Flux

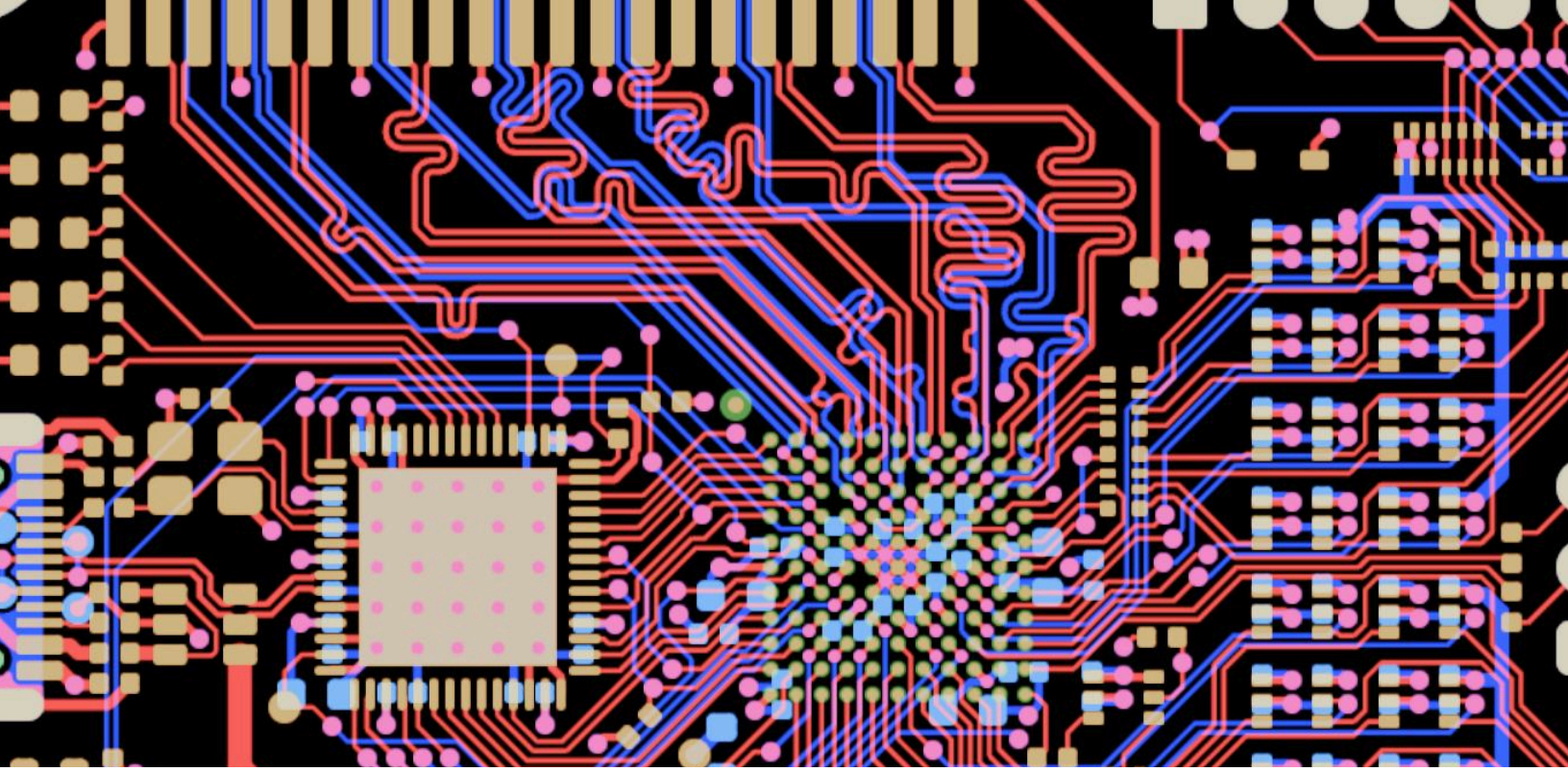
High Speed Design & Signal Integrity

Go beyond impedance matching with proven techniques that keep your high-speed signals clean, sharp, and compliant.

flux.ai

TABLE OF

Contents



Getting Started with High Speed Design

Dealing with signal and power integrity may feel overwhelming at first, but you're not alone. In this guide, we'll walk you through the essential concepts, explaining each term so you can build confidence as you go. Think of this as a cheat sheet to help you avoid late-stage surprises and board re-spins.

