

FOOTPRINTS AND LAND PATTERNS

What a footprint is, how package maps to footprint, what pads and courtyards and pin-1 markers do, and why you confirm every land pattern against the datasheet.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

A footprint is the pattern of copper pads, holes, and silkscreen that one real part solders to. Choose the footprint that matches the part's package, confirm it against the datasheet's recommended land pattern, and the part fits and connects. Get it wrong and the part physically will not sit down, or it will not make contact.

PACKAGE VERSUS FOOTPRINT

The package is the part's physical body: an **0402** or **0603** chip, a **SOT-23-5** transistor outline, a **QFN** with pads under its belly. The footprint is the board-side copper that body lands on. One package maps to one footprint family, so reading **SOT-23-5** off a datasheet tells you which footprint to place.

WHAT A FOOTPRINT CONTAINS

A footprint carries pads (the copper each pin solders to), a courtyard (a keep-out rectangle that reserves the part's space so neighbours cannot crowd it), a silkscreen outline, and a pin-1 marker. The pin-1 marker is the one that saves boards: it fixes the part's rotation so a chip cannot be assembled turned around.

SURFACE-MOUNT AND THROUGH-HOLE

Surface-mount parts solder to flat pads on one side of the board and dominate a modern layout. Through-hole parts have leads that pass through drilled, plated holes and solder on the far side, which is mechanically stronger for connectors and anything that takes stress. A footprint is drawn for one or the other, and they are not interchangeable.

CONFIRM IT AGAINST THE DATASHEET

Every part datasheet gives the package dimensions a footprint has to match, and many add a recommended land pattern too: the manufacturer's own drawing of the pad sizes and spacing that part wants. The AP2112K regulator's datasheet, for instance, specifies its **SOT-23-5** package and pinout (Diodes AP2112 datasheet). Industry land-pattern geometry follows the IPC-7351 standard, and the datasheet is the first place you check for a specific part.

- [Diodes Incorporated. AP2112 600mA CMOS LDO Regulator datasheet \(SOT-23-5 package and pinout\).](#) [diodes.com](#)

A FOOTPRINT: PADS TO SOLDER TO, A COURTYARD THAT RESERVES SPACE, AND A PIN-1 MARKER THAT FIXES ROTATION.

DEEP DIVE · IPC-7351 DENSITY LEVELS

The IPC-7351 land-pattern standard does not give one footprint per package; it gives three, called density levels. The Most level leaves the largest pads and solder fillets, which suits hand assembly and rework. The Least level is the tightest, for high-density boards assembled by machine. The Nominal level sits between and is the sensible default for most work. Footprint generators that follow IPC-7351 let you pick the level, so you trade board area against how forgiving the part is to solder.

CHECKPOINT**1. Where do you confirm a part's correct land pattern?**

- a. By measuring a similar part
- b. From the schematic symbol alone
- c. In its datasheet's recommended land pattern**

ANSWER · C

A datasheet gives the package dimensions and, for many parts, a recommended pad geometry; check it there first.

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

- a. The part being assembled rotated the wrong way**
- b. The part overheating in use
- c. The pads oxidizing before assembly

ANSWER · A

Pin-1 fixes orientation so a chip cannot be placed turned around.

3. The courtyard on a footprint is there to do what?

- a. Carry the part's main current
- b. Reserve the part's space so neighbours cannot crowd it**
- c. Show the soldermask color

ANSWER · B

The courtyard is a keep-out rectangle that guards the part's footprint area.

- Prerequisite: reading a datasheet
- See it on a real board: the L1.01 build
- Next: component placement