

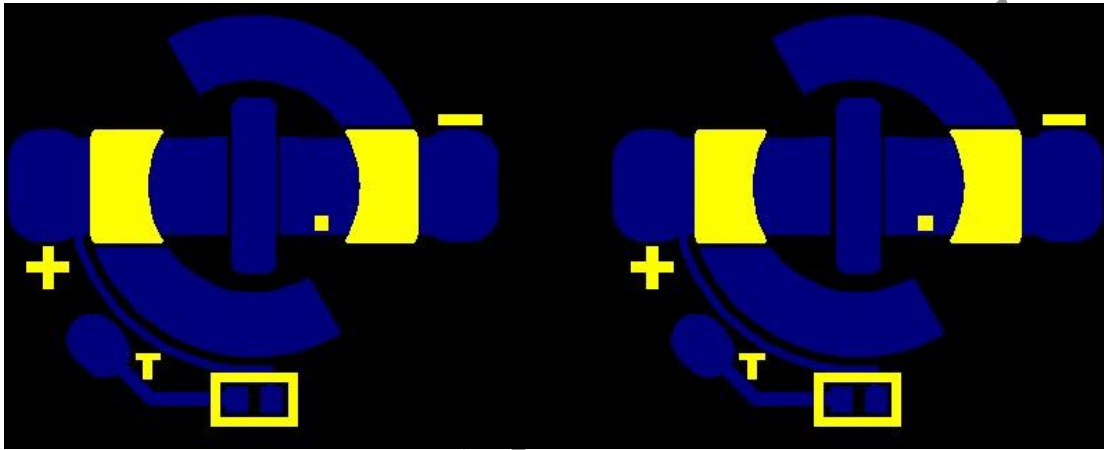


SinkPAD Board Manufacturing Process

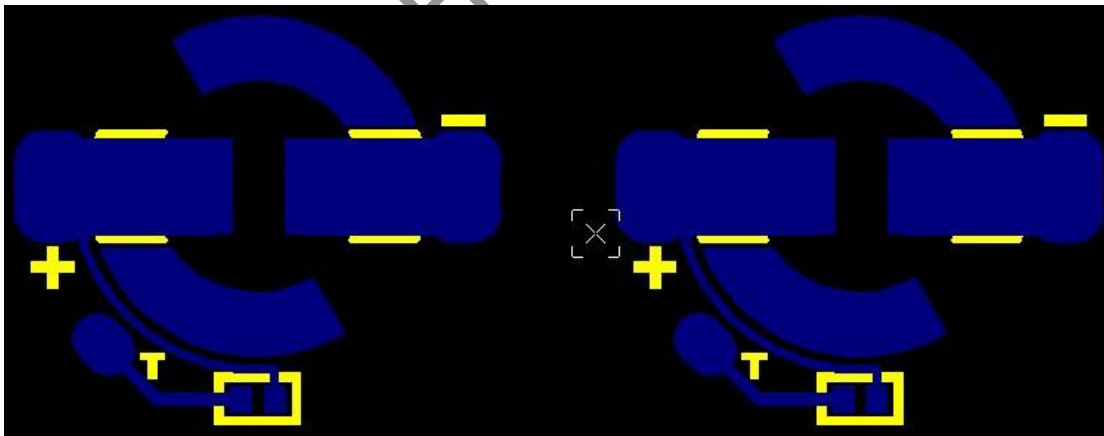
First of all, we need to understand there're two major parts of [SinkPAD board](#) ([SinkPAD PCB](#)): Top trace layer and copper core/substrate. Then you can see basic manufacturing process of SinkPAD board (PCB) bellowing:

1) Prepare the manufacturing files.

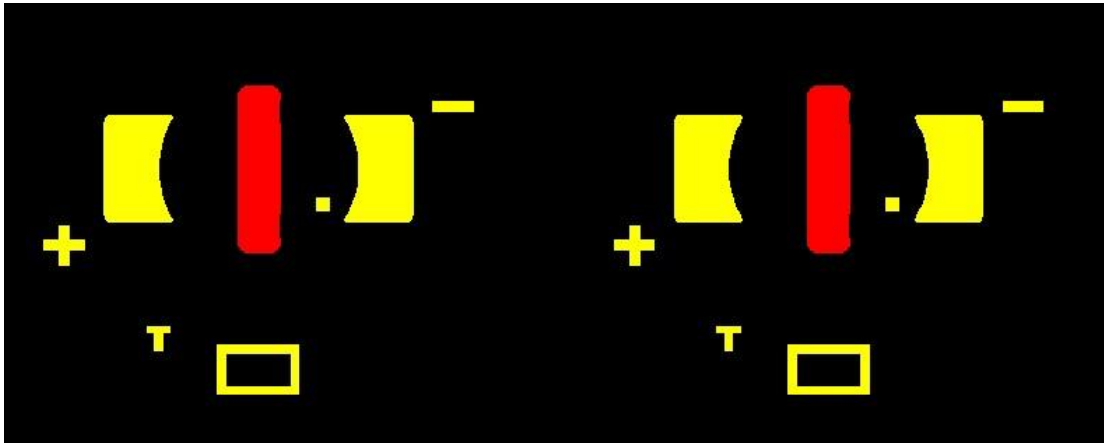
To make [SinkPAD board](#) ([SinkPAD PCB](#)), the major thing is to separate the copper core and trace, from one Gerber layer, to two different layers. On one layer, there's only trace but without center heatsink PAD of LED and that layer will be named as trace layer; and on another layer, there's only heatsink PAD, and that layer will be named as copper core layer.



