



Q: I used the CAM file on your website, why there isn't drill layer in the gerber file?

A: Display all the layers before you exporting the gerber file from the eagle file.

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

A: 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**

Q: Boards are outline milled, right?

A: 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.

ps: PCB panelize is only for the same boards panelize, and PCB panelize is not available for special offer green PCB and 4 layer PCB.

Q: Are blind and buried vias supported?

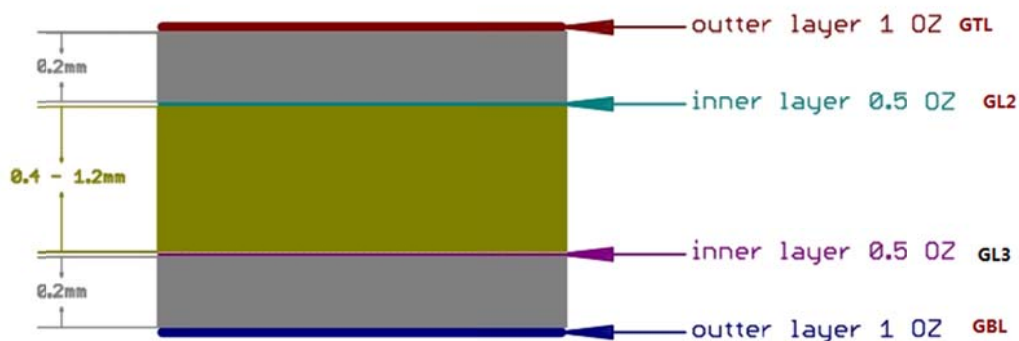
A: Blind vias are not available.

Q: What the minimum space or slot in the PCB can you fabricate?

A: The diameter of the milling cutter is 0.8mm. So the minimum space or slot is 0.8mm.

Q: What is the stack up for your 4 layers pcb?

A: Please look at the following picture and mark the layers as GTL, GL2, GL3 and GBL to avoid any misunderstanding.



Q: What is the minimum recommended annular ring size for your 2 layer (color free) service?

A: Please have a look at the following picture.



Q: What the minimum distance between two panelized PCBs?

A: For PCB thickness 0.8mm or 1.0mm, the minimum distance between sub-boards is 0.6 to 0.7 mm. For PCB thickness 1.2 or 1.6mm, the minimum distance between

Make your making more easy!

sub-boards is 0.7 to 0.8 mm.

ps1: The milling cutter can only run straight line. And the V-groove straight line should through the whole board.

ps2: For larger quantities (over 200pcs), we will combine your PCBs automatically to a panel. If you do not want it or want to define your own panel, please leave a comment.



www.elecrow.com