ProMask/ProLegend

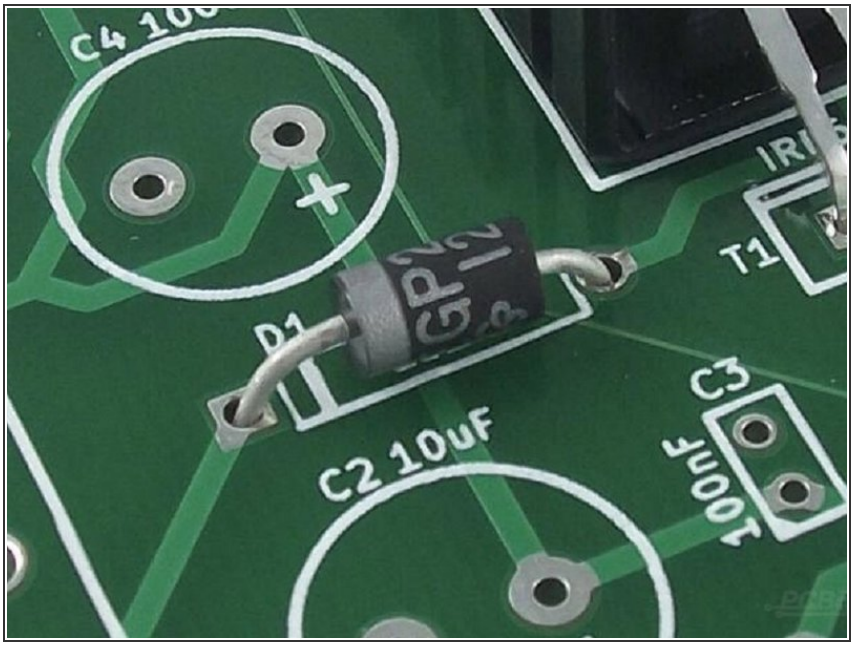
The ProMask and ProLegend kit is used for adding silkscreen graphics and the top insulating green layer on the PCB

Written By: Jimmy Nolan

Introduction

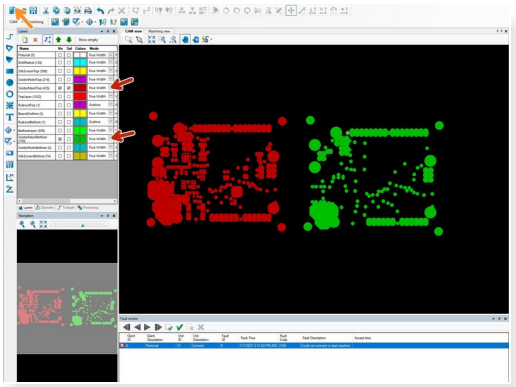
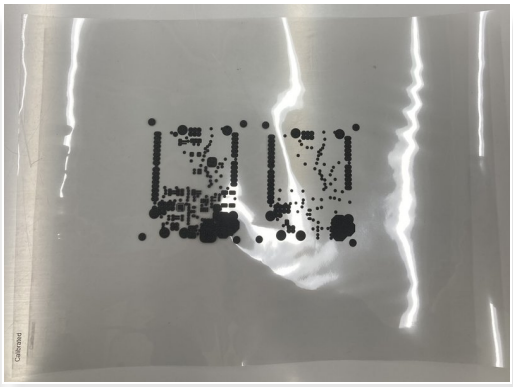
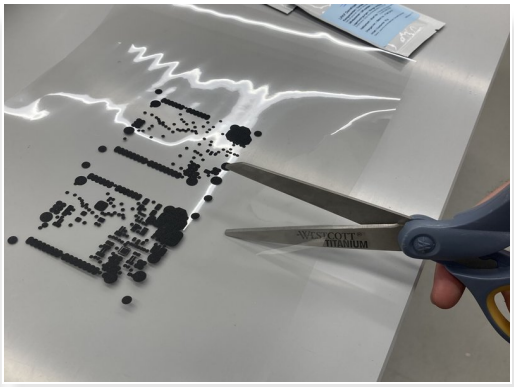
This guide will teach you how to use the ProMask and ProLegend kit to add the green solder mask layer silkscreen graphics to your PCB.