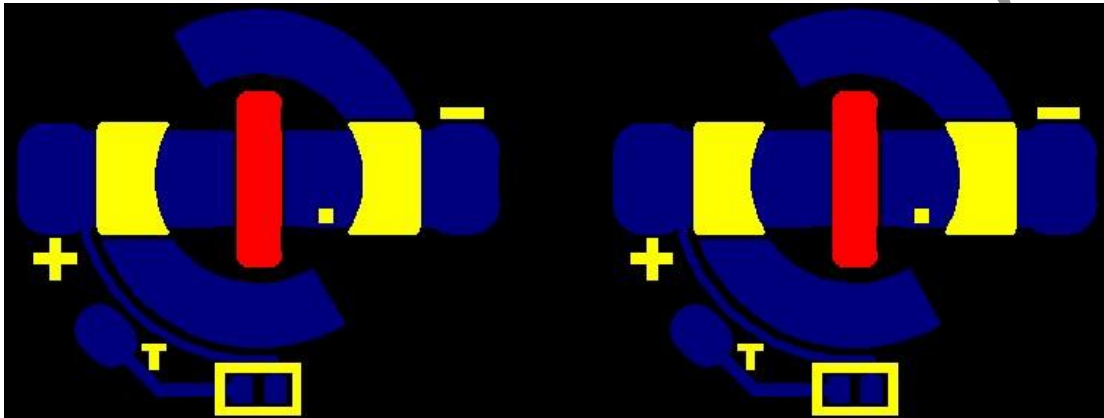
Original Gerber File (Top Layer + Top Silkscreen)



Revised Gerber file (Top Layer)



New Added Copper layer (convexity on copper core)



Final Revised Gerber File

BEST TECHNOLOGY

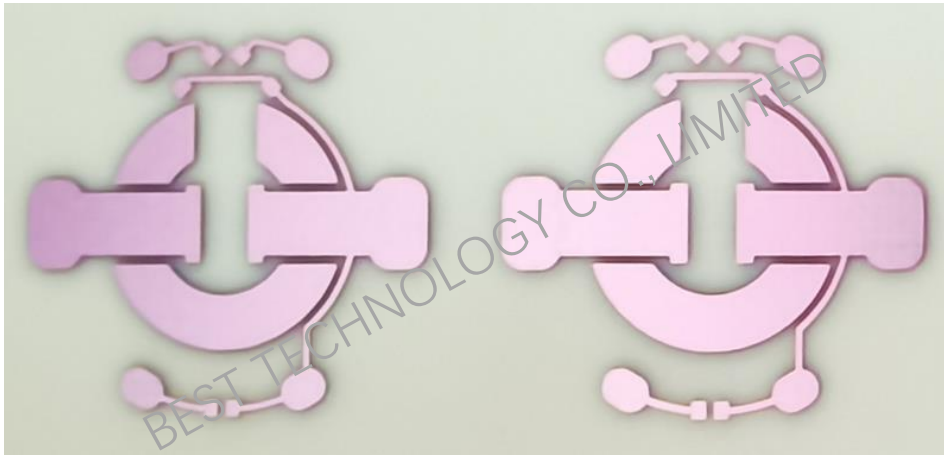


2) Produce both trace layer and copper core layer separately.

A. As of trace layer, it'll be manufactured according to normal [FR4 PCB](#), but the raw material will be BT (Bismaleimide-Triazine), instead of FR4. You can see more on [PCB manufacturing process](#).

Normally it's single copper trace layer on that trace layer. But we also have some customers like to design two copper trace layers. That's also no problem.

Here are pictures of finished trace layer. (one copper trace layer)



B. Regarding the copper core layer, that'll be a little complicated than normal.

b1. Normally the thickness of copper core/substrate is 0.8mm, 1.0mm, 1.2mm, 1.4mm, 1.6mm, 2.0mm. And there're protection film (blue color) on both sides. We will remove that protection film after raw material cutting, before convexity making, so that copper can be etched.





After that, we will grind the copper core to make sure it'll be clear enough, so that in next step another film can be added to make trace.



(copper core before grinding)

(copper core come out from grinding m/c)

b2. We will add one extra layer of film (normally blue color, and will refer to blue film hereafter) on whole area copper core, and after developing & etching, only selected area of blue film will be kept and other area will be removed away. In fact, the area of blue film will be the place heatsink PAD will be.

There're two types of film: dry film and wet film. Dry film will be stick on trace layer using a film laminator and that film has better dimension control while etching process, so it'll be recommended while the trace width and space in [SinkPAD board](#) ([SinkPAD PCB](#)) is smaller.



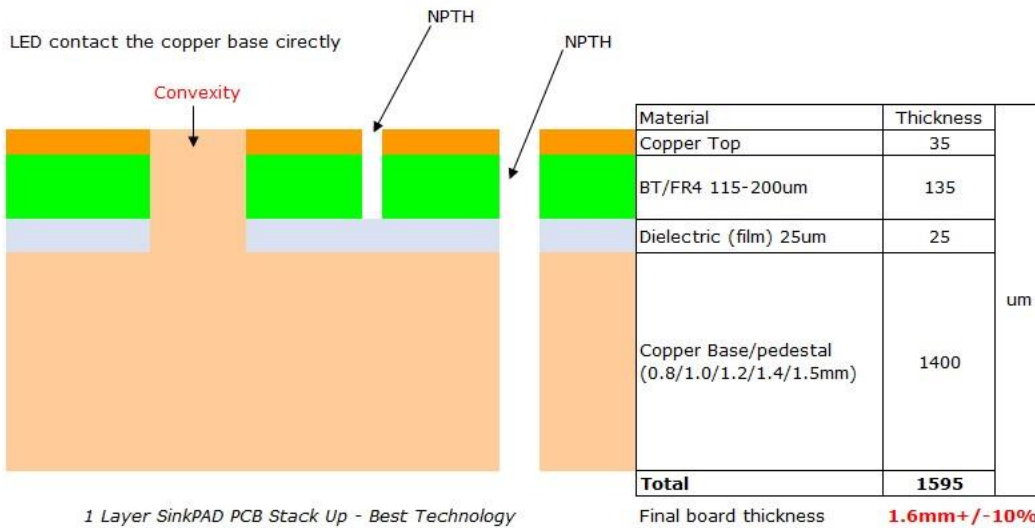
(dry film lamination process)

Wet film will be made by printing and baking process, similar like [soldermask making process](#). It'll be used for normal SinkPAD board (SinkPAD PCB) with normal trace width and space (>12mil).

