contents

Order steps and the meaning of the options

Please Choose:

Dimension	30.0cm x 40.0cm ▼
Thickness	Decide by Vendor ▼
Techniques	Polishing ▼
Mark point	No Fiducial ▼

***Upload the Gerber file must be packed into RAR orZIP format.**

File	File .

First, Choose the right Dimension of the stencil.

Dimension	20cm X 20cm no frame ▼
Thickness	20cm X 20cm no frame
	20cm X 25cm no frame (+100g)
Techniques	20cm X 30cm no frame (+200g)
	28cm X 38cm no frame (+$2.00) (+651g)
Mark point	Customized size no frame (+$2.00) (+651g)
	30.0cm x 40.0cm (+$2.00) (+651g)
*Upload the	37.0cm×47.0cm (+$4.00) (+1151g)
	42.0cm×52.0cm (+$12.00) (+1501g)
	45.0cm×55.0cm (+$24.00) (+1651g)
	58.4cm×58.4cm (+$39.00) (+2851g)
File	55.0cm×65.0cm (+$39.00) (+3051g)

Limitation on PCB size:

For stencil, not all the area of stencil can be used in soldering, that is, the effective area is smaller than the stencil dimension.

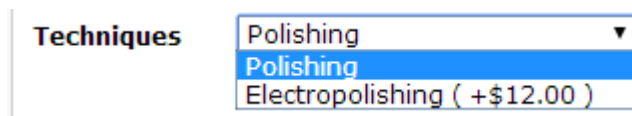| Frame Dimension	Effective Dimension
20cm X 20cm no frame	18.0cm x 18.0cm
20cm X 25cm no frame	18.0cm x 23.0cm
20cm X 30cm no frame	18.0cm x 28.0cm
28cm X 38cm no frame	19.0cm x 29.0cm
Customised size no frame(less than 28*38cm)	19.0cm x 29.0cm

30.0cm x 40.0cm	13.0cm x 21.0cm
37.0cm x 47.0cm	19.0cm x 29.0cm
42.0cm x 52.0cm	24.0cm x 34.0cm
45.0cm x 55.0cm	27.0cm x 37.0cm
58.4cm x 58.4cm	40.0cm x 40.0cm
55.0cm x 65.0cm	37.0cm x 47.0cmm

For PCBs with SMT parts both side, The effective area should double larger than the dimension of the PCB.

Second, For the techniques selection:
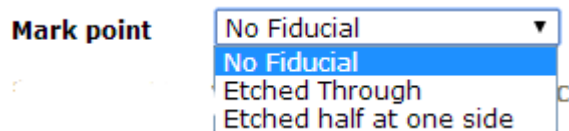
Polishing is manual grinding polishing.

Electropolishing is electrolytic etching in special liquid to make the stencil more clean on the surface and clear the inner needling of the pads holes. It main used for small pins of IC, BGA or 0201 parts.

Third, Mark point

the Etched through is main used to for machine to recognize the position or to hold position of the stencil without Frame.

Etched half at one side is used for Machine to recognize the position.

For Etched Through and Etched half one side, you need to mark it on the gerber file. Or the stencil engineer will ignore this option.

For Stencil ordered at the PCB page, the default stencil selection is No Fiducial, Polishing and Stencil Engineer decide the stencil thickness.

For customized size stencil without frame, if the effective area is less than 19cm*29cm and the stencil dimension is less than 28cmX38cm, you can choose the customized size option and design the stencil outline in the gerber file.

Stencil specifications

1. Elecrow will fabricate the stencil according to the Paste layer except the following situation.
    a. Obvious Gold finger   b. Obvious RF antenna.

2. If there isn't Paste layer, but there is solder mask layer, copper layer (top and bottom layer) and drill layer. We will fabricate the stencil according to the solder mask layer and the size pads of the PCB. All the solder mask layer will be opened except the following situations:

a. The solder mask for through hole will not be opened.

b. If the isn't copper under the solder mask, it won't be opened.

c. If you need to enhance the thickness of the trace with solder cream, please leave a comment and mark it out.

d. Gold finger, RF antenna won't be opened.

e. TP test point won't be opened

3. If there is only one layer there, we will fabricate the stencil without any modify.

4. If you choose the mark point as Etched Through or Etched half on one side and there is no mark point on the gerber file, we will ignore these selections.

5. About mirroring the bottom side of the PCB on the stencil.

If all the character is on the right side, we won't mirror the bottom layer.

If there isn't silk screen layer, we will mirror the GBP(BP) layer and don't mirror GTP(TP) layer.

Note, If there are some holes opened more , please cover it via scotch tape, you do not need to make another stencil again.

If there is any other needed, please feel free to contact with techsupport@elecrow.com