Why This PDF Matters

Every designer faces tight schedules and evolving requirements. Unfortunately, SI/PI problems often hide in subtle parasitic effects like tiny reflections on a trace or momentary ground bounce under a power pin. By following a clear, step-by-step workflow—built on best practices used in industry—you'll catch these issues early, save time (and money), and sleep better knowing your design is robust.

Signal Integrity Classifications

Signal integrity issues generally fall into three categories:

1. **Reflections (Return Loss)**: Mismatches in impedance (at connectors, vias, or trace discontinuities) cause portions of the signal to reflect back toward the source. These reflections can superimpose on the original waveform, creating ringing, overshoot, or undershoot.
2. **Insertion Loss (Attenuation & Distortion):** Conductive and dielectric losses in the PCB trace, connector, or cable attenuate high-frequency components and distort edge rise/fall times. Excessive insertion loss narrows eye openings and increases jitter.
3. **Crosstalk (Coupling Noise):** Capacitive and inductive coupling between adjacent traces transfers unwanted energy from an “aggressor” to a “victim” line. Crosstalk can appear at the near end (NEXT) or far end (FEXT) and degrades signal-to-noise ratio.

Each classification requires specific mitigation strategies—termination for reflections, material selection and equalization for insertion loss, and spacing or shielding for crosstalk.

Begin Assembling Your Project Requirements

ITEM	DESCRIPTION
Performance Targets	Timing budgets, maximum BER, and regulatory emission limits (e.g., FCC, CE)
Signal Specifications	Datasheet and measured rise/fall times, voltage swings, encoding schemes
Material & Stack-Up Data	Dielectric constants (Dk), loss tangents, copper weights, and thickness
Tools & Instruments	Field-solvers, oscilloscope (≥5×BW), TDR, VNA, layout software
Layout Constraints	Board area, layer count, via budget, design rules from your PCB house

> **Pro Tip:** Use a version-controlled checklist (e.g., Git) for requirements. Update it with every specification change to keep your entire team aligned.

By clearly defining these items, you can focus on the technical trade-offs that matter most. Keep this table visible as a living document—refer back whenever you make design decisions.

STEP 1

Characterize Your Signals

Accurate characterization of your signals is critical. Underestimating rise times or ignoring harmonic content often leads to unexpected reflections, jitter, and eye closure.

1. **Rise and Fall Time**
   - This measures how quickly a signal switches from its low level (10%) to its high level (90%), or vice versa. A shorter transition time means the signal has more high-frequency components, which can lead to reflections and ringing if not managed properly.
   - How to measure: Use a Time Domain Reflectometer (TDR) or a high-bandwidth oscilloscope. Ensure your measurement tools have at least five times the bandwidth of your expected signal edge to capture accurate results.
   - Why it matters: Slow or mismatched edges can cause timing errors, increase jitter, and reduce eye opening in your signal.

2. **Voltage Swing & Levels**
   - In single-ended signaling, your signal swings between a fixed low voltage (commonly 0 V) and a fixed high voltage (e.g., 3.3 V). Think of it like a light switch that turns entirely off or on against ground.
   - In differential signaling, two wires carry equal but opposite voltages around a middle reference (common-mode) point. The receiver looks at the difference between these wires, which helps cancel out noise that affects both equally.
   - Baseline Wander happens when the average voltage shifts over time, causing your “off” or “on” levels to drift. That can confuse the receiver’s threshold for deciding a 0 vs. 1.
   - Noise Margin is the safety buffer between your signal levels and the receiver’s decision thresholds. If your swing is too small or shifts, that buffer shrinks and data errors become more likely.