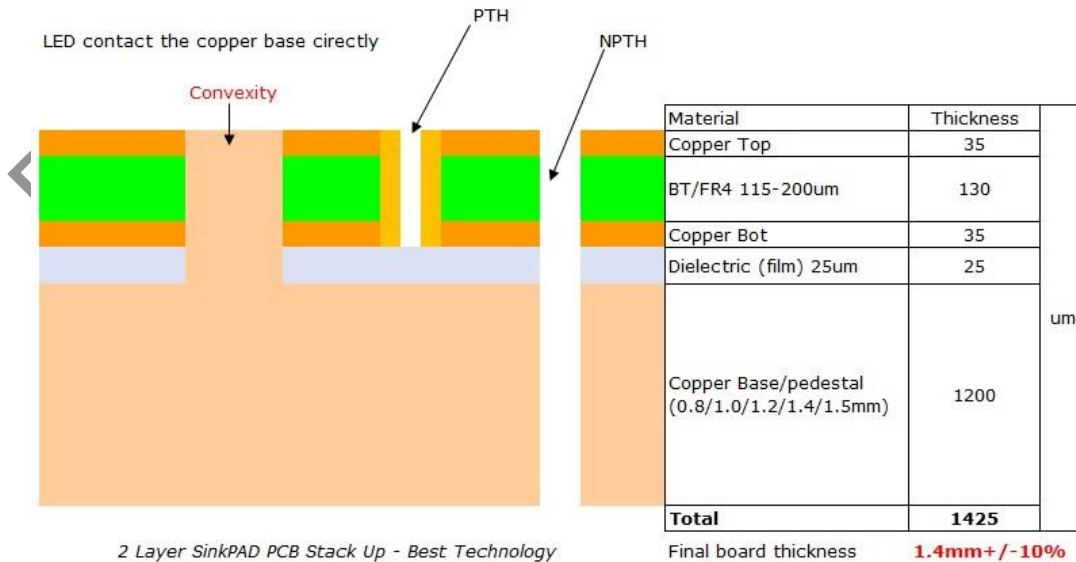


(copper printed with wet blue film) (printing wet blue film on copper core)

If that is a [single layer SinkPAD board](#), then the blue film will on the top side of copper core and there'll be a transparent film (refer to transparent film) on bottom side of copper, to protect bottom copper from etching.

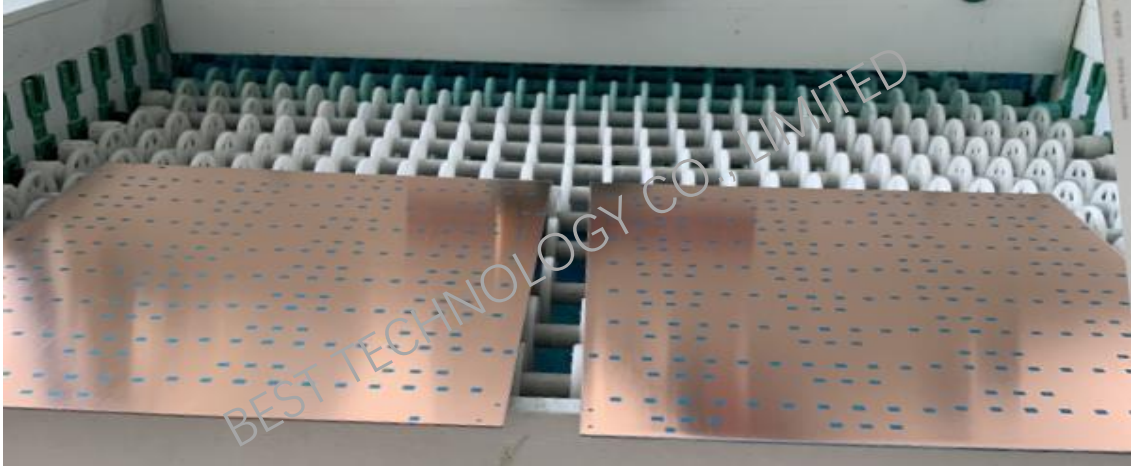


And of course, you can design two layers of trace circuit on the same side, then we will name it as [2L SinkPAD Board](#), and manufacturing process is the same as [2L FR4 PCB](#).

