

# Instructions for ordering

This description is aiming at explaining some confusion that our users have during placing the orders and some frequent problems they encountered in the past. It contains some routine internal operations JLCPCB will make when we receive the order. In order to avoid unnecessary trouble, please read those instructions carefully when ordering.

## Instructions for PCB orders

### 1. Solder-resistance bridges

To do solder-resistance bridges, usually, the spacing between the pads /pins needs to be 0.254 mm at least, and special notes are required when ordering such as : solder mask bridge is required. Or it will be ignored easily.

### 2. The English descriptions and other attachments in the zip file

The English descriptions and other statements in your zip file, such as some PDF files, DXF files, excel, etc., will be ignored and the PCB will be done according to the Gerber file provided only. So please make sure that the correct parameters are chosen and all those needed are converted into your Gerber file when you place your order. If any important, please kindly add it to the remark filed when you place your order.

### 3. PCB file and GERBER file in the zip file at the same time

If there are both PCB files and GERBER files in your zip file , we will follow the Gerber file and ignore the PCB file.

### 4. Plated edges/ slots( no longer than 6mm)

If there is any edge plating needed, or any plated slots are needed, please make sure that is not longer than 6 mm, and make a note about it when you place your order, or we can not make it for you. If there is no any note about it, then we will make it with the normal process.



### 5. Issues about software incompatible

Since different clients will use different software to design the board, and the software we process the Gerber data is also not the same as the software used for designing, so sometimes the software incompatibility problem arises. Since there is no warning about the issue when importing the data into the software, so we have no way to notice it and thus cannot confirm with our customers. Therefore, we are responsible for half of the order problems caused by software incompatibility.

### 6. Markings on base materials

For the sake of security, there are some markings on the base material, which is based on the base material itself, and we have no way to remove it. We have no idea about which PCB will be with such kind of markings, so your PCB will also have the chance to have the markings. Please kindly check whether it is acceptable before you place your order.

## **7. Designs about slots/CUT OUT/ millings/ v-cut**

Please kindly make sure that the v-cut lines, cut outs, millings and slots are in the same layer with the board outline. If it is not in the same layer with the board outline, it will be missed. So please kindly check it before you place your order. If it is missed due to they are not in the same layer with the board outline, we will not be responsible for it. (If the slots are to be plated, it needs to be with the drill holes in the same layer, or it will be missed easily).

## **8. Gerberview**

As to the preview / Gerberviewer on the website, it is only for the purpose of reviewing the file quickly before paying for your order. But sometimes, there might be something wrong on the website to display it EXACTLY, so you had better to check your file carefully when you place your order if there is any doubt.

## **9. Silkscreen/ texts**

To make the texts printed clearly, the width for the texts (filled lines) need to be 0.15 mm with a height of 0.8 mm. Basically, we will not check the silkscreen for you and will not modify it for you so that it will be clearly printed, so you had better to check whether it can meet the requirement or not before you place your order.

Besides, basically, for those solder mask openings area, there will be no texts printed, if you want to print silkscreen on those areas, please kindly make a note about it when you place your order.

## **10. 4 Layer PCB**

Basically, we don't make 3 layer PCB, if your order is a 4 layer and there is one inner layer only, we will process it with the 4 layer process directly and will confirm with you again. So please check whether there is any inner layer miss or not when you place your order.

## **11. About repeat order**

Usually the repeat order is directly produced according to the production file for your previous order, and we will not make any change to the production file anymore. So make sure you don't leave any note related to order changing when you place your order. The repeat order will no longer be manually checked by our engineers.

Note: Engineering fee will still charge by system since our engineers need to re-panelise the boards manually.

## **12. About the items in the board outline**

Basically, the board outline is to show what your PCB looks like when it is finished. So all those need to be cut with the board outline should be added into the layer with the board outline, such as v-cut lines, slots, milling, cut outs. Please remove those unnecessary to avoid any confusion.