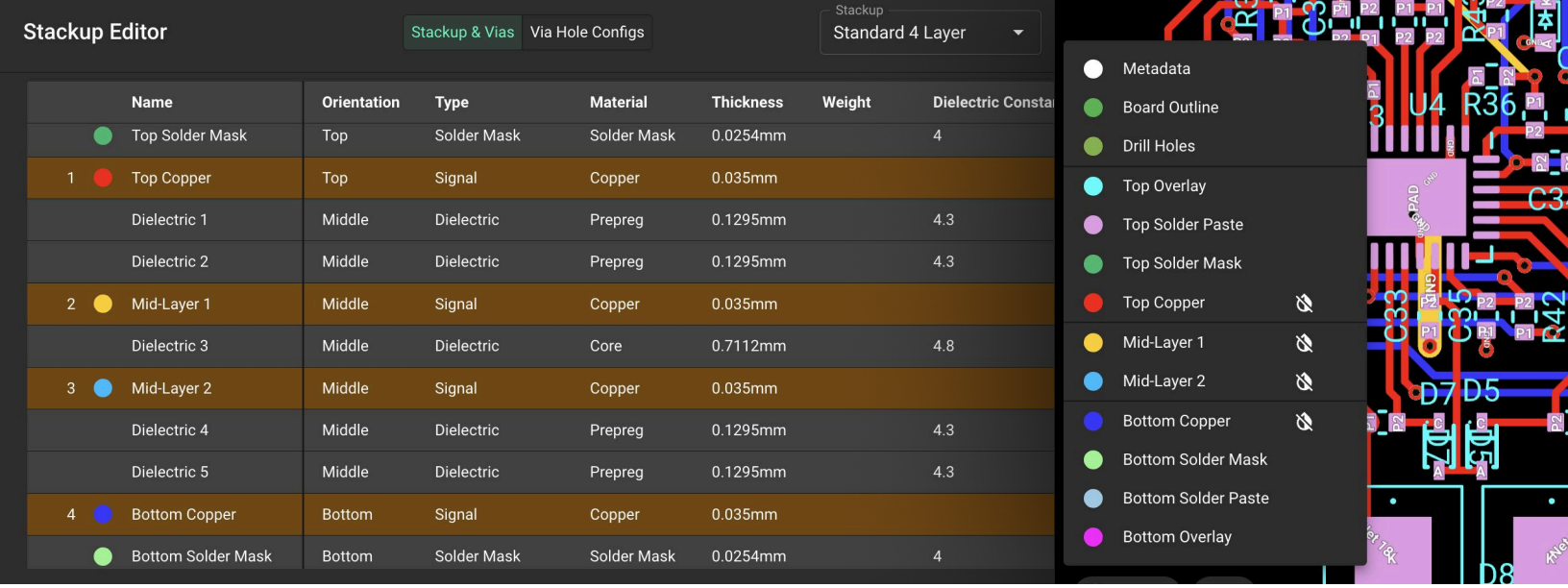
3. **Data Rate & Encoding**
   - Data Rate refers to how many bits your channel moves each second. For instance, 1 Gbps means 1 billion bits per second. However, not all these bits carry actual “message” data—some are reserved for framing, error checking, or balancing the signal, which we call overhead. For example, 8b/10b encoding sends 10 bits on the wire for every 8 bits of real data, so you only get 80% of the raw rate as useful payload.

- Encoding Schemes like 8b/10b or PCIe's 128b/130b help maintain a steady voltage average (avoiding long runs of 0s or 1s) and embed clock information so the receiver stays synchronized. While these schemes improve reliability and timing recovery, they also increase the total bits sent and can make equalization—the process of undoing signal distortion—more challenging.