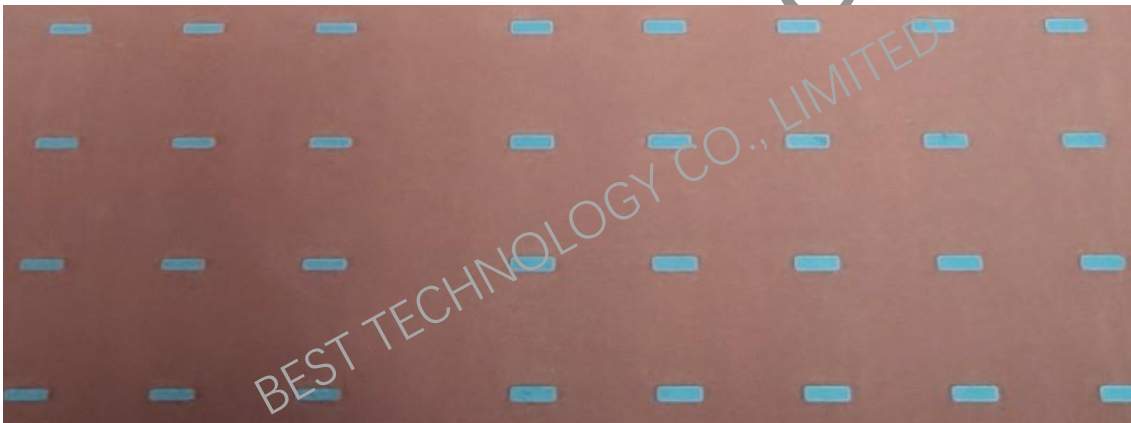




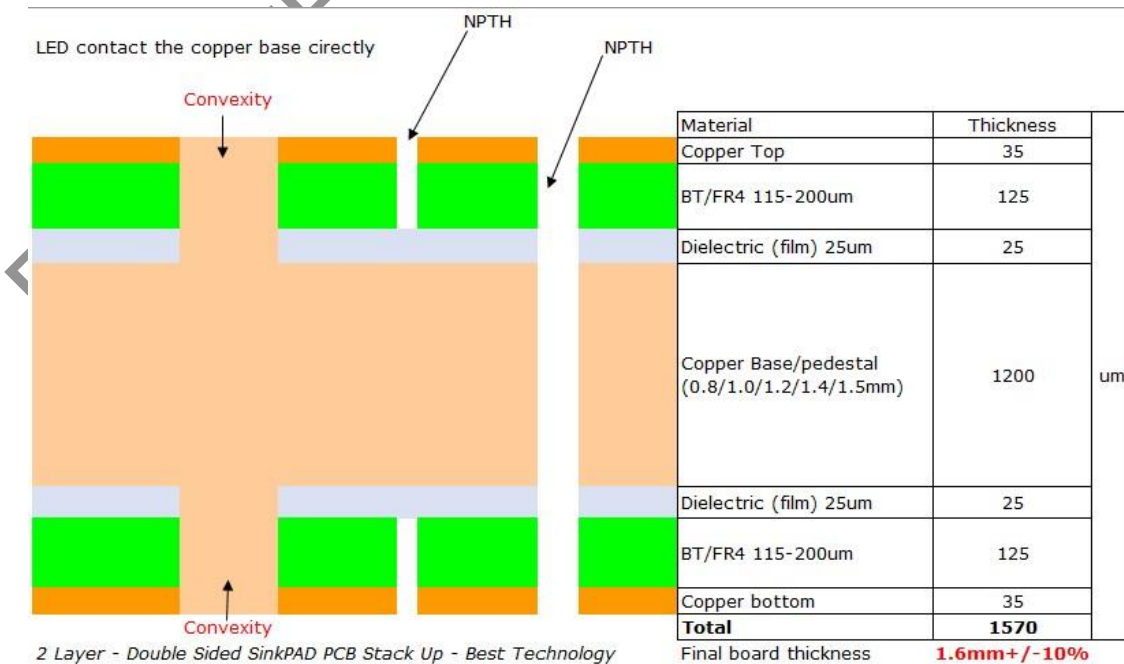
If there're SinkPAD on both sides, then we will put blue film on both top and bottom side of copper core.



(copper core with blue film on convexity, before etching)

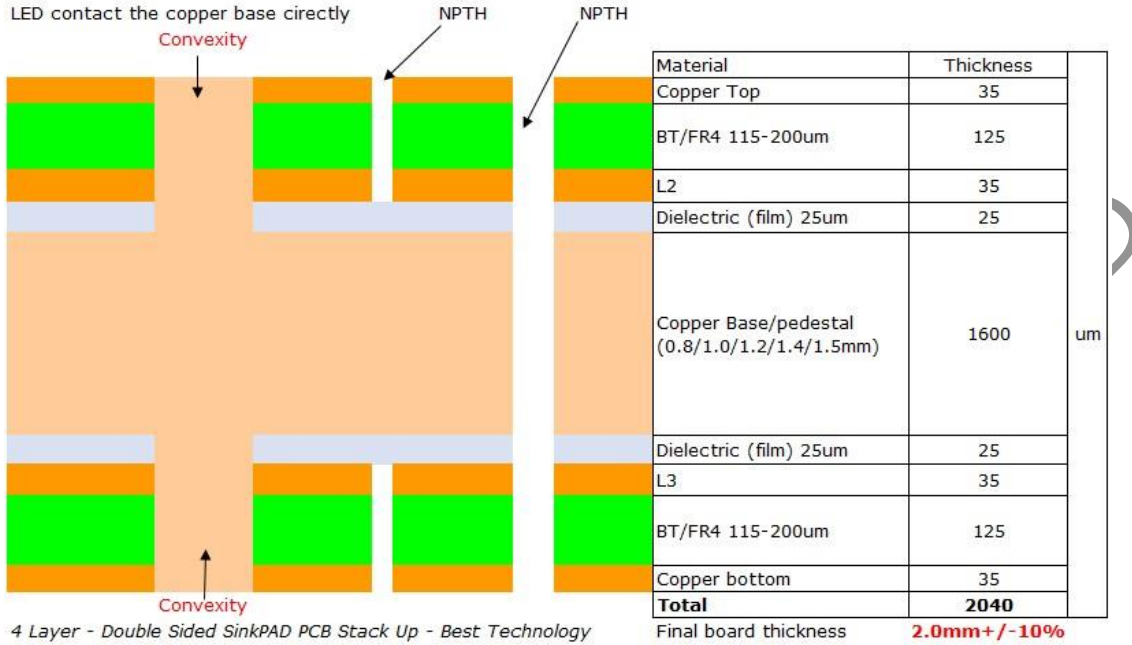


(copper core with selected area covered with blue film)