Step 1 — Solder mask and silkscreen intro



- Solder mask is what makes circuit boards green and is used to protect the board and make soldering easier.
- Silkscreen is used to embed white text on the surface of the circuit board.
- Both of these layers must be applied to the surface of the PCB and selectively cured using a printed transparency sheet under a UV lamp.
- Solder mask and silkscreen are added after the vias are drilled/plated and the top/bottom traces on the PCB are milled.

Step 2 — Print the transparency sheets

- Open your project in CircuitPro and in the CAM view make only the top and bottom solder mask layers of your design visible and drag them apart.
- Select *File->Print* and make sure the paper comes from *Manual*.

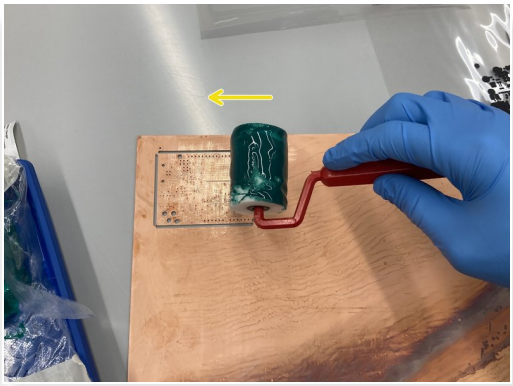
  ⓘ You can choose *Print preview* to make sure the print will turn out correctly.
- Click *Print* and insert a transparency sheet into the front of the printer located next to the computer.
- Repeat the procedure for the silkscreen layers but check *Invert* before printing.

Step 3 — The kits

- The solder mask/silkscreen kits are kept in a brown LPKF box in one of the tall white cabinets.
- One part contains a roller, a paintbrush, handles for holding the board, and popsicle sticks
- The other contains tips for the roller, gloves, a beaker, and a bottle

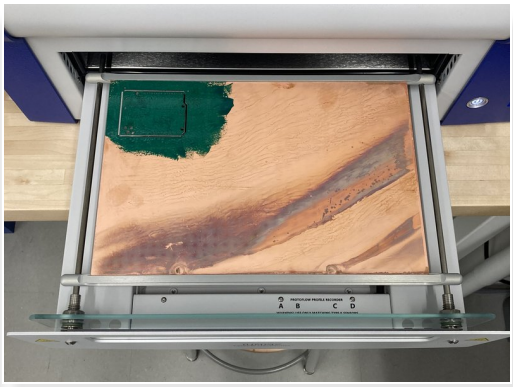
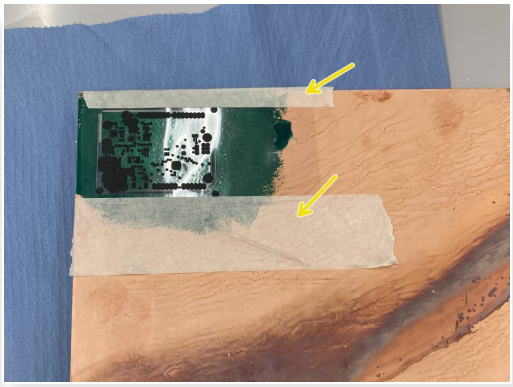
Step 4 — Apply the solder mask/silkscreen





- Retrieve one packet each of ProMask components A/B, ProLegend components A/B, LPKF Developer, and LPKF Conditioner.
  - ⓘ They are located in the fridge outside the AFL
- Place a plastic glove over the roller bin and mix ProMask compounds A and B with a popsicle stick (ProLegend A and B if you're doing silkscreen).
  - ⓘ It may take several minutes until they are fully mixed
- Use the roller to apply the solution on both the front and the back of the PCB.