4. **Bandwidth Estimation**
    - What it means: Bandwidth describes the band of frequencies your digital pulse occupies. A wider bandwidth preserves sharper edges; a narrower one rounds them off.
    - Approximate calculation: For a simple edge with a single-pole roll-off, estimate analog bandwidth as: , where t**r is the 10%→90% rise (or fall) time.
    - Harmonic content: Digital edges build from sine waves at multiples of the fundamental frequency. To keep the edge shape, include up to the 5th harmonic (i.e., 5× the bit rate) when that harmonic is still within ~20 dB of the fundamental.
    - Why it matters: If your PCB or cable attenuates these frequencies too much, edges blur, causing inter-symbol interference (ISI) and reducing your eye opening.
    - Layman analogy: It's like the difference between a crisp photograph (full detail) and a blurred one (missing high-frequency details)—you need enough bandwidth to keep your "picture" sharp.

Properly documenting these values allows you to calculate channel loss, dispersion, and reflections with confidence.

> **Pro Tip:** If you only know datasheet values, add a 20% margin when budgeting bandwidth to account for real-world effects.



STEP 2

Material & Stack-Up Selection

The PCB stack-up profoundly influences impedance control, interlayer coupling, plane resonances, and power distribution. A well-designed stack-up reduces SI/PI issues, improves EMI performance, and streamlines layout iterations.

MATERIAL	DIELECTRIC CONSTANT (Dk)	LOSS TANGENT (TAN δ)	COMMON APPLICATIONS
Standard FR-4	4.3 – 4.8	0.02 – 0.03	General-purpose digital/ analog boards
High-Speed FR-4	3.8 – 4.2	0.015 – 0.02	Up to 2.5 Gbps SerDes, moderate-frequency RF
Rogers 4350B	3.66	0.0037	RF/microwave, mmWave, high-Q circuits
Megtron 6	3.55	0.0025	56 Gbps+ SerDes, data-center interconnects

Key Considerations

- **Plane Pairing:** Power/ground plane pairs act as distributed decoupling capacitors—adjacent copper layers lower PDN impedance.
- **Resonances:** Compute plane cavity resonances using $f_{\text{res}} = \frac{c}{2 \cdot \sqrt{\varepsilon_r} \cdot d}$ where d is plane separation, to avoid SI/EMI hotspots.

Layer Stack-Up Considerations (2–6 Layer PCBs)

Select the number of layers and assign roles based on signal count, speed, and power needs:

NUMBER OF LAYERS	TYPICAL CONFIGURATION	NOTES
2 	• Top: signals • Bottom: reference (GND)	Simple, but poor isolation; only one reference layer
4 	• Top: signals • Layer 2: reference (GND) • Layer 3: power (VCC) • Bottom: signals	Standard high-speed; clear separation of signal and planes
6 	• Top: signals • Layer 2: reference (GND) • Layer 3: signals/high-speed pairs • Layer 4: power (VCC) • Layer 5: reference (GND) • Bottom: signals	Supports multiple high-speed buses; dual reference planes improve isolation

Design Guidelines

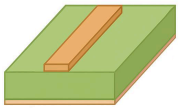
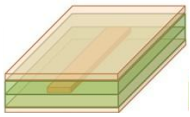
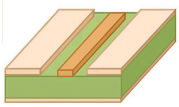
- **Signal Layers:** Place the most critical high-speed nets on internal layers adjacent to reference planes to maintain controlled impedance.
- **Ground Planes:** Always dedicate at least one continuous GND layer; avoid splits under high-speed routes.
- **Power Planes:** Pair power/ground layers to form capacitance; for multi-layer boards, spread VCC planes centrally to minimize loop inductance.
- **High-Speed Layers:** Use internal microstrip or stripline layers to shield critical nets from external noise; prefer stripline for ultra-high-speed nets.

> **Pro Tip:** For 4-layer boards, route all high-speed differential pairs on the internal layer adjacent to the ground plane for best impedance control.

STEP 3

Trace Geometry & Impedance Control

Each trace behaves like a distributed RLC network. Precise geometry ensures the line's characteristic impedance (Z0) matches driver and receiver.