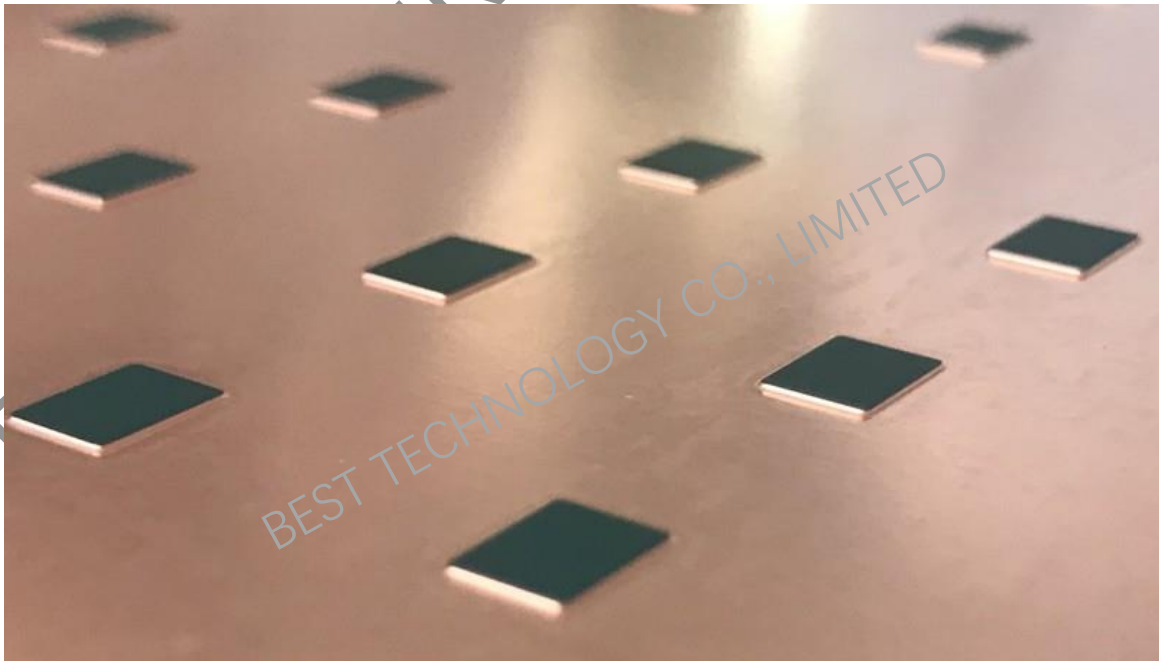




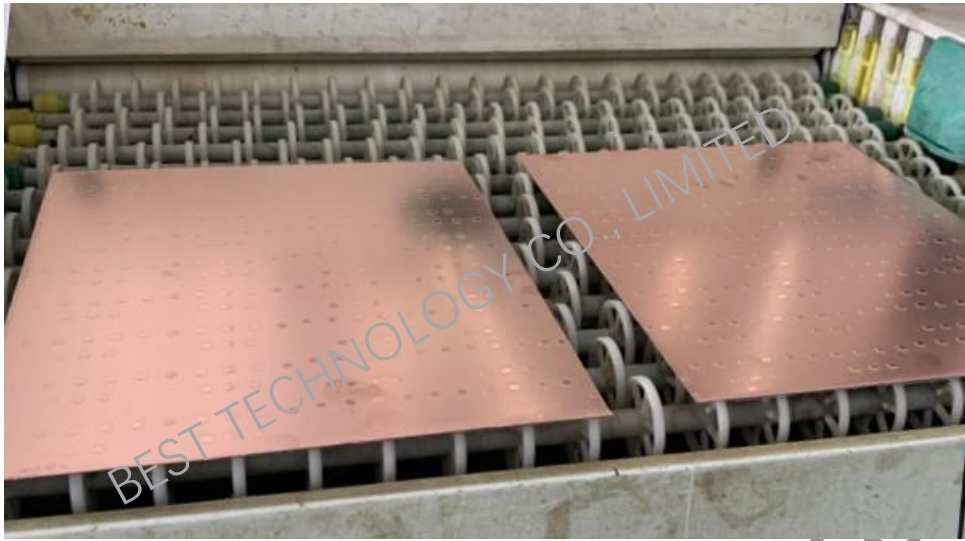
And of course, you can put two layers of trace circuit on both top and bottom sides, then to make 4 Layers double sided SinkPAD Board.



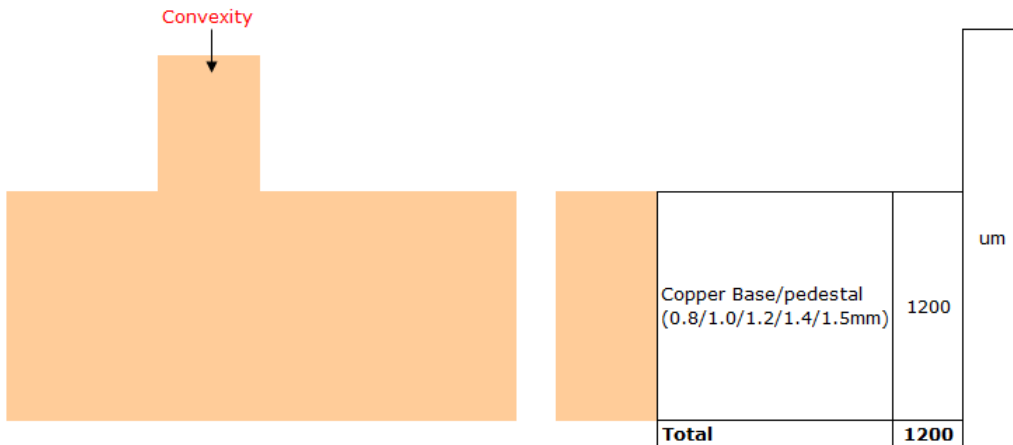
b3. Etching the copper layer, the copper in the area without the protection of blue film or transparent film will be etched away and depth will be 0.20mm-0.25mm. Only the copper in the area with blue film keep the unchanged and that area we normally named it as convexity.



