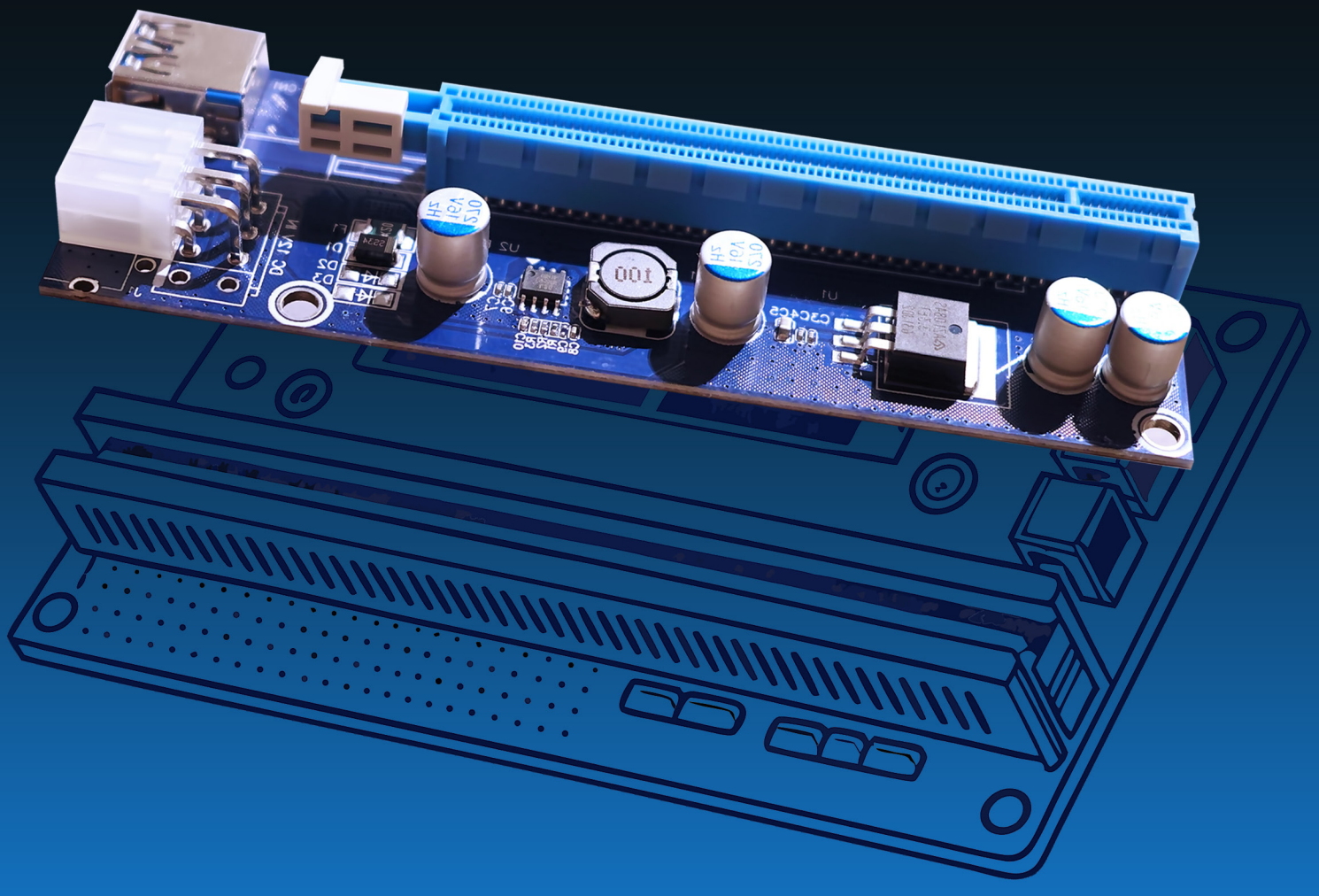
DESIGN **GUIDE**

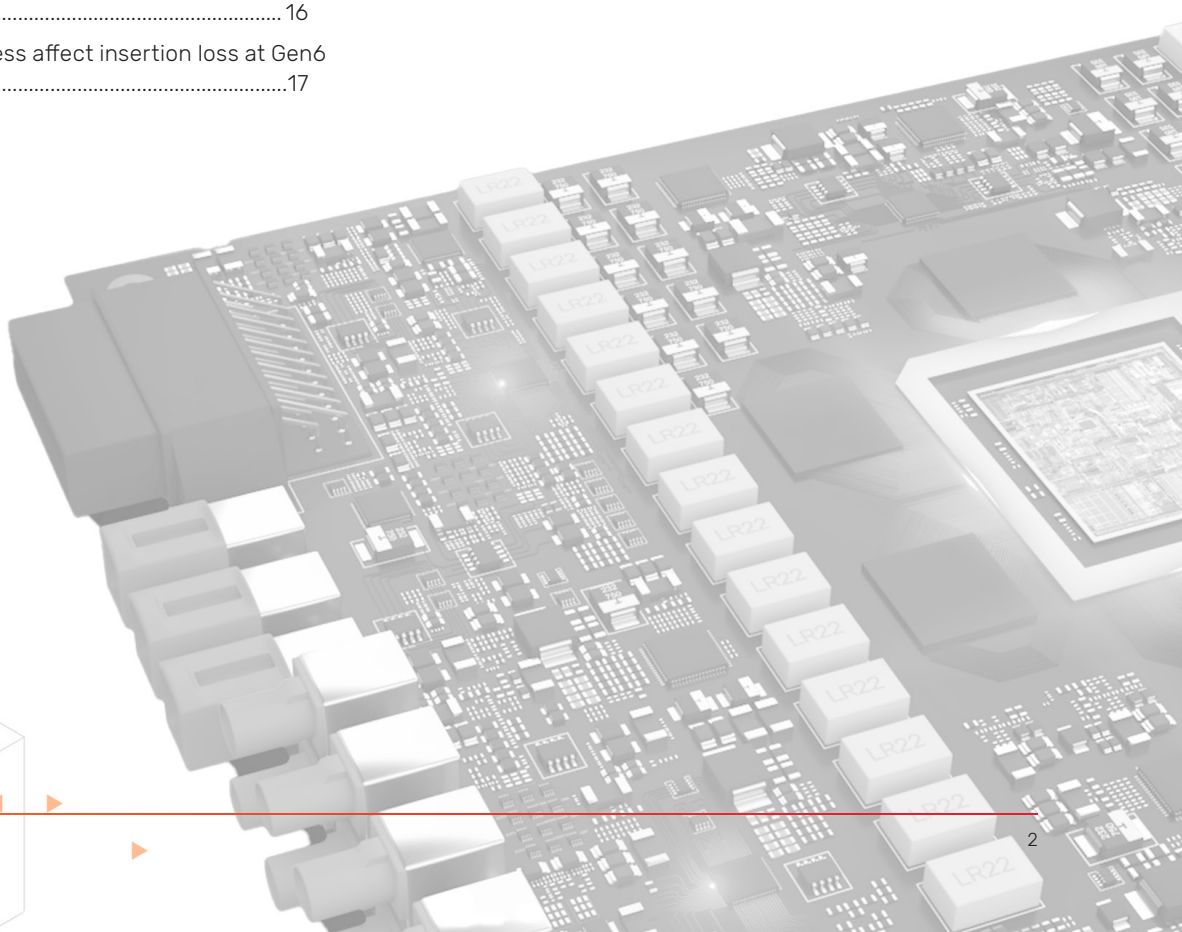
cadence®

PCIe Design Guide – Q&A (Gen 4, 5, 6) - Part 1

For Engineers Building High-Speed PCB Systems



Contents



Routing & Stackup

1. How do vias affect high-speed PCIe signals?

Vias are vertical interconnects between PCB layers, and in high-speed designs, they act as lumped parasitic elements. These parasitics affect both the impedance and the bandwidth of the channel.

Key parasitic behaviors:

- Capacitance: Pad and antipad regions introduce localized capacitive loading.
- Inductance: The via barrel behaves like an inductor, impeding high-frequency currents.
- Stub resonance: If a via is not fully traversed (e.g., signal stops at layer 4 but via goes to layer 8), the unused portion forms a resonant stub, reflecting high-frequency energy.

Why It Matters:

A via stub as short as 20 mils can resonate at 8-16 GHz, which overlaps Gen4/Gen5 signaling. For PCIe Gen6, signaling extends up to 32 GHz, where via stubs become even more disruptive. These reflections distort the eye diagram, degrading return loss, and introduce ISI (inter-symbol interference). Vias also interrupt the reference plane, affecting return current continuity and creating common mode noise or mode conversion, both of which are detrimental at Gen6 speeds.

Engineering Tip:

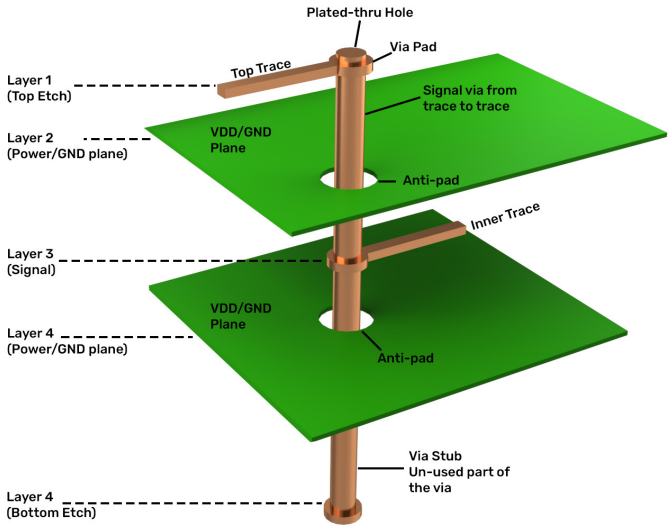
Use backdrilling to eliminate unused via stubs and avoid square antipads. Model full via structures including pads, antipads, and plating, in 3D EM solvers (e.g., Cadence Clarity).

2. How do pad sizes and anti-pad geometries affect via performance?

The pad and antipad geometry of a via govern its parasitic capacitance and inductance, which strongly influence local impedance. At high frequencies, vias behave as resonators, especially when stubs are present or poorly damped. Pad and antipad design is critical to maintaining low return loss and minimizing discontinuities at via transitions.