	GEOMETRY	Z0 FORMULA (APPROXIMATE)
	**Microstrip**	$Z_0 \approx \frac{87}{\sqrt{D_k + 1.41}} \ln\left(\frac{5.98h}{0.8w + t}\right)$
	**Stripline**	$Z_0 = \frac{60}{\sqrt{D_k}} \ln\left(\frac{4h}{0.67\pi(w + t)}\right)$ $Z_0 = \frac{60}{\sqrt{\varepsilon_r}} \ln\left(\frac{1.9B}{0.8W + T}\right)$
	**Coplanar**	Use numerical conformal mapping or 2D field solver; Z0 depends on w, s (spacing), t, and h of adjacent planes

- w: trace width, s: gap to adjacent ground, h: dielectric height, t: copper thickness.
- **Skin Effect:** At high frequencies, current crowds near the conductor surface, increasing effective resistance.
- **Dielectric Loss:** Energy dissipated in the board material increases attenuation with frequency.

Use full 2D/3D field-solvers when tight tolerances (<±5%) are required, or when using non-standard dielectrics.

> **Pro Tip:** When adjusting for manufacturing tolerances, consult your PCB vendor's copper weight and etch factor to back-calculate w and s.

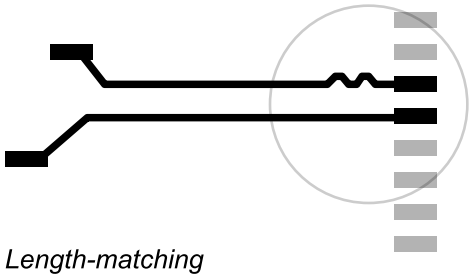
STEP 4

Routing Topology

Topology dictates signal path integrity, skew, and mode conversion:

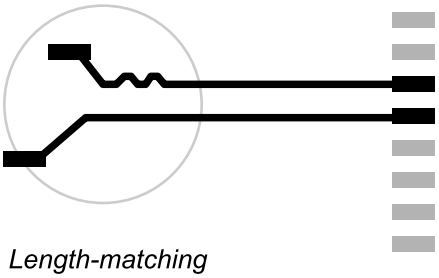
- **Differential-Pair Routing:** Maintain constant width (w) and spacing (s); avoid layer transitions to prevent impedance changes; trace symmetrically and parallel to each other.
- **Length Matching:** Use serpentine traces for fine adjustments; max allowable skew < 3% of bit period (e.g., <12 ps at 10 Gbps).
- **Stub Elimination:** Remove unused branch connections; if unavoidable, keep stub lengths < ¼ of the shortest wavelength to minimize reflections.
- **Avoid Layer Changes in Critical Paths:** Each via adds ~50 ps of delay and discontinuity; if needed, use back-drilling to remove via stubs.

DON'T



Length-matching at Matched Ends

DO



Length-matching at Mismatched Ends

DON'T

DO

Differential Pair Symmetry

> **Pro Tip:** Use 45° bends or mitered corners; right-angle turns add capacitance and radiate more.

STEP 5

Termination & Loss Mitigation

Proper termination absorbs energy at discontinuities:

TYPE	BEHAVIOR	SELECTION CRITERIA
SERIES	Source series resistor forms a match at launch; reduces overshoot but slightly slows edge rate.	Rseries ≈ Z0 – Zdriver
PARALLEL	Shunt resistor at receiver to match Z0; best for minimizing reflections at load.	Rshunt = Z0
THEVENIN	Voltage-divider network at driver for both edges; splits energy across resistors.	R1 + R2 ≈ 2Z0, Vout= VCC·R2/(R1+R2)

Advanced Techniques:

- **Pre-Emphasis/De-Emphasis:** Boost high-frequency content at the transmitter to counteract channel loss.
- **Receiver Equalization:** Adaptive filters in the RX compensate for inter-symbol interference (ISI), improving eye opening.

> **Pro Tip:** Monitor eye height and width; if bottom leg droops, add pull-up termination or adjust equalization settings.

