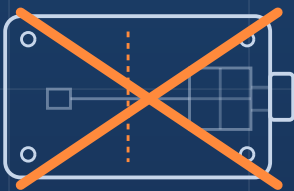


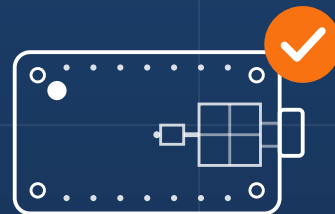
11 RULES TO NEVER BREAK

PCB LAYOUT CHECKLIST

Eleven layout rules that separate a board that passes EMI testing from one that resets randomly or has almost no wireless range. Plus what to do instead, for every rule.



RULES BROKEN
FAILS EMI TESTING



RULES FOLLOWED
PASSES FIRST TIME

PREPARED BY
JOHN TEEL

DOC NO.
PCB-11

REV
A

SOURCE
[PREDICTABLEDESIGNS.COM](https://predictabledesigns.com)

PCB Layout Rules Checklist

Eleven layout rules you should never break

Two boards can be built from the exact same schematic, and one works perfectly while the other fails EMI testing, resets randomly, or has almost no wireless range. The schematic is not the whole design, because it is really just an approximation of what actually gets built. The layout is where the real physics happens, and small placement and routing decisions can make or break your entire product.

This checklist covers eleven PCB layout rules you should never break, counting down to the one that is the most invisible and the most expensive to fix. For every rule you will find what goes wrong when you break it and exactly what to do instead, so run through the whole list before you order your next boards.

THE ELEVEN RULES AT A GLANCE

#	NEVER DO THIS	WHAT IT COSTS YOU
#11	Via-in-pad without knowing the cost	Starved joints and extra cost on every board
#10	A board with no test points	Units you cannot test in production
#9	No thermal path for hot components	Thermal shutdown and resets in the field
#8	High-speed traces near the board edge	Failed EMI scans and weeks of schedule
#7	A board without stitching vias	Plane gaps that leak radiation
#6	Ignoring the switching regulator hot loop	Noise sprayed across your whole board
#5	Differential pairs routed like plain traces	Random errors that are brutal to debug
#4	A starved power trace or via	Sagging rails and resets under load
#3	Copper under the antenna	Dead wireless range and lost module certification
#2	Decoupling caps far from the pin	Supply dips, random glitches, and resets
#1	Signals routed across a plane split	EMI failures and glitches nobody can reproduce

HOW TO USE THIS CHECKLIST

The rules run in countdown order from #11 to #1, the same order as the video. For every rule you will find what goes wrong when you break it, the exact fix, and how to apply it in your layout.

#11

Never Use Via-in-Pad Without Knowing the Cost

ASSEMBLY AND FAB COST

WHAT GOES WRONG WHEN YOU BREAK IT

- Via-in-pad means placing a via directly inside a component's solder pad instead of next to it. It is tempting because it saves space, and for dense parts like fine-pitch BGAs it is sometimes the only option.
- During assembly, the molten solder wicks down into the open via like a straw, which starves the joint and leaves you with weak or even open connections.
- The fix for that wicking is having your board fab fill each via with epoxy and plate copper over the top, and that process adds real money to every single board you ever build.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Place vias next to the pad	Unless you truly need via-in-pad, place vias beside the pad and connect them with a short trace.
Specify filled and capped vias	If your design does require via-in-pad, specify filled and capped vias in your fab notes, and budget for that extra cost from day one.

TWO COSTS MOST PEOPLE NEVER SEE COMING

The first cost shows up on the assembly line as weak or open joints. The second cost shows up on every fab invoice for the life of the product. Know both before you place a single via inside a pad.

#10

Never Design a Board Without Test Points

PRODUCTION TEST

WHAT GOES WRONG WHEN YOU BREAK IT

- A board with no test points cannot be tested in production. That means you are either shipping units that were never verified, or you are paying someone to probe every single board by hand.
- Factories test boards using a fixture with spring-loaded pogo pins, and those pins need dedicated copper pads to land on. If you did not put those pads in your layout, there is nowhere for the pins to go.
- Adding test points later means a new board revision.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Add test points to every power rail	Test points are basically free, since they are just small exposed copper circles placed on your signals and power rails. Put one on every power rail.
Cover every bus and debug signal	Add them to every communication bus and to anything you would need for programming or debugging the board.

A LIFESAVER DURING DEBUGGING

Test points also save you during debugging, especially with leadless packages like QFNs and BGAs where the pins are hidden underneath the chip and there is nothing to probe.