- Larger pads increase capacitance, lowering the local impedance.
- Smaller or asymmetric antipads (the void in the reference plane) increase inductance and can result in incomplete field containment or unwanted resonance.
- Stub length determines resonance frequency; even 20 mils can resonate near 8-16 GHz.

Target impedance for via transitions is still 100 Ω, but it's rarely achieved without simulation and tuning.



Basic via configuration in 4 layer stackup

Why It Matters:

Gen4/Gen5 Nyquist frequencies, via transitions can significantly affect the signal path. At Gen6 frequencies (up to 32 GHz), the situation becomes even more sensitive: parasitic capacitance, inductive loading, and stub resonances can shift return loss by more than 10 dB, reducing signal margin, collapsing the eye, or causing differential to common mode conversion. These effects are not isolated to individual nets, they propagate back through the channel, degrading equalizer performance and introducing inter-symbol interference (ISI). Even when the trace layout looks correct, a poor via transition can become the dominant source of channel loss and compliance failure.

Engineering Tip:

Via geometries should be carefully optimized for high-speed performance. A good starting point is:

- Drill size: 8-10 mils
- Pad size: 18-22 mils
- Antipad: 30-35 mils (round)
- Backdrill stubs and avoid square or “dogbone” antipads that introduce resonant cavities.
- Simulate all via structures in a 3D EM solver, especially when designing breakout regions, to ensure broadband impedance continuity cross the full channel bandwidth, especially for Gen 6 designs operating above 30 GHz.

3. What trace separation rules should be followed?

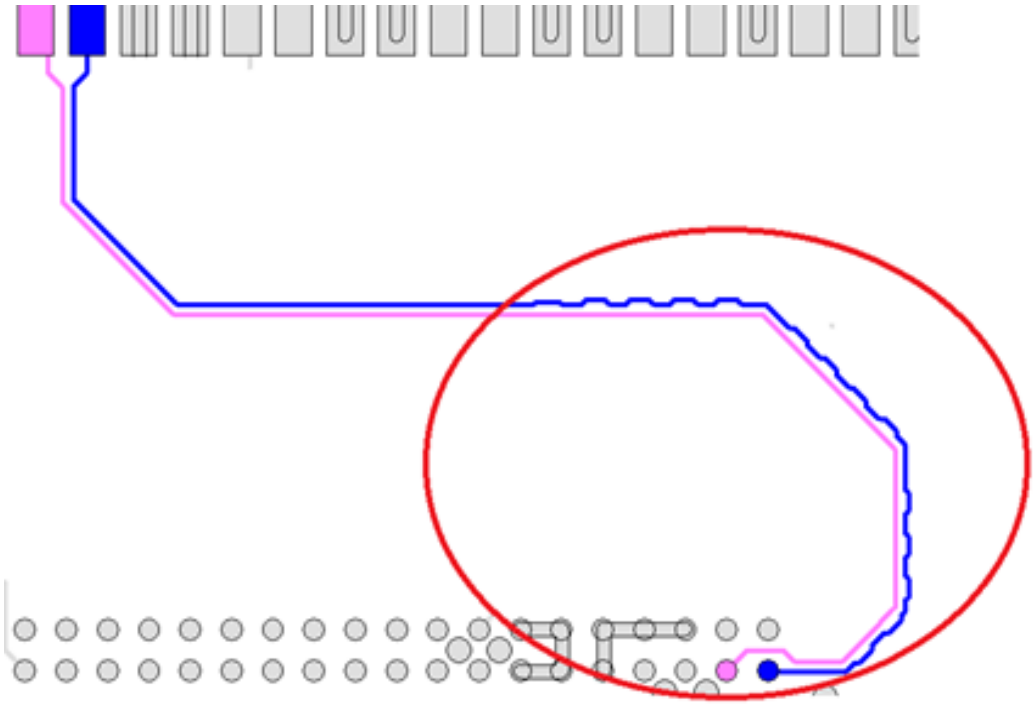
Maintaining appropriate trace separation is critical in high-speed PCIe designs to ensure controlled impedance, minimize crosstalk, and preserve differential symmetry. This becomes increasingly important at Gen5 (32 GT/s) and Gen6 (64 GT/s PAM4) where signal integrity margins are exceptionally tight and even minor coupling can degrade performance.

Separation Guidelines:

1. Intra-Pair Spacing (Differential Pair):

- Achieving a 100 Ohm differential impedance requires careful control of the differential pair geometry. This is accomplished by selecting the proper trace width and defining the entire inter-pair spacing, measured from the centerline of one trace to the centerline of the other.
- Use field solvers or impedance calculators to adjust this based on stackup and dielectric properties.
- Ensure tight coupling to prevent skew and minimize common-mode noise.
- Precise length matching of differential pairs is critical, as excessive skew can result in signal reflections and degrade signal integrity. To avoid these issues, the differential pain lengths should be matched within a tolerance of ±2mils.

