LIBRARY · REFERENCE GUIDE

SILKSCREEN, SOLDERMASK, AND POLARITY MARKS

Silkscreen and soldermask are the board's labels and protective coat. Good silkscreen (reference designators, pin-1 dots, polarity marks) makes a board buildable and debuggable.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

Silkscreen and soldermask are the board's labels and its protective coat. Silkscreen prints reference designators, pin-1 dots, and polarity marks that make a board buildable and debuggable. Soldermask is the colored layer that covers the copper and keeps solder only where you want it.

WHAT SOLDERMASK DOES

Soldermask is the thin lacquer, usually green, that coats the copper everywhere except the pads. It stops solder from bridging between close pins, protects the copper from oxidation, and insulates traces you might otherwise touch. The openings in the mask define where solder can wet, so on a fine-pitch part the mask openings matter as much as the pads.

THE SILKSCREEN LEGEND

Silkscreen is the printed text and symbols on top of the mask. A good legend gives every part its reference designator (`R1`, `C3`, `U2`) so you can find it, marks pin 1 on every chip, and shows polarity on the parts that have it: the band on a diode, the stripe or plus on an electrolytic capacitor, the marked cathode on an LED, and the pin-1 corner on a connector.

KEEP SILK OFF THE PADS

Silkscreen ink over a pad stops solder wetting and ruins the joint, so the fab and the design rules push silk clear of copper openings. When a reference designator will not fit beside its part without touching a pad, move the text; do not shrink it onto the pad. Legible silk that sits in the free space is the goal.

THE SILKSCREEN LEGEND: REFERENCE DESIGNATORS, A PIN-1 DOT, AND POLARITY MARKS THAT MAKE THE BOARD BUILDABLE.

A TITLE BLOCK EARNS ITS SPACE

Print a small title block somewhere on the copper or silk: the board name, a version, and a date. When you are holding three revisions of the same board, the one thing that tells them apart is the version you printed on them. It costs nothing and saves an afternoon of confusion at the bench.

DEEP DIVE · MASK-DEFINED VERSUS COPPER-DEFINED PADS

Most pads are copper-defined: the copper is smaller than the mask opening, so the copper edge sets the pad shape. On some fine-pitch or BGA parts the fab makes a mask-defined pad instead, where the mask opening is smaller than the copper and the mask edge defines the solderable area. It gives tighter, more repeatable pads on dense parts, at the cost of a little solderable area. For most boards you will build, copper-defined is the default and you never think about it.

CHECKPOINT

1. What does soldermask do?

a. Covers the copper and keeps solder off everything but the pads

b. Carries the board's main current

c. Labels the parts by name

ANSWER · A

Mask stops bridges, resists oxidation, and its openings define where solder wets.

2. What does a pin-1 marker on the silkscreen prevent?

a. The board drawing too much current

b. Solder bridges between pins

c. Installing a chip rotated the wrong way

ANSWER · C

Pin-1 fixes orientation so a part is not assembled backwards.

3. Why keep silkscreen off the pads?

a. It looks cleaner on the render

b. Ink over a pad stops solder wetting and ruins the joint

c. It saves silkscreen ink

ANSWER · B

Silk on a pad blocks the solder joint; move the text into free space instead.

- Prerequisite: reading a schematic (reference designators)
- Next: gerbers and the fab package