STEP 6

Cross-Talk Analysis

Coupling mechanisms between adjacent PCB traces are primarily governed by two parasitic effects:

- **Mutual Capacitance (Cm):** When two conductors run in parallel, an electric field forms between them, creating a parasitic capacitance. Rapid voltage changes on the “aggressor” trace inject displacement currents into the adjacent “victim” trace, causing unwanted voltage coupling. Cm coupling is strongest at high frequencies and for closely spaced nets.
- **Mutual Inductance (Lm):** Time-varying currents in one trace produce a changing magnetic field that links to nearby traces, inducing a voltage proportional to the rate of change of current ($V = Lm \cdot di/dt$). Lm coupling is broadband and particularly significant for fast edge rates and longer parallel runs.

Types of Crosstalk:

- **NEXT (Near-End Crosstalk)**: Measured at the driving end of the victim trace, NEXT occurs when coupled energy travels backward toward the source. It can interfere with transmitters or adjacent receivers and typically dominates at shorter separations.
- **FEXT (Far-End Crosstalk):** Measured at the far end (load end) of the victim trace, FEXT results from coupled energy propagating in the same direction as the signal. FEXT magnitude grows with trace length but is often partially attenuated by the transmission-line losses in the channel.
- **Residual Crosstalk:** The combination of FEXT and NEXT that remains after mitigation measures, often characterized in mixed-mode S-parameters (Sdd21 for differential-to-differential, Scd21 for common-mode conversion).

Enhanced Mitigation Strategies

STRATEGY	NOTES
Optimal Spacing	Increase spacing between parallel traces. ≥3× trace width reduces Cm coupling by >80%; ≥5× for ultra-high-speed lanes.
Guard Traces & Stitching	Insert grounded guard traces between aggressive nets; stitch to reference planes with vias (1–2 mm) to confine parasitic fields.
Differential Pair Routing	Route tightly coupled differential pairs to convert common-mode crosstalk into rejectable differential signals; maintain constant geometry.
Layer Stack-Up Optimization	Place high-speed signals on internal stripline layers between ground planes; alternate signal and reference layers for shielding.
Controlled Return Paths	Ensure continuous reference planes under all high-speed routes; avoid plane splits and add return vias adjacent to signal vias.

> **Pro Tip:** When validating differential links, examine mixed-mode S-parameters (Sdd21, Scd21) to quantify differential and common-mode crosstalk, then refine pair spacing or stack-up as needed.

STEP 7

PDN Design

Your power-distribution network (PDN) must deliver fast transient currents to ICs while keeping voltage ripple within the specified tolerance (often <5% of VDD). Poor PDN design can lead to logic failures, jitter, and thermal stress. The following provides an in-depth breakdown of PDN design elements and decoupling strategies.

STRATEGY	NOTES
Target Impedance (Ztarget)	• Sets the maximum PDN impedance across frequency. • Compute Ztarget: $\Delta V_{\max} = \Delta I_{\max} \times Z_{\text{target}} \quad (\Delta V \text{ often } \leq 3\text{–}5\% V_{\text{DD}}$ • Example: For ΔI=1 A and ΔV=50 mV, Ztarget=50 mΩ.
Guard Traces & Stitching	• Bulk Caps (≥10 µF): Supply energy for low-frequency events (<10 kHz), placed near board entry or regulator. • Mid-Frequency Caps (0.1–1 µF): Bridge cavities around 100 kHz–10 MHz; distribute grid-like across board. • High-Frequency Caps (1 nF–100 nF): Handle fast edges (≥100 MHz); located <0.5 mm from each IC power pin to achieve <1 nH loop inductance.
Differential Pair Routing	• **Via Inductance:** Typical via contributes 0.5–1 nH; minimize with short, multiple parallel vias. • **Loop Inductance:** Total pad-to-pad path; target <1 nH by minimizing cap pad spacing and via length. • **Stacked Microvias:** In high-density designs, microvia-in-pad can reduce inductance to <0.2 nH.
Layer Stack-Up Optimization	• **Plane Cavity Resonances:** $f_{\text{res}} = \frac{c}{2 \cdot \sqrt{\varepsilon_r} \cdot d}$ where d is plane separation. • **Damping Techniques:** Scatter decoupling caps to break up standing waves; consider ferrite beads on bulk feeds to damp high-Q peaks.