(convexity coming out, still with blue film)



(convexity coming out, without blue film)



BEST

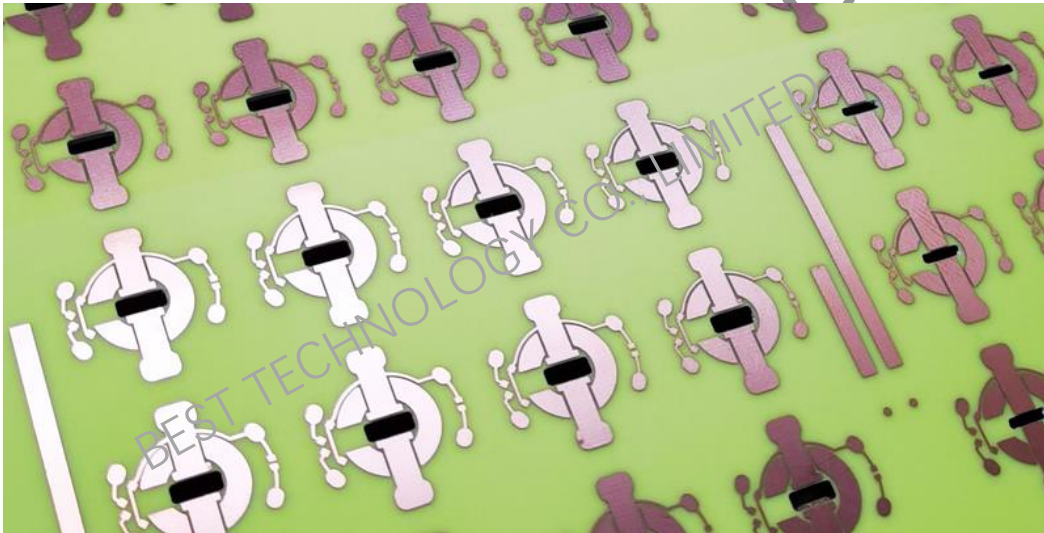


3) Laminate trace layer and copper core layer together.

- a. Add glue on bottom side of trace layer so that it be stick on copper core

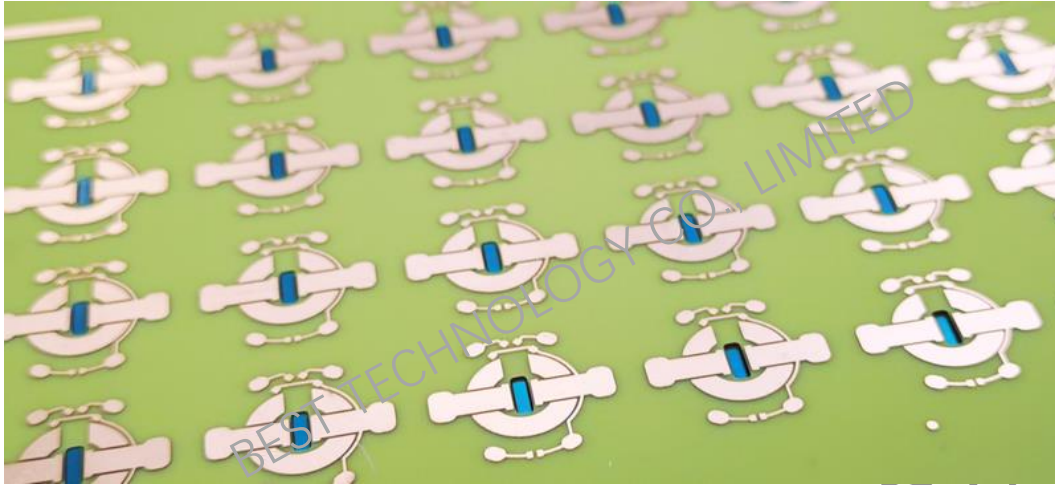


- b. CNC milling/routing the trace layer with glue, to remove the cutout where the convexity of copper core will be.



- c. Take off the white release paper, align and put the trace layer on copper core





(trace layer & convexity aligning process)



(trace layer & convexity aligning process)

c. Lamination process: separate each layer with high temperature paper and silicon gel, and put them into high temperature laminator with vacuum condition. After several hours with suitable parameters, the trace layer and copper core will be stick to each other firmly and become a new board, like normal [single layer MCPCB](#) or [double sided MCPCB](#).



(lamination process: laminate & align board before hot process lamination)

d. SinkPAD come out.

After these steps, you can see, the electronics connection will be on trace layer and then the direct thermal conducting PAD is coming from copper core. And the trace PAD on trace layer seemed in the sink area of copper core, so it named [SinkPAD board](#) ([SinkPAD PCB](#)).

Because the electronic related function was on trace layer and thermal conduction was on copper core, they are separated from each other. So that layer also named as Thermoelectric separation circuit board.

By this way, the thermal PAD of LED can be touched the copper PAD and core directly, so that heat can be conducting directly from LED to copper core, so it is also named as DTP (Direct Thermal Path). By this way, we can reach 400W/m.K thermal conductivity.