Step 5 — Cure the solder mask/silkscreen in the oven

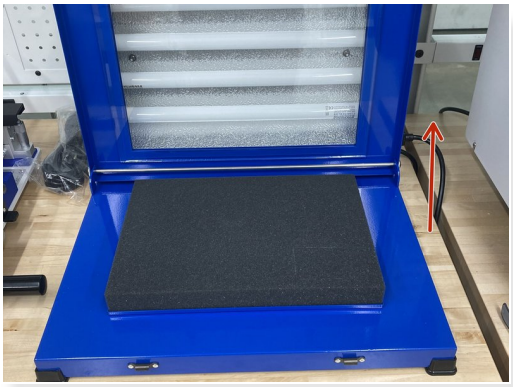




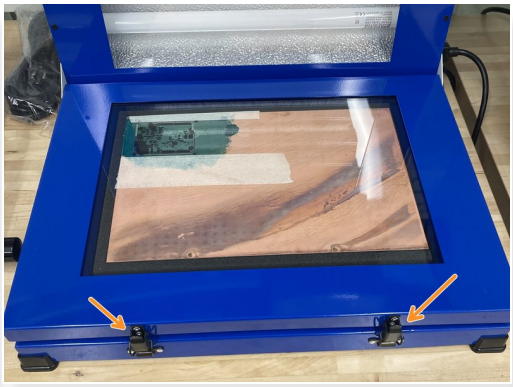
- Once a thick layer of solder mask/silkscreen is present on both sides of the PCB it's time to cure it.
- Place the PCB in the reflow oven and choose the *ProMask-PD* profile

  ⓘ Follow the guide for the reflow oven here.
- Once it is finished and cooled, line up and tape the appropriate cut out solder mask /silkscreen transparency sheet

  ⚠ Be sure not to mix up the top and bottom layer transparencies

Step 6 — Cure the solder mask/silkscreen in the UV lamp

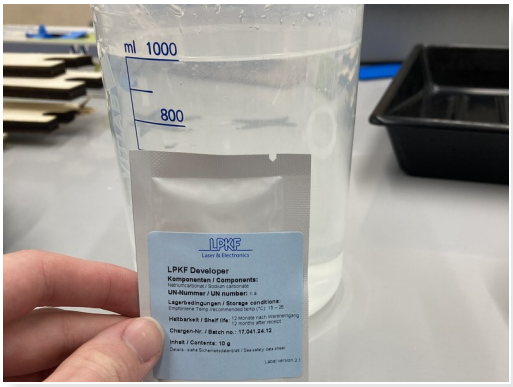


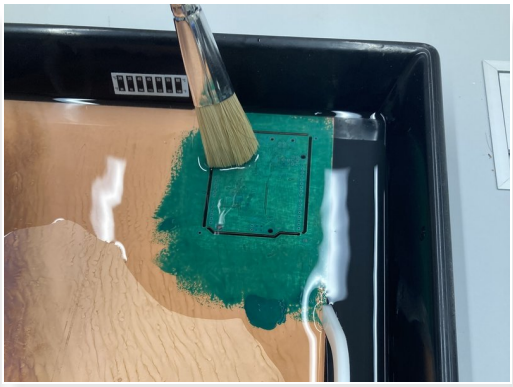


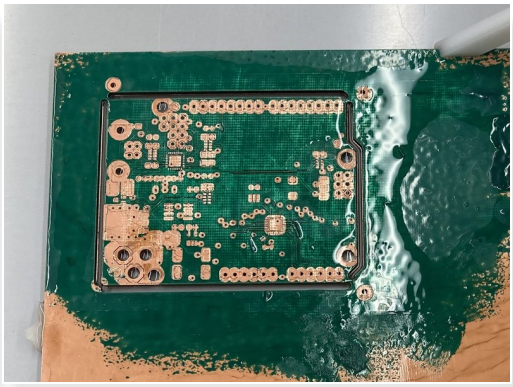
- Open the UV lamp
- Place the PCB under the glass and latch it shut
- Close the lid, set exposure to 60 seconds for solder mask and 120 seconds for silkscreen, and push start.

  Remember to cure both sides.

Step 7 — Remove the uncured solder mask/silkscreen







- Add the developer power to 1000mL of water at 40-50°C
  - The water does not need to be this warm for the silkscreen so you can reuse the same developer solution for both processes.
- Pour the solution into the bin provided with the kit and scrub off the uncured solder mask/silkscreen with the paintbrush

Step 8 — Postcure in the reflow oven

- Rinse the board off with water
- Place the board in the reflow oven and choose *ProMask-PC*
  - Follow the guide for the reflow oven here.
- Repeat the procedure starting from step 3 for the silkscreen if necessary.

Step 9 — Cleaning up

- Neutralize the developer solution with LPKF Conditioner before pouring it down the drain.
- Throw away the used plastic gloves, popsicle sticks, rollers, and transparency sheets.
- Pack up the kits and store the box away in the cabinet.