

## Problem/Procedure Description

- How to use the ProMask and Prolegend

## Requirements

- Finished PCB board (still in material)
- Promask kit
- Hot Air oven
- Circuit CAM 5.2 or 6.0
- UV exposure table

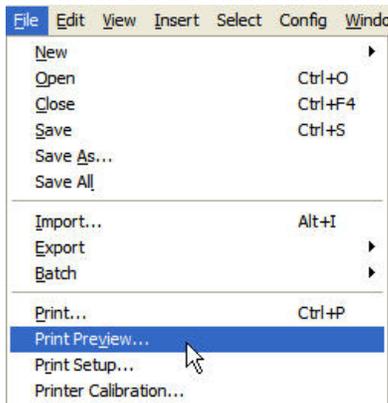
## Problem/Procedure Solution

Printing the solder mask layers on transparency sheet.

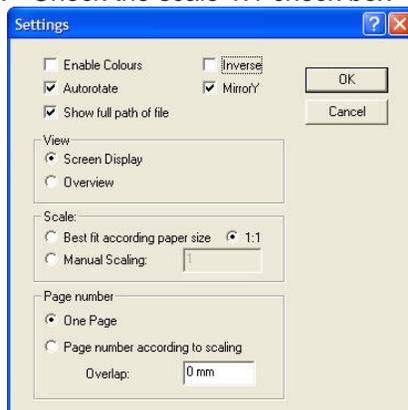
1. Calibrate printer

*Note: only do this step if this is the first time using the promask/prolegend system, print from a different computer/printer. The printer calibration settings are saved within Circuit CAM.*

- a. Open Circuit CAM by double clicking on the circuit cam icon.
- b. Go to the file menu and select print preview.

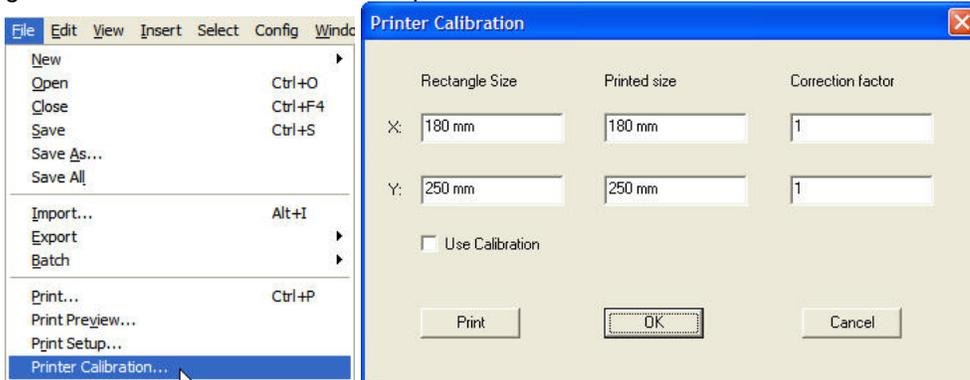


- c. Press the properties  button
- d. Verify the following information
  - i. Uncheck the colors check box
  - ii. Check the scale 1:1 check box



- e. Press OK
- f. Press the close button

g. Go to the file menu and select printer calibration.



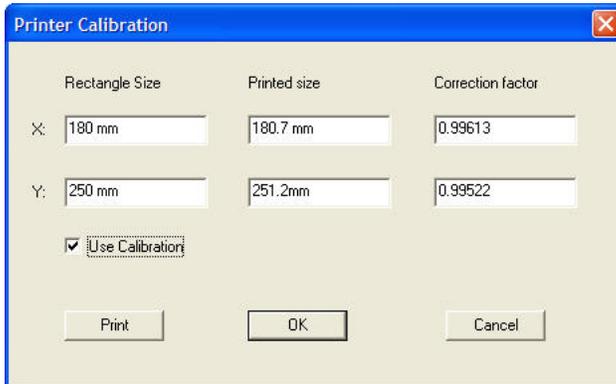
h. Click on print.

i. This will print a 180mm x 250mm rectangle.