Escape from BGA or connector pins introduces skew



Adjust Intra-pair skew in differential pair refers to the timing difference between the positive and negetive signals of a single differential pair

2. Inter-Pair Spacing (Adjacent Lanes):

- Maintain at least 3x trace width between adjacent differential pairs (≥15-20 mils typical).
- For Gen6, increase this to ≥4x where feasible to avoid near-end crosstalk (NEXT) due to PAM4's tighter eye margins.

3. Critical Signals (e.g., REFCLK, PERST#, SMBus):

- Maintain ≥5x trace width separation from high-speed aggressors.
- Sensitive control signals can pick up noise that triggers false resets, clock modulation, or margin loss.

4. Avoid Spacing Disruptions During Breakout:

- Do not fan out differential pairs asymmetrically.
- Avoid abrupt changes in trace spacing during routing transitions or layer changes, as they cause impedance mismatches and mode conversion.

Why It Matters:

Inconsistent or insufficient spacing increases mutual capacitance and inductive coupling, leading to:

- Crosstalk between lanes
- Skew between pairs
- Mode conversion (differential to common)
- Eye closure, especially in PAM4 systems like PCIe Gen6

Uniform spacing maintains signal fidelity across the channel and reduces susceptibility to noise and reflections. Even well-matched pairs can degrade if they experience local asymmetry.

Engineering Tip:

Use stackup-aware tools (e.g., Cadence Allegro X) to determine the ideal width/spacing combination for your board materials. Once validated, lock these values into your PCB layout constraints (e.g., Allegro X Constraint Manager) to avoid accidental violations during fanout, tuning, or autorouting.

For Gen6, simulate crosstalk aggressor-victim coupling using IBIS-AMI crosstalk models to confirm that eye closure does not exceed FEC recovery margins.

4. What are best practices for breakout routing in BGA packages?

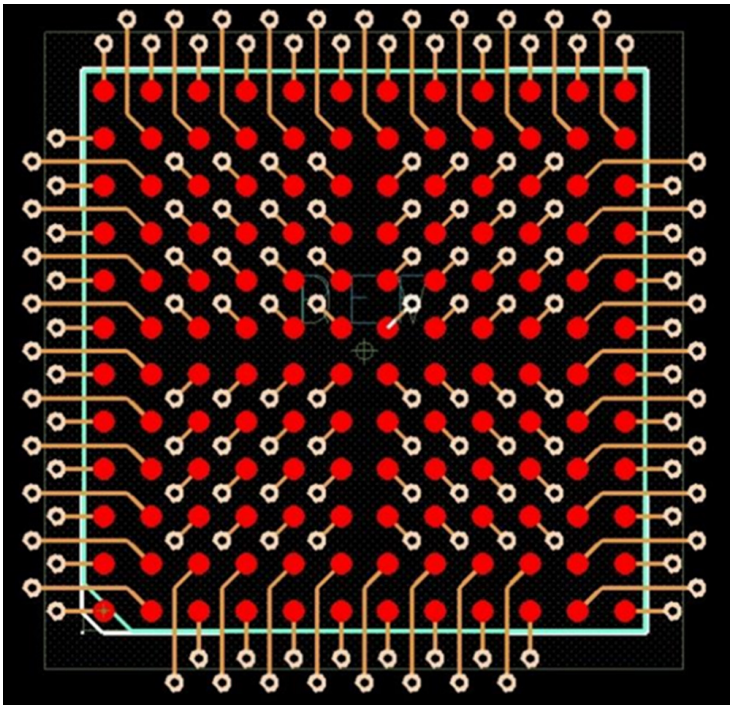
The BGA breakout region (where signals transition from BGA pads to inner-layer traces) is one of the most constrained, sensitive, and failure prone areas in high-speed PCB design. Especially in dense PCIe interfaces (Gen4 through Gen6), poorly designed breakouts can dominate return loss, cause differential skew, and significantly degrade signal integrity before the trace has even left the component footprint.

Challenges in BGA breakout design:

- Stub formation from via-in-pad or through-hole via transitions
- Impedance discontinuities caused by pad diameter, via size, fanout topology, and copper plane interruptions
- Asymmetric breakout paths, which cause skew or mode conversion in differential pairs
- Inconsistent reference planes, especially when escaping to stripline or different routing layers

Key Breakout Patterns and When to Use Them:

- Via-in-pad (VIP):
  - Best used when routing space is extremely limited (e.g., fine-pitch BGAs < 0.8 mm). It enables direct breakout without trace stubs but requires filled and plated vias to prevent solder wicking and impedance mismatch. Ideal for PCIe Gen5/Gen6 breakouts where stub length must be minimal.
- Dogbone fanout:
  - The most traditional breakout method, where a short trace exits the BGA pad and connects to a via offset from the pad. Easier to fabricate, but may introduce stub length unless backdrilled. Suitable for Gen3/Gen4 or where via-to-trace transitions can be carefully controlled.
- Microvia and stacked microvia escape:
  - Common in HDI boards. Staggered or stacked microvias allow high-density escape routing without the signal stubs associated with plated through-hole vias. Essential for ultra-high pin count PCIe retimer or switch packages in Gen6 designs.
- Channel breakout (pair-preserving):
  - Ensures both signals of a differential pair exit the BGA with identical path length and geometry. Use when signal symmetry is critical to maintaining mode balance, such as in Gen5/Gen6 PAM4 differential channels.