#9

Never Skip a Thermal Path for Hot Components

THERMAL

WHAT GOES WRONG WHEN YOU BREAK IT

- A voltage regulator that works fine for 10 minutes and then browns out is commonly a layout problem, not a component problem.
- Many regulators and other hot components dissipate their heat through an exposed pad on the bottom of the package called a power pad. That pad needs somewhere for the heat to go, and on most boards that means an array of thermal vias connecting it down to a large copper pour on another layer.
- Skip those vias, and the heat just piles up until the part hits thermal shutdown, and then your product resets over and over out in the field.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Follow the datasheet land pattern exactly	Copy the land pattern in the datasheet exactly, including the recommended via array under the power pad.
Give the part as much copper as you can spare	Copper is free, and it is the cheapest heatsink you will ever get.

YES, THERMAL VIAS ARE VIA-IN-PAD

Thermal vias are technically via-in-pad, but a power pad is the one place where it is normally acceptable, since a little solder wicking there mostly causes small voids instead of failed joints.

#8

Never Run High-Speed Traces Near the Board Edge

EMI

WHAT GOES WRONG WHEN YOU BREAK IT

- High-speed traces routed near the edge of the board are a classic way to fail EMI testing.
- Near the edge, the electric fields around a trace are no longer fully contained between the trace and its ground plane. Part of the field fringes out past the edge of the board, and that fringing field radiates like a little antenna.
- Everything will look fine on your bench, but the antennas at the test lab will pick it up immediately, and every failed EMI scan costs you money and weeks of schedule.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Keep fast signals away from the edge	Keep fast signals like clocks, USB, and memory buses away from the board edge by at least 4 to 5 times the distance between the trace and the ground plane below it. On most boards that works out to around 1 mm or more.
Route them on inner layers	Move high-speed signals to inner layers when you can, where the ground planes above and below contain the fields.

WHAT COUNTS AS HIGH-SPEED

Clocks, USB, and memory buses are the usual suspects. Treat any of them running along the board edge as a small antenna until you move it inward.

#7

Never Leave the Board Without Stitching Vias

GROUNDING

WHAT GOES WRONG WHEN YOU BREAK IT

- Stitching vias are the rows of ground vias that tie your ground planes together across the board.
- Without them, ground currents on different layers cannot take the shortest path back to their source, and the gaps between your planes start acting like slots that leak radiation.
- Stitching vias along the board edges also form a fence that helps contain noise inside the board.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Sprinkle ground stitching vias generously	Place them across the whole board, especially along the edges, around any high-speed routing, and around any RF circuits or antenna feedlines. Adding them costs nothing.

A BOARD I REVIEWED RECENTLY

I recently reviewed a board that looked clean everywhere else but had almost no stitching vias, and we caught it right before the fab order went out. Adding them cost nothing, and it meaningfully improved that product's odds of passing FCC testing on the first try.

#6

Never Ignore the Switching Regulator Hot Loop

POWER SUPPLY

WHAT GOES WRONG WHEN YOU BREAK IT

- Every switching regulator has a hot loop, which is the current path where current starts and stops in just a few nanoseconds.
- That fast-changing current makes the loop behave like a tiny transmitting antenna, and the bigger the loop area, the more noise it sprays across your board.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Place the input capacitor as close as possible	Put the input capacitor as close to the regulator as physically possible, and keep the loop on a single layer.
Copy the datasheet layout	Chip makers like TI and Analog Devices publish recommended layouts in their datasheets, and those layouts exist specifically to keep this loop tiny. Copy the datasheet layout instead of improvising your own.

LOOP AREA IS THE WHOLE GAME

The noise scales with the loop area, so every bit of distance you add between the regulator and its input capacitor turns directly into more noise sprayed across the rest of your board.

#5

Never Route Differential Pairs Like Ordinary Traces

HIGH-SPEED SIGNALS

WHAT GOES WRONG WHEN YOU BREAK IT

- Differential pairs like USB, Ethernet, and HDMI carry equal and opposite signals on two traces, and the receiver only reads the difference between them.
- This makes them extremely immune to noise, because interference hits both traces equally and cancels out at the receiver, but only when the layout is done correctly.
- Get the layout wrong and the signals arrive skewed or reflected, which turns into random errors that are brutal to debug.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Meet all four requirements	Both traces need matched lengths, a controlled impedance, an unbroken reference plane underneath, and no stubs hanging off the lines.
Use the differential pair tools	Your PCB software has built-in tools for routing differential pairs with matched lengths, so use them.
Ask your fab for a controlled stackup	Ask your board fab for a stackup with the impedance control you need, since they will gladly tell you the exact trace width and spacing to use.

#4

Never Starve a Power Trace or Via

POWER DELIVERY

WHAT GOES WRONG WHEN YOU BREAK IT

- A power trace that is too thin drops voltage under load, heats up, and in the worst case burns right off the board.
- The same goes for vias, since a single small via might only handle an amp or so before it becomes the bottleneck.
- Product creators often route power using the same default trace width as their signals, and everything seems fine at low current during early testing. Then the product hits a high-load condition, the supply rail sags, and the microcontroller resets at the worst possible moment.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Size every power trace for its actual current	Trace width calculators are free and take seconds to use, so size every power trace for its actual current with plenty of margin.
Use copper pours for higher currents	For higher currents, use copper pours instead of traces.
Parallel vias at every layer change	Always place multiple vias in parallel wherever power changes layers.