i. Measure the length of the rectangle.

j. If the measurements are not exactly 180mm x 250mm, place the actual measurements into the printed size column. Circuit CAM will determine the scale factor.

k. Check the use calibration factor



l. Re-print the rectangle.

m. Measure the rectangle as you did in step 1i.

n. Place film (transparency) into tray 1

o. Print from tray 1

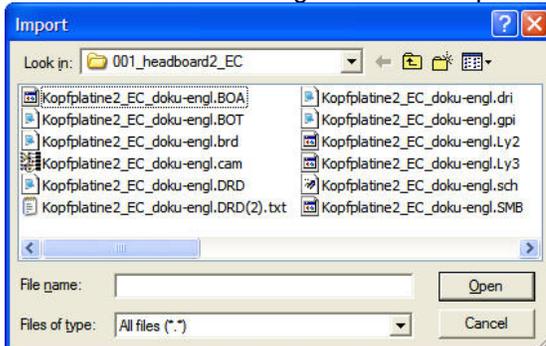
p. Measure the rectangle on the film as stated above.

2. Importing the solder mask/silk screen layers.



a. Click on the import icon

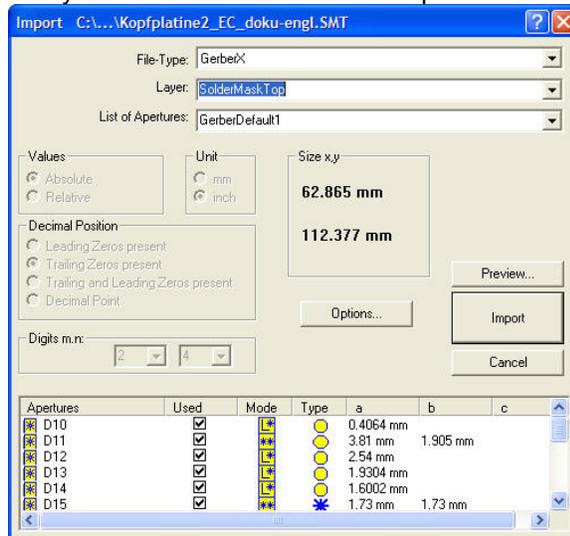
b. Click on the desired gerber file to import



c. Click on open.

d. On the import screen, verify the following information

- i. File type: Gerber X
- ii. List of Apertures: Gerber Default
- iii. Layer: Solder Mask Bottom or Top



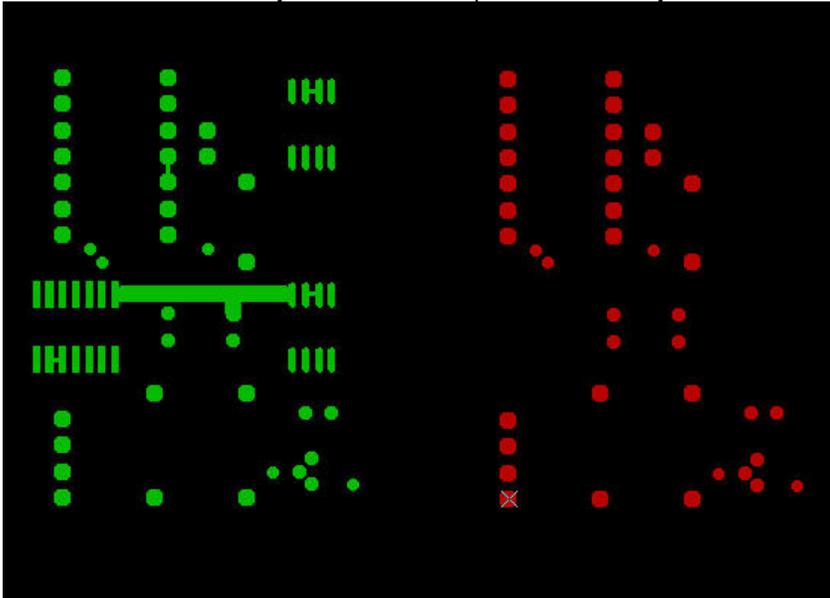
*Note: If printing silkscreen layers, place them onto the silkscreen top or bottom layers*

- e. Click on the import button.
3. Separate top and bottom layers
  - a. Select a pad on the solder mask bottom layer.
  - b. Go to the select menu and choose layer.