Example of a simple BGA breakout in Cadence Allegro X

Why It Matters:

The launch zone (where a signal transitions from the BGA pad to the routed trace) is often the highest point of impedance discontinuity in the entire channel. If not properly optimized, this region can cause return loss exceeding 10 dB, collapse the eye diagram, and invalidate equalization training at the receiver. In Gen6, where vertical eye openings can be less than 20 mV, launch discontinuities alone can cause the channel to fail even if the rest of the routing is clean. Ensuring that breakout regions preserve impedance, minimize stub resonance, and maintain symmetry is essential for channel margin and compliance.

Engineering Tip:

- For best performance, model BGA breakout regions using 3D full-wave EM solvers like Cadence Clarity to capture parasitics from pad shapes, via transitions, and reference plane interactions.
- Use VIP or backdrilled dogbone breakouts where possible to eliminate stubs.
- Always maintain symmetrical breakout between the positive and negative sides of differential pairs and avoid routing only one trace of a pair around an obstacle.
- Use 45° angles instead of 90° bends to reduce current crowding and minimize capacitance.
- For PCIe Gen6, favor HDI stackups with microvia routing to escape dense BGA retimers or switch ASICs with minimal discontinuity and better fabrication yield.

5. PCIe lanes, and how is it calculated?

The target differential impedance for PCI Express lanes is 100 Ω ±10%, a requirement established by the PCI-SIG to ensure consistent signal behavior across different systems and vendors. Differential impedance refers to the opposition a differential pair presents to alternating current, and it's a function of the geometry and material properties of the PCB trace environment.

It depends on:

- Trace width (W): Wider traces reduce impedance.
- Trace spacing (S): Closer spacing lowers differential impedance.

- Dielectric height (H): Distance from trace to reference plane; greater height increases impedance.
- Dielectric constant (Dk): Affects signal velocity and impedance.
- Geometry type: Stripline (buried between planes) offers tighter impedance control than microstrip (top-layer over a single plane).

While simplified formulas exist: $Zdiff \approx 60\sqrt{(Er\_eff)} * \ln(4H / (0.67(W+S)))$

Field solvers are required for accuracy. These tools account for fringing fields, copper roughness, and asymmetry in real-world stackups.

Why It Matters:

Impedance mismatches cause signal reflections, reducing signal quality and increasing bit error rates. This effect becomes especially critical at Gen4/Gen5/Gen6 speeds (16-64 GT/s), where the eye diagram is narrow, and even small discontinuities can cause link instability or failure.

Engineering Tip:

Always specify TDR test coupons on every panel and verify real-world impedance before assembly. Use 2D or 3D field solvers (e.g., Cadence Sigrity, Cadence Clarity) to model impedance accurately and align the design with fab capabilities.

6. What routing topologies are recommended for PCIe lanes?

PCIe is a strictly point-to-point interface, where each transmitter must be directly connected to exactly one receiver. Any deviation from this architecture introduces discontinuities that degrade performance and can cause link training failures, particularly in high-speed PCIe Gen4, Gen5, and Gen6 systems. The routing topology must reflect the logical connection model and maintain signal integrity across the full channel.

Preferred Topology:

- Direct, point-to-point differential pair routing with no stubs or branches
- Matched trace lengths and symmetric geometry for each lane
- Smooth breakout paths and minimal via transitions to reduce discontinuity

Topologies to Avoid:

- T-branches or stubs: Create unmatched loads and generate strong reflections
- Star topologies: Violate PCIe point-to-point behavior; cause uncontrolled impedance mismatches
- Mismatched routing styles or trace lengths between lanes: Introduce skew and modal imbalance, leading to degraded eye diagrams or failed link negotiation

Why It Matters:

At PCIe Gen4 and Gen5 speeds, routing discontinuities already pose a risk of degrading the channel's insertion loss and return loss performance. But at Gen6 (64 GT/s, PAM4 signaling), the tolerance for topology induced artifacts becomes razor thin. Any form of stub, fan-out asymmetry, or path imbalance can introduce branch reflections and timing skew that adaptive equalization cannot recover from. Even a small mismatch in routing style, such as mixing stripline and microstrip on different lanes, can result in eye closure, common-mode radiation, or equalization failure, rendering the link nonfunctional.

Engineering Tip:

- Always keep PCIe lanes fully symmetric in both routing and stackup.
- Maintain tight coupling and identical topology between the positive and negative traces of each differential pair and avoid crossing plane splits or changing reference planes unnecessarily.
- Prefer stripline routing over microstrip whenever possible, as it offers superior EMI shielding and better impedance control, especially critical for Gen6.

