email about this would be also OK.

- **I want to make my board as 1 layer with mask both sides, is it possible? Or it could just be with mask one side?**

We will make PCB with mask one side in default.

If you need to make 1 layer with mask both sides, please leave us a comment in your order.

- **I would like to order the PCBs without any surface finish at all, just the bare board/pure copper. Is it possible?**

That is no problem. Please choose any of the option “surface finish” and leave us a comment about your requirement to avoid making the wrong board. You could also send us email about this.

- **There is no need to make the board with solder mask, just exposed copper for the whole panel, what should I do?**

Please don't put the solder mask layer in your gerber or keep the solder mask layer empty.

Or make the solder mask layer full with exposed copper in your gerber.

Then leave us a comment about this requirement or send us email.

➤ **I don't want to make silkscreen for my PCB, could I leave the silkscreen empty?**  
That would be better to leave the silkscreen empty.

➤ **Could you make yellow board with white silkscreen, not black?**  
Yes, please leave us a comment to let us know this when you order the yellow board.

➤ **I want to make PCB with colorful silkscreen, like red, blue and so on. Could you do that?**  
Sorry, you may try this by using our Premium PCB service. Please send us the gerber and requirement to get a quotation via [service@elecrow.com](mailto:service@elecrow.com).

➤ **Could you please don't put any ID numbers in the board?**  
Sure. Our company would not put any extra ID numbers in PCB.

● **Is there a way to provide the information regarding non-plated through holes to you in future designs?**

We will plate all the holes by default. If you need some holes not be plated, please name the plated holes/via layer as PCBname-PHT.txt and name the non-plated holes layer as PCBname-NPTH.txt.

**PS:** For holes display on the GML/GKO layer, we will fabricate it as Non-plated default. If it need to fabricate as plated holes, please leave a comment.

● **Could you make the PCB without NC drill hole layer?**  
That is OK to do that if there is no need to make PCB with NC drill hole layer.  
Please leave us a comment.

➤ **When will you ship my order with 24H/48H/72H/96H rush service?**

Please make sure that you place the order before 17:30 p.m.

Here is the list of rush service.

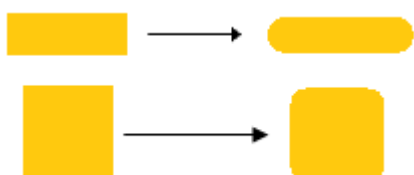
<i>Order day</i>	<i>In production</i>	<i>24H rush ship</i>	<i>48H rush ship</i>	<i>72H rush ship</i>	<i>96H rush ship</i>
<b>Feb. 1<sup>st</sup>.</b>	Feb. 2 <sup>nd</sup> .	Feb. 3 <sup>rd</sup> .	Feb. 4 <sup>th</sup> .	Feb 5 <sup>th</sup> .	Feb 6 <sup>th</sup> .

➤ **If I place an order for 48H rush on Thursday, will you ship my order on Sunday?**

Sorry, all rush orders will be delayed one day if this order should be shipped on Sunday. If the day is legal holiday, the order should be postponed after the holiday. You can refer our website announcement to make a reasonable arrangement.

● **Could you make rectangle hole or oval hole for PCB?**

We will make the square hole and oval hole as the right part shown, the right angle will be make as filleted corner. The radio for the corner is about 2.0 mm.



- **What is the depth of v-groove/v-cut?**

It is about 0.3mm.

- **What is the minimum width of Technological Edge that you can make?**

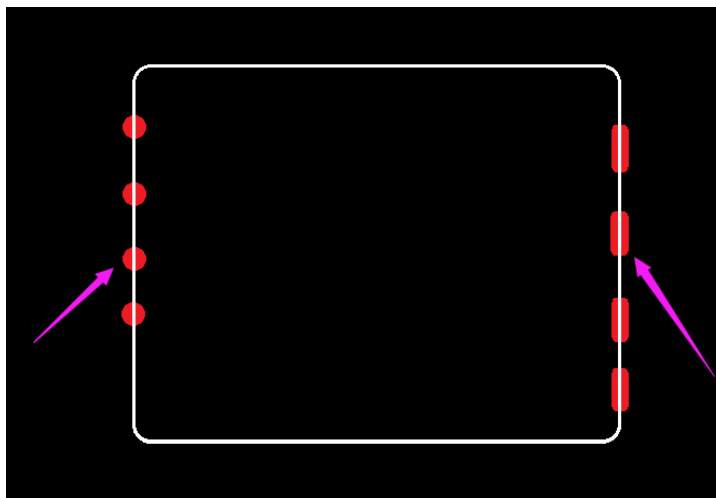
The minimum width of Technological Edge is 0.3mm.

- **What is castellated hole/half hole?**

The castellated holes as the picture shown, which are also plated holes.

If these holes on the board edge are non-plated, they will not be regarded as castellated hole.

If you need to make this in your board, please show us in your gerber and choose “castellated hole” in “Yes”. Then we will have a check and make it.



- **I want to make my board with golden finger(and beveled edge), but there is no option for this, how to do that?**

If what you need is just the board with golden finger without beveled edge, it is better to choose “surface finish” with “Immersion Gold” and leave us a comment. Then we will do that.

If the board should be with golden finger and beveled edge, it will cost extra fee and please contact us via email to pay for extra. Or leave us a comment. Then we will contact you as fast as possible.

**Elecrow Support Team**

**Feb., 2019**