Detailed PDN Design Workflow

1. **Model & Simulate**
   - Measure plane-pair S-parameters with a VNA; build a Z-matrix PDN model.
   - Simulate impedance vs. frequency using SPICE or PI tools (Keysight PIPro, HyperLynx PI).

2. **Place Decoupling Caps**
   - Mount HF caps (<100 nF) within 0.5 mm of power pins; ensure loop inductance <1 nH.
   - Distribute mid-frequency caps to fill impedance troughs.
   - Cluster bulk caps at the power entry and regulator outputs.

3. **Optimize Layout**
   - Use wide power/ground pours with stitching vias beneath high-current areas.
   - Position cap pads adjacent to vias; avoid via stubs under critical nets.

4. **Validate & Tune**
   - Perform impedance scan from 10 kHz to >1 GHz; verify ZPDN remains below Ztarget.
   - Overlay transient load profiles to confirm voltage droop meets ΔV spec.

> **Pro Tip:** Plot PDN impedance on a log-log scale. Address any peaks above Ztarget with targeted decoupling at those frequencies.

STEP 8

EMI Prevention & Best Practices

Unchecked SI/PI issues often manifest as electromagnetic interference (EMI), which can compromise system functionality and fail compliance testing. Below is a concise table summarizing the four primary EMI concerns and their core mitigation strategies, followed by key elaborations.

EMI CONCERN	ISSUE DESCRIPTION	KEY MITIGATION
Return-Path Continuity	Splits or gaps in reference planes force return currents on longer loops, increasing radiated emissions and loop inductance.	Ensure solid, continuous ground/power planes; add stitching vias at layer transitions.
Ferrite Beads & Chokes	Undesired high-frequency currents on power or signal lines propagate noise and degrade EMC performance.	Place ferrite beads with impedance peaks at noise frequencies close to noise sources; respect DC current ratings.
Cable Shielding & Grounding	Poor shield coverage or multiple ground connections create ground loops, injecting noise back onto shields and into the system.	Use ≥85% braided/foil shields; tie shields at a single chassis-ground point; ensure robust connector bonding.
Edge Rates vs. Emissions	Fast dV/dt transitions generate broad-spectrum emissions that can exceed regulatory limits.	Add series resistors (5–20 Ω) or adjust gate-drive slew rates to reduce high-frequency content by 10–20 dB.

> **Detail:** Every high-speed signal must have a closely adjacent return conductor—typically a solid reference plane. Confirm no signal crosses a plane split without return stitching, as even small gaps can double loop inductance.
>
> **Detail:** Ferrite beads should be selected based on their impedance vs. frequency profile. Place them immediately at the source of switching noise (e.g., regulator output or connector pin) to maximize attenuation.
>
> **Detail:** Cable shields act as a Faraday cage. Avoid ground loops by connecting shields at one point only, and verify ≥85% coverage in braiding or foil to intercept radiation effectively.
>
> **Pro Tip:** Use near-field probes (electric and magnetic) during pre-compliance scans to locate hot spots, then apply targeted fixes from the table before full EMC testing.

Verification & Sign-Off

A rigorous sign-off combines simulation and measurement:

- **S-Parameter Simulation:** Extract or measure single-ended and mixed-mode S-parameters; simulate insertion loss, return loss, and crosstalk up to Nyquist.
- **Time-Domain Measurements:** Use TDR for impedance profile, and high-speed scope for eye diagrams and jitter analysis.
- **PDN Impedance Scan:** Network-analyzer sweep from 10 kHz to several GHz; ensure ZPDN < Ztarget.
- **EMC Pre-Scan:** Quick radiated/conducted emissions check in a semi-anechoic chamber; confirm margin to regulatory limits.