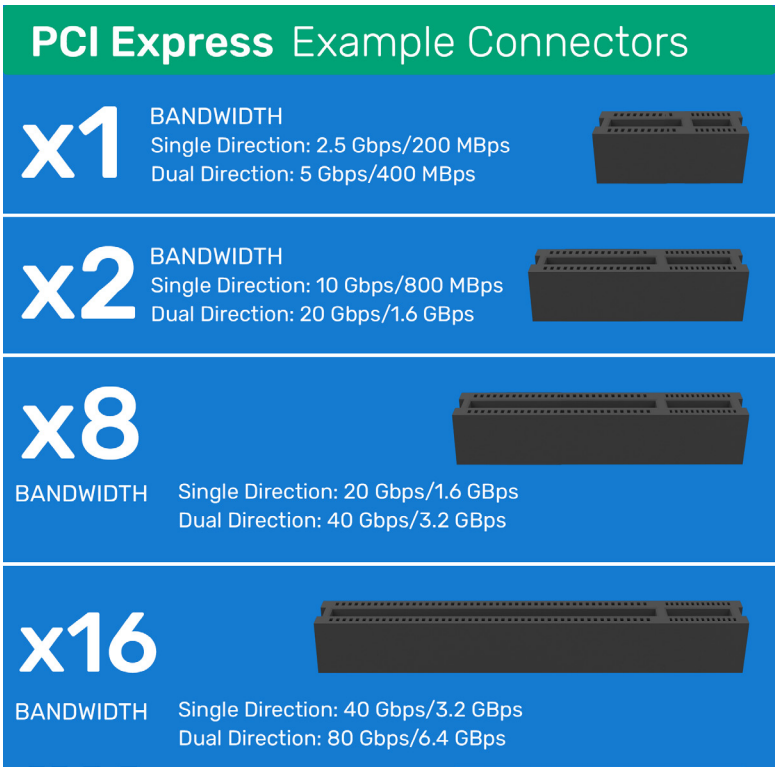
- If bifurcation or switch routing is used, document lane assignments carefully, and simulate to ensure impedance and timing consistency are preserved across all switched paths.

7. How are connectors and card-edge interfaces designed to maintain signal integrity in PCIe?

Connectors and edge fingers present a significant challenge in high-speed PCIe design because they often create discontinuities in impedance and add stub-like structures if not carefully modeled and tuned.

Sources of signal degradation include:

- Pad-to-trace transitions: Sudden changes in geometry cause reflection points.
- Breakout regions: Inconsistent impedance during signal escape from BGA to connector pads.
- Stub under pads: If through-hole vias are not backdrilled, unused barrel sections resonate at PCIe frequencies.
- Pin escape topology: Non-uniform routing from connector pins to PCB traces introduces skew and impedance shifts.



Standard PCIe connectors - x1, x4, x8, and x16 slots for graphics cards, high-speed Ethernet cards, RAID cards and other devices that require high-speed data transfers

PCIe rated connectors (e.g., Samtec Q-Series, Molex EdgeLine) are designed to support 85-100 Ω differential impedance, but the layout around them must also maintain this impedance consistently through careful breakout region design.

Why It Matters:

At Gen4 and Gen5 speeds, connector transitions are often the dominant contributor to return loss and insertion loss. But at PCIe Gen6 frequencies (up to 32 GHz), even minor mismatches or stub resonances in the connector region can cause severe eye diagram closure, especially near the receiver, where equalization headroom is limited. Poor connector design can reduce signal margin to the point where the channel fails to train or exhibits excessive bit error rates under stress. This is particularly problematic in Gen6 PAM4 systems, where vertical eye openings are extremely narrow and any asymmetry or reflection can overwhelm the receiver.

Engineering Tip:

Model the entire connector region, including via transitions, anti-pads, and pad launches, in a 3D EM tool (e.g., Cadence Clarity). Validate targets by PCIe Generation:

- Gen4 (16 GT/s): validate S11 (return loss) and S21 (insertion loss) up to16 GHz
- Gen5 (32 GT/s): validate S-parameters up to 32 GHz
- Gen6 (64 GT/s, PAM4): Validate up to 32 GHz, but also evaluate signal integrity through eye diagrams and COM (Channel Operating Margin) due to PAM4 sensitivity

Modeling and Layout Guidelines:

- Use manufacturer provided S-parameters or 3D models for the connector itself whenever available to ensure accurate frequency dependent behavior.
- Model connector footprints, vias, and launch geometry together in 3D to capture localized impedance mismatches.
- Backdrill or optimize stub lengths beneath connector pads:
  - Aim for <10 mils stub length for Gen4
  - <7 mils for Gen5
  - ≤5 mils or ideally no stub for Gen6
- Minimize return path discontinuities (e.g., anti-pad shapes, reference plane interruptions) to preserve return loss and reduced mode conversion
- For Gen6, carefully validate differential to common mode conversion (Sdd21 to Scd21), as PAM4 is especially vulnerable to mode conversion and reflection induced jitter

8. How does stackup design affect PCIe performance?

Stackup design defines the electrical environment surrounding PCIe signals and is one of the most critical factors for signal integrity. It affects:

- Impedance control: Determined by trace geometry, dielectric height, and Dk
- Return path stability: Solid reference planes below traces reduce noise and radiation
- Crosstalk isolation: Strategic layer separation reduces coupling between signals
- Layer symmetry: Minimizes warpage and maintains trace balance

Common PCIe stackup strategy:

- Stripline routing for critical signals (buried between two planes)
- Microstrip reserved for non-critical or low-speed signals
- Alternating routing layers to manage congestion and preserve orthogonality

Why It Matters:

A poorly designed stackup results in impedance mismatches, skew, and uncontrolled return paths, all of which degrade eye quality. At Gen5/Gen6 speeds, even dielectric asymmetry between layers can increase jitter.