4) Continue the stripping, solder mask, silkscreen printing and then surface finishing process, like normal FR4 PCB, MCPCB

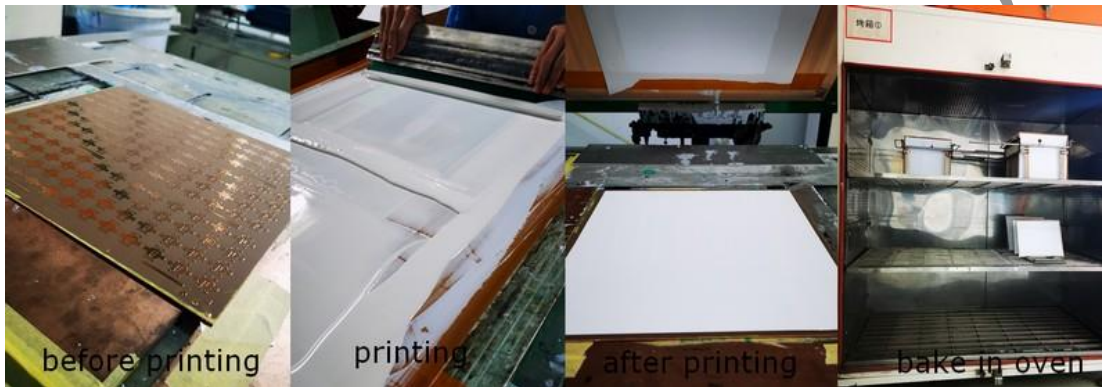
Similar like FR4 PCB, soldermask and silkscreen making is the same like other normal PCB. Here are some simple basic process showing soldermask making of [SinkPAD board \(SinkPAD PCB\)](#).

After hot lamination process SinkPAD board need to be cleaned again and then go to next step of solder mask making.

Firstly, using correct stencil and align SinkPAD PCB with stencil;

Secondly, print the oil (solder mask) (normal color is white and black, green, blue, yellow, red color available too), via handle (samples & small order) or auto-printer (big volume production), on surface of SinkPAD board;

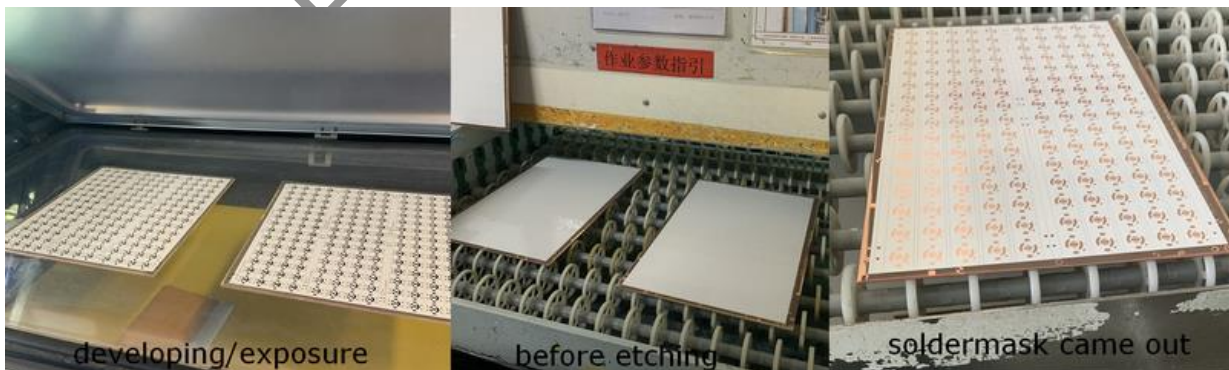
Thirdly, after 1 hours standing by, the boards will be sent to oven for baking;



(Soldermask printing & baking)

Fourthly, using soldermask film, to develop the soldermask using UV light or other type of lighting, to exposure the soldermask;

Fifthly: Etching the board and finally the soldermask will come out.



(soldermask developing & etching process)

Silkscreen is also be made by silk printing technology only and didn't need developing & etching process.

Normally the surface finishing of SinkPAD board are OSP, ENIG. And there's no HAL(LF) as the process is not good enough for Sink PAD PCB. Other special finishing is available upon request.



8949 2899

Email: sales@bestpcbs.com

9E, Jindacheng Building, Center Rd., Shajing Town, Bao'an District, Shenzhen, 518104, China

BEST TECHNOLOGY CO., LIMITED

Tel: +86-755-2909 1601/1602

Fax: +86-755-



[Http://www.bestpcbs.com](http://www.bestpcbs.com)

Each process will be monitored to make sure beautiful soldermask, silkscreen, surface finishing will be made according to customers' drawings. Please [contact Best Technology](#) if you want to know more about that series of manufacturing process.

After soldermask, silkscreen will be printed, before or after surface finishing which normally is ENIG, OSP.

BEST TECHNOLOGY CO., LTD



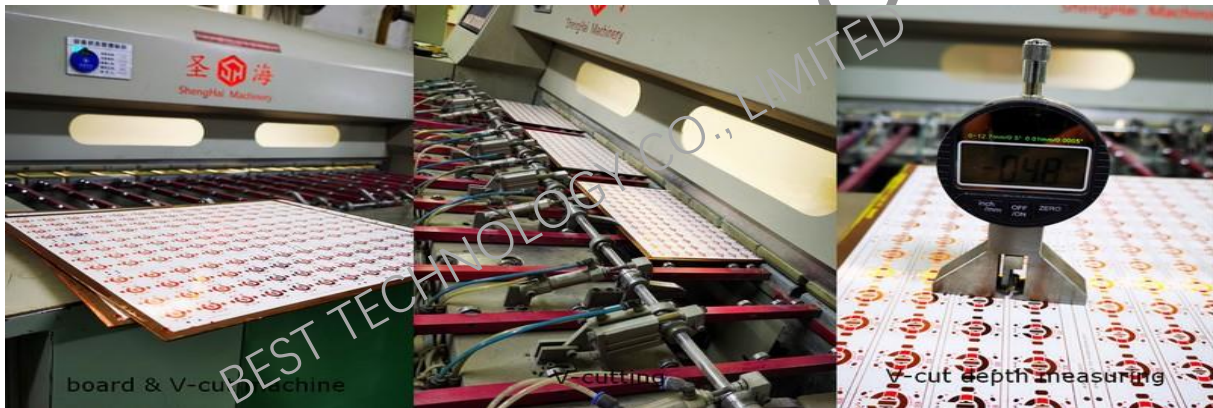
5) Outline/shape making, electric testing, FQC, shipment, etc.