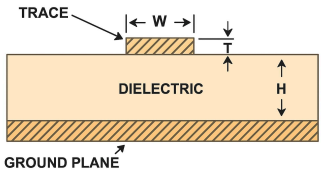
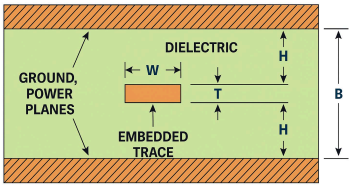
Document all findings, attach relevant plots and screenshots, and archive for future reference.

Cheat Sheet

Core SI/PI Concepts

- **Reflections:** Impedance mismatches cause signal echoes, leading to ringing and overshoot. Terminate lines to match Z_0.
- **Insertion Loss:** Conductor skin effect and dielectric loss attenuate high-frequency content, closing eye openings. Compensate with equalization and low-loss materials.
- **Crosstalk:** Capacitive (Cm) and inductive (Lm) coupling transfers noise between nets. Mitigate via spacing, guard traces, and differential routing.
- **Impedance Control:** Z_0 depends on geometry (w, h, t) and dielectric constant (Dk). Use formulas or field solvers to maintain ±5% tolerance.
- **PDN Target Impedance:** Z_target = ΔV / ΔI; hierarchical decoupling (bulk, mid, HF) keeps Z_PDN below this limit.
- **EMI Prevention:** Continuous return paths, proper shielding, and controlled edge rates (add 5–20 Ω series) reduce radiated emissions.

Key Formulas

QUANTITY	FORMULA	NOTES
Bandwidth (Hz)	$\mathrm{BW} \approx \frac{0.35}{t_r}$	tr: 10-90% transision time
Microstrip Z_0	$Z_0 \approx \frac{87}{\sqrt{D_k + 1.41}} \ln\left(\frac{5.98h}{0.8w + t}\right)$	
Stripline Z_0	$Z_0 = \frac{60}{\sqrt{D_k}} \ln\left(\frac{4h}{0.67\pi(w+t)}\right)$ $Z_0 = \frac{60}{\sqrt{\varepsilon_r}} \ln\left(\frac{1.9B}{0.8W+T}\right)$	
Plane Resonance Frequency	$f_{\text{res}} = \frac{c}{2 \cdot \sqrt{\varepsilon_r} \cdot d}$	d: plane separation
PDN Target Impedance	$Z_{\text{target}} = \frac{\Delta V}{\Delta I}$	ΔV: allowable ripple, ΔI: transient current

Quick-Pro Tips

- **Version-Control Checklists:** Track evolving requirements to keep all stakeholders aligned.
- **HF Decoupling:** Place 1 nF–100 nF caps within 0.5 mm of IC pins to achieve <1 nH loop inductance.
- **Series Termination:** Add 5–20 Ω at source to slow edges and cut EMI by ~10 dB.
- **Guard Traces:** Grounded traces stitched every 1–2 mm intercept crosstalk and confine fields.
- **Field-Solver Calibration:** Verify closed-form impedance with 2D solvers; account for vendor etch factors.

Use this cheat sheet as a rapid reminder during layout reviews, simulations, and pre-compliance checks.

Additional Resources

For deeper dives and real-world examples, explore these authoritative resources:

CATEGORY	TITLE & AUTHOR
Books	• High-Speed Digital Design by Howard Johnson & Martin Graham • Signal Integrity and Radiated Emission by Eric Bogatin • Transmission Lines and Waveguides by Jovan Popović
Application Notes	• TI AN-1186: Layout Guidelines for PCB Design • Intel® Signal Integrity Design Guide (2014) • Keysight Technologies: PCB Transmission Lines • TI High Speed Interface Layout Guidelines
Papers & Tutorials	• “A Practical Guide to AC Power Distribution” by P. M. Anderson • ANSYS HFSS Signal Integrity Application Note

> **Note:** Many vendors (TI, Analog Devices, Xilinx) publish detailed application notes on SI/PI topics; check their websites for the latest updates.
>
> **Tip:** Bookmark IEEE Xplore and vendor-design repositories to stay current with emerging best practices.

Author

Ryan Fitzgerald