Engineering Tip:

Use a symmetrical stackup with alternating ground and signal layers. Avoid sharing return planes between high-speed and noisy signals. Specify exact laminate materials (e.g., Megtron 6, Tachyon 100G) and validate impedance with a field solver before layout.

9. What design rules apply to 8-layer stackups for Gen6 routing and isolation?

Designing an 8-layer PCB for PCIe Gen6 requires careful layer planning to maintain impedance control, minimize crosstalk, and provide continuous return paths. At 64 GT/s PAM4, signal integrity is highly sensitive to dielectric quality, inter-layer coupling, and power plane noise.

A typical Gen6-compatible 8-layer stackup follows this pattern:

- L1: Top layer - components and routing
- L2: Solid ground (GND) plane
- L3: Gen6 stripline signals
- L4: Solid ground or power (with isolation)
- L5: Solid ground
- L6: Gen6 stripline signals
- L7: Ground or low-noise power (with stitching caps)
- L8: Bottom layer - components and routing

This layout ensures that all high-speed lanes are routed as buried stripline between solid ground layers, maintaining consistent impedance (~100 Ω differential) and low EMI. Routing is orthogonal between adjacent layers to reduce broadside coupling.

Key design constraints:

- One solid reference plane per signal layer, no splits or voids
- Dielectric thickness control to maintain target Zdiff across entire channel
- Use low-loss, low-Df laminates (e.g., Tachyon 100G, Megtron 7)
- Maintain ≥3× spacing between Gen6 signals and adjacent aggressors or power nets

Why It Matters:

Gen6 PAM4 margins are extremely tight, eye openings are ~30-40 mV, and crosstalk can collapse multiple eyes at once. If ground planes are shared, discontinuous, or too far from signal layers, the return current path becomes inductive, leading to impedance ripple and mode conversion.

Moreover, power plane proximity couples supply switching noise into sensitive lanes unless decoupled properly. As stackups become denser (e.g., when routing multiple x16 slots), routing over compromised reference planes without shielding becomes a leading cause of failed COM or random BER errors in field testing.

Engineering Tip:

- Always place Gen6 signal layers between two solid ground planes for stripline routing.
- Use tight dielectric control (±5%) and confirm fab stackup matches the model.
- Avoid routing Gen6 lanes on top or bottom layers; surface microstrip is more sensitive to EMI, etch width variability, and glass weave skew.
- Route clock signals and noisy buses at least two layers away vertically from Gen6 lanes and avoid parallelism.
- Simulate stackup in a 2D field solver (e.g., Cadence Sigrity) and check for crosstalk, skew, and impedance uniformity.

10. What is reference plane stitching and why is it important?

Reference plane stitching refers to the use of ground vias placed near signal vias during layer transitions to maintain a continuous low impedance return path.

Return current for high-speed signals follows the lowest impedance path, which is typically the nearest reference plane. If a signal via changes layers without a nearby stitching via, the return current must detour - often via long, high impedance loops, leading to:

- Common mode conversion
- EMI radiation
- Signal distortion
- Return path resonance

Where Stitching Is Needed:

- Any layer change (e.g., L3 L5)
- Routing over power/ground boundaries
- Plane splits, slots, or cutouts

Why It Matters:

High-speed differential signals rely on tight electromagnetic coupling between forward and return paths. Interruptions force the signal to radiate, become asymmetric, and induce jitter or bit errors.

Engineering Tip:

Place at least 2 ground stitching vias within 100-200 mils of each signal via. For differential pairs, place vias on both sides to maintain symmetry. Include these in pre-layout DRCs and validate return path continuity in stackup planning.

11. How are backplane or multi-board PCIe channels handled?

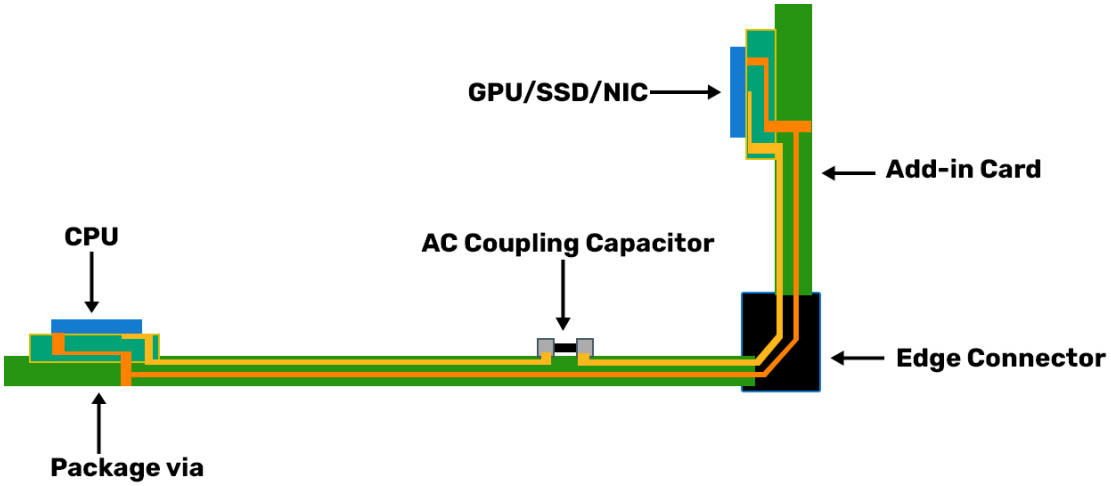
Backplanes and multi-board systems (such as risers, midplanes, or pluggable modules) introduce extra channel segments composed of multiple connectors, vias, layer transitions, and board materials. Each segment contributes to total insertion loss, impedance mismatch, and timing skew, making signal integrity much harder to control than in a single board design.

