

# DFM AND ORDERING A BOARD

*Design for manufacturing means a board the factory can build cheaply and reliably. Respect the fab minimums, read the cost drivers (layers, size, quantity, finish), and order.*

---

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

Design for manufacturing means drawing a board the factory can actually build, cheaply and reliably. Respect the fab's minimums, pick sane options, and read the cost drivers before you order. A manufacturable board is boring in all the right ways.

## THE MANUFACTURABILITY BASICS

DFM is mostly the DRC numbers plus a few board-level ones: minimum trace and space, minimum hole and annular ring, and a sensible board size and layer count. Stay comfortably inside the fab's limits rather than right at the edge, because a design that hugs the minimums yields worse and costs more to build.

## WHAT DRIVES THE COST

A few choices move the price far more than the rest: the layer count (two is cheap, four steps up, more climbs fast), the board size, the quantity, the surface finish, and the lead time you pick. For a small board the biggest levers are usually layer count and quantity, so a two-layer board bought in a modest batch is the cheap sweet spot most projects start from.

## ENIG VERSUS HASL FINISH

The surface finish is the coating on the exposed pads. HASL (hot-air solder levelling) is a tinned finish, cheap and solderable, with a slightly uneven surface. ENIG (electroless nickel immersion gold) is flat and gold-topped, better for fine-pitch parts and nicer to hand-solder, at a higher price. For most through-hole and larger surface-mount work HASL is fine; reach for ENIG when a part is fine-pitch or the flatness matters.

## THE ORDERING FLOW

Ordering is: upload your zipped gerbers and drill file, let the fab's checker parse them and show you a render, choose the layers, thickness, finish, color, and quantity, and place the order. Many fabs return a short DFM report flagging anything marginal in your files, which is a last free check before your board is committed to copper.

[Order this board at PCBWay](#) · Upload your zipped gerbers and drill file for a fabrication quote.

[Order this board at JLCPCB](#) · Low-cost two-layer and four-layer PCB fabrication.

THE COST DRIVERS: LAYER COUNT AND QUANTITY MOVE THE PRICE MOST, THEN SIZE, FINISH, AND LEAD TIME.

#### DEEP DIVE · PANELIZING TO SAVE ON SMALL BOARDS

A fabricator builds on a large sheet and charges partly by the area of it, so a tiny board leaves most of that sheet wasted. Panelization arrays several copies of your board into one panel that the fab builds and ships as a unit, which you then separate into individual boards, either by snapping them along a scored line or by breaking the small tabs that hold each board in a routed slot. For a first small run you usually let the fab arrange it or skip it; laying out your own panel becomes worthwhile once you are making many of one board.

#### CHECKPOINT

##### 1. Which choice usually drives a small board's cost the most?

- a. The silkscreen text
- b. The reference designators
- c. The layer count and quantity**

ANSWER · C

*Layer count and quantity move price far more than cosmetic choices.*

##### 2. When is ENIG the better surface finish?

- a. Only on the very cheapest boards
- b. For fine-pitch parts and when flatness matters**
- c. When you never plan to solder the board

ANSWER · B

*ENIG is flat and gold-topped, better for fine-pitch and hand-soldering; HASL is the cheaper default.*

##### 3. What is the safe way to treat a fab's minimum trace and hole sizes?

- a. Stay comfortably inside them**
- b. Design right at the minimum every time
- c. Ignore them and let the fab adjust

ANSWER · A

*Hugging the minimums yields worse and costs more; leave margin.*

- Prerequisite: gerbers and the fab package
- See it on a real board: the L1.01 build
- Next: soldering and assembly basics