#3

Never Put Copper Under the Antenna

WIRELESS

WHAT GOES WRONG WHEN YOU BREAK IT

- Copper placed under an antenna will kill your wireless range.
- An antenna works by coupling energy into the space around it, and a ground plane or any copper sitting underneath detunes the antenna and absorbs the energy before it ever leaves the board.
- A pre-certified module is only certified when it is used the way the datasheet specifies. Violate the keepout, and the module in your product no longer performs as certified, which can come back to bite you during compliance testing.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Enforce the keepout zone on every layer	Every antenna and every pre-certified wireless module has a keepout zone defined in its datasheet, which is an area where you are not allowed to put copper, traces, or components on any layer.
Keep metal hardware away	Keep metal mounting screws and standoffs away from the antenna too, since metal that close will detune it just like copper does.
Copy the dev board	Look at any dev board for a wireless module and you will find a bare region around the antenna. Copy that keepout exactly in your own layout.

#2

Never Place Decoupling Caps Far From the Pin

POWER INTEGRITY

WHAT GOES WRONG WHEN YOU BREAK IT

- Decoupling capacitors only work when they are close to the power pin they are protecting.
- A decoupling cap acts like a tiny local energy tank that supplies the chip during sudden current spikes, faster than the main power supply can respond.
- Every mm of trace between the cap and the pin adds inductance, and inductance resists exactly those fast current changes. Put the cap too far away, and you have built a filter that blocks the cap from doing its one job, so the chip's supply dips and you get random glitches and resets.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Smallest cap closest to the pin	Place the smallest cap closest to the pin, before any larger capacitors.
Short, wide traces	Connect the cap to the pin with short wide traces to keep the inductance as low as possible.
Ground vias right at the pads	Drop the ground vias right at the cap's pads instead of running a trace to reach one.

#1

Never Route Signals Across a Plane Split

RETURN PATHS

WHAT GOES WRONG WHEN YOU BREAK IT

- Routing a signal across a split in your ground or power plane is the most invisible and most expensive mistake on this entire list.
- Every signal current has to return to its source, and at high frequencies that return current flows in the plane directly underneath the trace. When the trace crosses a split, the return current cannot follow it, so it detours all the way around the gap.
- That detour creates a huge loop that radiates noise, couples interference into every nearby circuit, and is a leading cause of both EMI failures and mystery glitches nobody can reproduce.
- Nothing about it looks wrong, since the trace appears perfectly normal and every net checks out in your design software.

DO THIS INSTEAD

THE FIX	HOW TO APPLY IT
Check every trace against the planes beneath it	Before you order boards, check every trace against the planes beneath it, on every layer, and reroute anything that crosses a gap.

SPLITS THEMSELVES ARE NOT THE PROBLEM

Plane splits are sometimes the right choice, like separating a sensitive analog section. The rule is about what you route over them, not whether they exist.

Run This Checklist Before You Order Boards

Eleven boxes to tick before your layout goes to the fab

- ☐ **Via-in-pad:** Vias sit next to their pads with a short trace, or filled and capped vias are called out in the fab notes and budgeted from day one.
- ☐ **Test points:** Every power rail, every communication bus, and every programming or debug signal has a dedicated copper pad for the pogo pins to land on.
- ☐ **Thermal path:** Every power pad follows the datasheet land pattern, including the thermal via array, with as much copper area as the board can spare.
- ☐ **Board edge:** Clocks, USB, and memory buses stay at least 4 to 5 times the trace-to-plane distance from the board edge, and run on inner layers where possible.
- ☐ **Stitching vias:** Ground stitching vias run across the board, along the edges, around high-speed routing, and around any RF circuits and antenna feedlines.
- ☐ **Hot loop:** Every switching regulator's input capacitor sits as close to the part as physically possible, on a single layer, following the datasheet layout.
- ☐ **Differential pairs:** USB, Ethernet, and HDMI pairs have matched lengths, controlled impedance, an unbroken reference plane underneath, and no stubs.
- ☐ **Power traces:** Every power trace is sized for its actual current with margin, higher currents use copper pours, and layer changes use multiple vias in parallel.
- ☐ **Antenna keepout:** No copper, traces, components, or metal hardware sit inside the antenna keepout zone on any layer, matching the dev board exactly.
- ☐ **Decoupling caps:** The smallest cap sits closest to each power pin, connected with short wide traces, with ground vias right at the cap's pads.
- ☐ **Plane splits:** Every trace has been checked against the planes beneath it on every layer, and nothing crosses a gap.

Get Experienced Engineers On Your Layout

If you would like experienced engineers to review your layout and catch these exact mistakes before you spend money on boards, that is exactly what my team and I do inside the Hardware Academy.

[JOIN AT PREDICTABLEDESIGNS.COM/ACADEMY](https://predictabledesigns.com/academy)