As the raw material of SinkPAD board is copper in most of cases, so outline making is also the same like other type of [Metal Core PCB](#), dividing into three basic types:

- A. V-Cutting
- B. CNC routing/milling
- C. Die-punching

V-cutting will be used if the board is shipped via panel (several pieces into the same panel), and that process will be done before CNC routing/milling or Die-punching, as V-cutting process will not affect the location of each board, no alignment issue. If CNC routing firstly, the location of board will be shift and then V-cutting will be a big problem as it machine can't V-cut the board outline correctly.

After V-cutting, the depth of V-cut will be measured, to make sure the balance on the [SinkPAD board](#) ([SinkPAD PCB](#)) meets customers' requirements, to support the weight of board itself and at the same time, not difficult to be de-panel.



(V-Cutting & Measuring)

If the board outline is not square or rectangle, then outline have to be made via CNC routing or Die-punching. While design the panel layout, at least 2mm space between the edge of [SinkPAD board](#) ([SinkPAD PCB](#)) should be keep in order for routing or Die-punching.

Several layers of SinkPAD PCB will be routed together, in order to save time. But for punching, there're only one layer of [SinkPAD board](#) ([SinkPAD PCB](#)) each time. And Die-punching will be used only while there're big volume order as the cost of die-punching tooling is higher comparing with other ways.



(CNC Routing For SinkPAD Board (SinkPAD PCB))



If there're a lot of through holes, normal dimension tolerance, volume production order, or complex outline which is difficult for routing, Die-punching will be used to make [SinkPAD board](#) ([SinkPAD PCB](#)) too. There're pictures showing Die-punching of an aluminum core PCB, which is the same as SinkPAD board (PCB) punching. Each time, there'll be only one layer of board will be punched and will be punched through bottom to top, to avoid potential trace or soldermask peeling off.

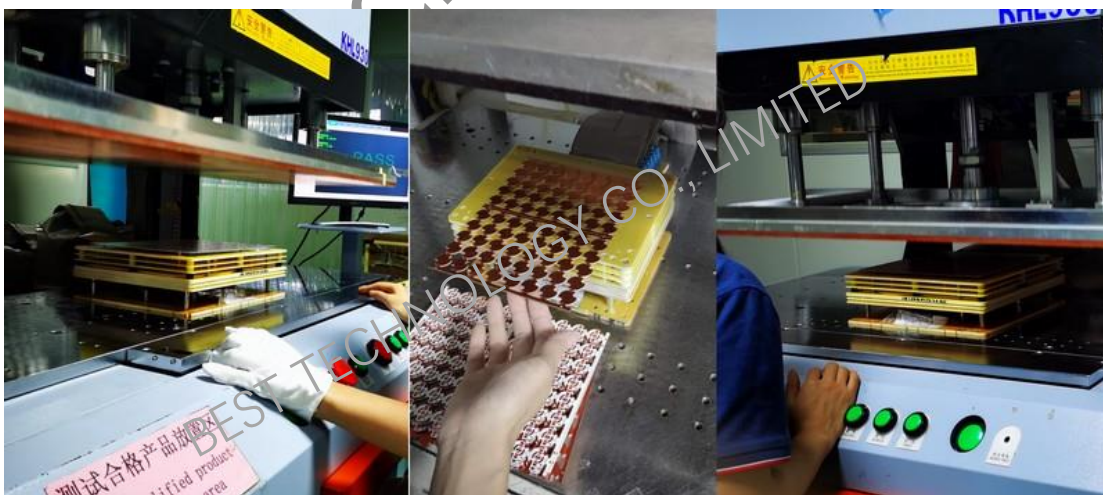


(Die-punching of aluminum core PCB)

Of course, these three processes can be used alone, or will be used together, upon how complicated the board is. So make simple outline/shape of SinkPAD board (PCB), or any type of [MCPCB](#) will be make manufacturing cost cheaper.

During these outline making processes, the burs of outline will be removed by human handle, to make the edge of circuit board outline is smooth enough.

After outline making, electronic testing will be happened, most of them are opening and shorting testing, and sometimes, high voltage, breakdown testing, to make sure each piece of SinkPAD board is in good condition.



(SinkPAD Board (PCB) Electronic Testing)



6) Finished the SinkPAD board making.

After these steps, the [SinkPAD board](#) ([SinkPAD PCB](#)) comes out. It can be panel (several pieces together), or single piece, upon customers' requests.



Top and bottom view of finished SinkPAD board- in panel)

(top



(top and bottom view of finished SinkPAD board- in single piece)

Now the all manufacturing process of SinkPAD board (PCB) were finished and the circuit boards can be vacuum packed now. Normally 10-20 pcs panels per bag, and desiccant and or humidity indicator will put together with boards, and cardboard will be put on both top and bottom of. And we will put white paper between each board, if surface finishing is ENIG, to avoid potential surface finishing rubbing.



To be noted that the thickness of trace layer with glue is 0.20mm and the depths of copper/thickness of convexity is also 0.20mm, but considering manufacturing process, there'll be a +/-0.05mm thickness tolerance comparing trace layer and convexity. And this is one of reason why you see the uneven of finished board.

Also the distance between edge of trace layer cutout and edge of convexity of copper is +/-0.10mm, so you can see the gap between in finished board, even it'll be covered by soldermask.

If you still have any question on the manufacturing process of our [SinkPAD board \(SinkPAD PCB\)](#), welcome to [contact us](#) any time! [Best Technology](#) always to be your most reliable partner of [SinkPAD board \(SinkPAD PCB\)](#) in China!