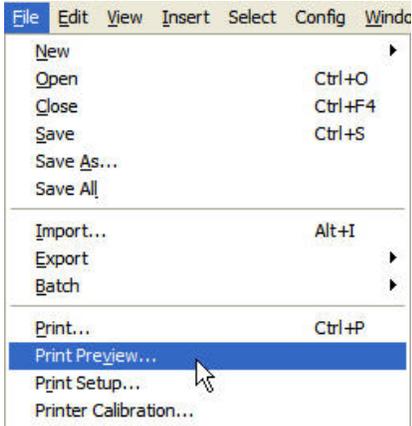


- c. Press the space bar to enable the move function.

d. Move the bottom layer so that the top and bottom layer are stacked side by side.

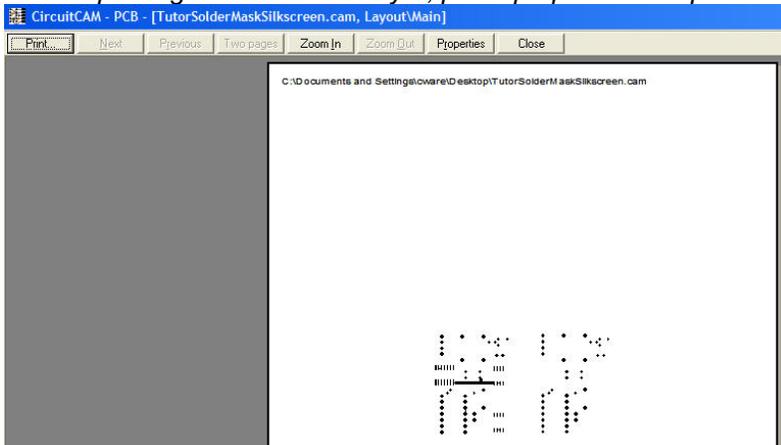


4. Go to the file menu and select print preview



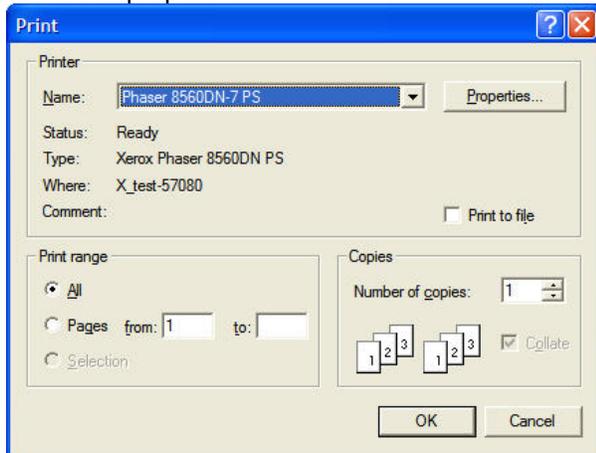
5. Verify that the solder mask is going to print on the same page.

*Note: If printing the silk screen layer, press properties and place a check in the inverse check box and press ok.*



6. Press print.

7. Go to properties

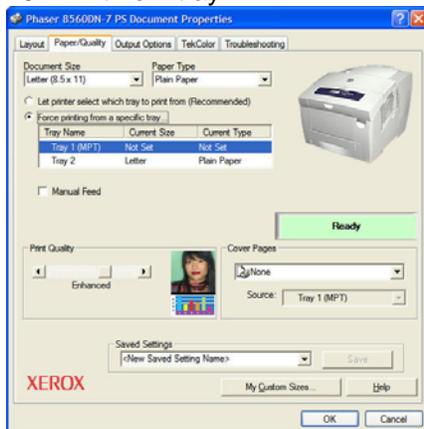


8. Load transparency sheet into printer tray 1

**Caution: Use the manual loader tray as the transparency will not travel through the rollers and can cause a paper jam in the printer.**