## **13. About file generated from Easyeda**

Easyeda is a free online tool that we provide to design the PCB, and you can place your order on JLCPCB easily and quickly. But if there is manufacturer error due to the design error, we may not be responsible for that.

## **14. About JLCPCB panel**

Generally, if you choose JLCPCB to panel the PCB for you when you place your order, basically, we will panel your order with V-cut, but we panel those PCB with rectangle and round shape only.

## **15. About the remark field**

There is a remark field when you place your order so that you can leave a note in case any import, but it is not recommended to use this option since all the order with an English note will take a longer time to get it through the audition process.

## 16. About the board that larger than 200mm\*250mm

About the board which larger than 200mm\*250mm, since the board is so large, and we will try our best to make that. But if the good board is not enough for the quantity that you order in the website, due to huge cost of production, we will send the good boards and refund the difference to you.

## 17. Design about the thickness of board outline /slots/CUT OUT/ millings

Usually, we suggest our clients design 0.15mm for the thickness of board outline, slots, cut out, millings. But if the thickness of the board outline, slots, cut out, millings are bigger than 0.15mm, we will follow the center of the line to make the board outline, cut out, millings.



## 18. About the board within or equals to 50mm\*50mm

For the small board within or equals to 50mm\*50mm, since the size of the board is small and it might hard to mill perfectly during the milling process. Usually, we will suggest you panelise the small board to avoid this issue.

Other instructions:

### 1. About order cancellation

For those order that there are some layers missed, such as no solder mask layer, no board outline and drilling, we will cancel the order directly. If there is no solder mask or drill is needed, etc, you had better to make a note about it when you place your order, or it will be canceled directly during the audit process.

### 2. About orders beyond our capabilities

For orders that do not meet our capabilities, we will cancel the order directly and will send e-mail to inform you about the reason. So please kindly check your file carefully before you pay for your order to save the lost due to the cancellation.

### 3. About stencil production file

Basically, you choose JLCPCB to panelize the PCB for you, if there is no stencil order with, we will not make the production file for the stencil since the engineers who make the production file do not know how to make the production file for the stencil.

#### **4. About Removing Order Number:**

For those orders choose remove order number service or specify a location service, usually we will follow clients' instructions to make the boards. Just in case, we also suggest clients to choose Confirm production service which can double check it .

Generally, it will not affect the function of the boards if JLCPCB miss to remove the order number or put it by random, in this case, we will refund the remove order number fee.

### **Instructions for Stencil order**

#### **1. Stencil thickness**

Basically, there is no option to choose the thickness for the stencil on the website, and our engineers will choose the thickness for you according to your file. If there is any specific thickness required, please kindly make a note about it when you place your order.

#### **2. Tooling holes**

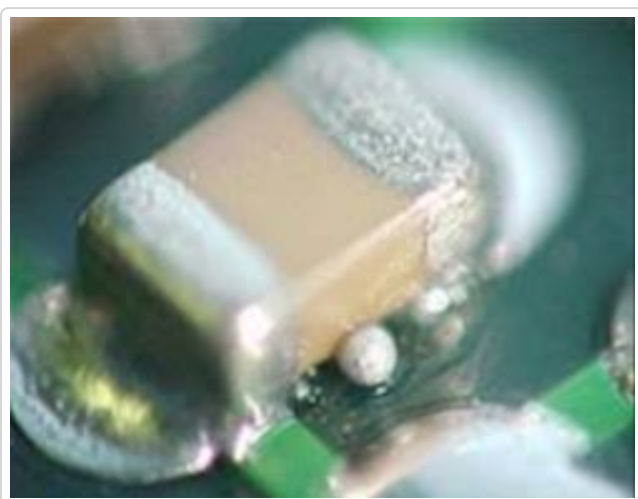
If there are tooling holes needed for your order, please kindly ensure that it is stuffed circles in the solder paste layer, or we will not make it for you.

#### **3. Solder beading treatment**

Solder beading, a defect that can result in short circuits, generally is related to an excessive solder paste deposit that, because of its lack of "body," is squeezed underneath a discrete component and then becomes a solder bead. The most common place to see solder beads are at the side of a chip components like resistors and capacitors.