9. Select the paper/quality tab

10. Print from tray 1



11. Repeat for Pro Legend (Silkscreen) Layers

### Applying Promask/ProLegend to the PCB board

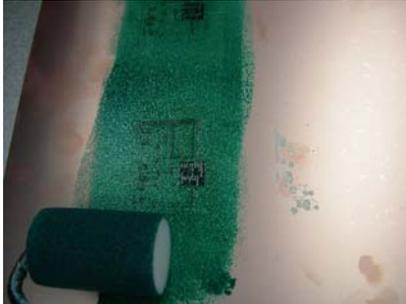
*Note: Promask and ProLegend must be completed in two separate sequences.*

1. Mix promask/prolegend parts A and B into paint dish.

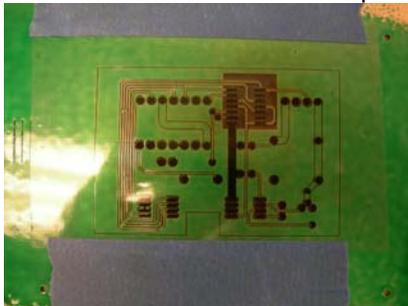


*Note: Recommend placing a plastic bag around the paint dish to speed clean up. The plastic bag can be placed in the trash when finished.*

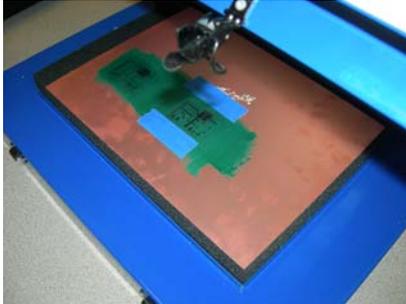
2. With supplied roller, apply a thin layer of promask/prolegend onto both sides of the PCB board.



3. Place PCB board into the hot air oven and cure at 80 degrees Celsius (160 degrees fahrenheit) for 10 minutes.
4. Let PCB cool down for approximately 5 minutes.
5. Trim printed transparency sheet.
  - a. Match up the film to the pads of the PCB board.
  - b. Secure the film with tape.



6. Place the PCB board on the UV exposure table.



- a. Expose each side of the PCB board for 60 seconds.
- b. ProLegend UV exposure time for each side is 120 seconds.



7. Remove the transparency sheets

- a. Keep film for additional boards or discard in the trash.

8. Mix the developer with 1000mL of water. (40 to 50 degrees Celsius)

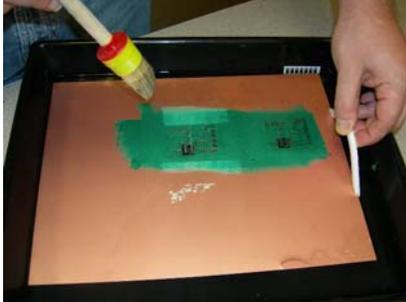
Note: The water for the Pro Legend must be less than 25 degrees Celsius, so you are able to use the same water from the promask sequence.



9. Pour water into developer dish.



10. Attach supplied plastic handles to the PCB board and place the PCB board into the developer.



11. Brush away the solder mask or silkscreen from the PCB board.
  - a. Repeat for both sides.
12. Rinse the board to stop the developing process.
13. Place the PCB board into the hot air oven and bake at 160 degrees Celsius (325 degrees fahrenheit) for 30 minutes to cure the promask/prolegend.
14. Let PCB cool down.
15. If necessary, repeat steps 1 through 15 for the ProLegend.
16. Mix conditioner into the finished developer dish.
  - a. This neutralizes the developer.



17. Pour water down sink drain.
18. Throw away roller pad and any opened packages.
19. Clean the paint dish with alcohol.
  - a. If you used a plastic bag on the paint dish, throw it away in the trash.

## **References**

- ProMask/ProLegend Operations Manual
- LPKF Technical Support