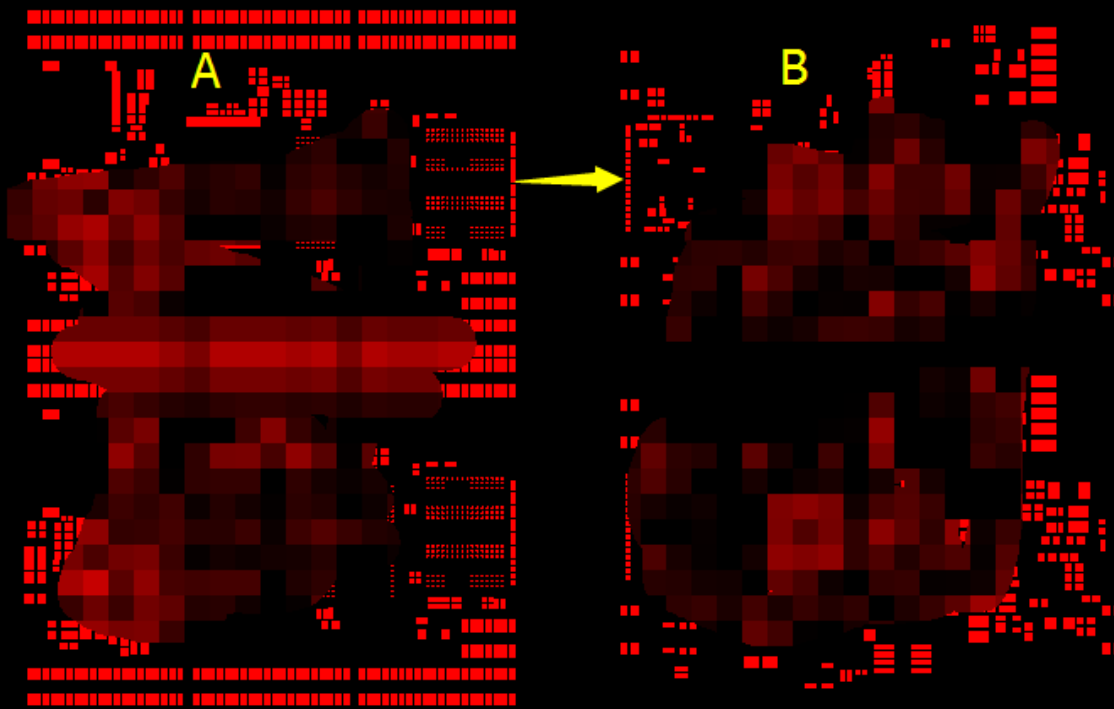
At JLCPCB, the stencil aperture for components (except for diode) larger than 0805 will be slightly reduced from the pad size like below image to avoid the solder beads.

Note: If you don't want to apply this solder beading treatment for your stencil, please make a remark when you place an order.



#### **4. About the spacing for different designs in your stencil file**

Basically, if the size for the stencil is big enough to add different designs on the stencil, we will not change the space between your designs (For 2 designs or more in the same stencil file). But in the case that the stencil is not big enough, we may adjust the spacing for those designs so that it can be added on the stencil. If there is no change can be made to the panel, please kindly make a note about it when you place your order.



Do not change the space between a and b

## 5. About the direction of stencil

Basically, if there is no any note about where to how to put the cutting on the stencil, the long side of the PCB will be on the same side with the long side of the stencil automatically.

### Other instruction

#### 1. Complaints about the reorder order

Please kindly check or test the order when you get them, should there is any issue, once the issue is confirmed, if it is the issue from JLCPCB, we will offer refund for the order or remake. If there is no any feedback and the order is reordered again, should there is any issue with the order that order the first time and the reorder one, we may bear for the loss for the order that order the first time only.

[Still need help? Contact Us \(/contact\)](#)

*Last updated on April 20, 2021*