Challenges in multi-board PCIe channels include:

- Cascaded insertion loss accumulation from board traces, vias, and connectors
- Connector reflections from poor pad transitions or mismatch
- Dielectric mismatch between boards (e.g., low loss Megtron modules vs. FR4 risers)
- Skew or mode imbalance from routing asymmetry across connectors or cables

Design Practices:

- Keep total channel insertion loss (IL) below:
  - ≤28 dB (Gen4)
  - ≤36 dB (Gen5)
  - For Gen6, typical limits are ≤32 dB at 32 GHz due to PAM4 eye compression
- Use retimers at key intervals to re-establish signal strength and reset jitter
- Choose low-loss PCB materials and impedance matched high-speed connectors
- Simulate the full end-to-end system using stitched S-parameters from each segment, not just individual boards in isolation.



Example of PCIe back plane and edge connector

Why It Matters:

In multi-board systems, the connector region or mid-board interface often becomes the dominant source of insertion loss and return loss, even if the board routing itself is clean. For PCIe Gen5, maintaining link stability is difficult beyond 12–16 inches of FR4-equivalent channel unless retimers are added to reset jitter and restore signal integrity. At Gen6 speeds (64 GT/s), the tolerance is even lower: every connector and segment must be co-optimized as part of a tightly managed system level loss budget. Without proper modeling and equalization planning, Gen6 multi-board links will fail to train or exhibit unacceptable BER.

Engineering Tip:

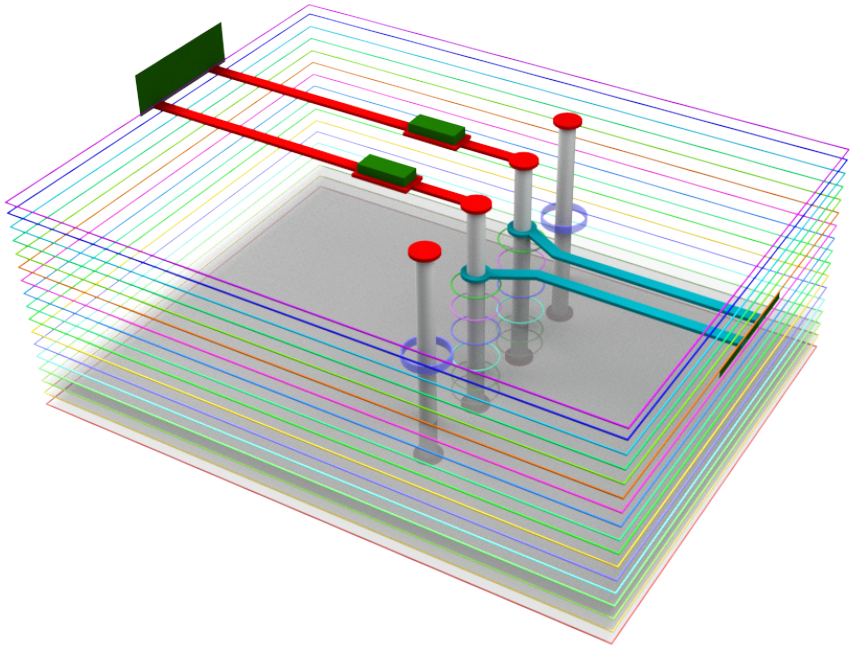
- Include insertion loss test coupons on every board in a multi-board PCIe platform to validate stackup loss in manufacturing.
- When designing modular systems, use measured S-parameters to stitch together an accurate end-to-end model of the full channel, including cables, connectors, and riser boards.
- Retimers should be used strategically in long or lossy segments to reset equalization and recover margin.
- Avoid using redrivers in PCIe Gen5 and Gen6 systems, they do not recover jitter or correct timing skew and may make debugging more difficult in complex chains.

12. Where should AC coupling capacitors be placed?

AC coupling capacitors are required in PCIe designs to block DC bias between the transmitter and receiver. While their function is simple, their physical placement has a major impact on signal integrity - particularly at high data rates where even small discontinuities act as impedance mismatches. If not carefully positioned, capacitors can behave like lumped stubs, degrade return loss, and introduce reflections that impair equalization or prevent link training.

Placement Guidelines:

- Place capacitors as close to the receiver as possible, especially in Gen5/Gen6 links where the Rx is less tolerant to reflections.
- Keep capacitors on the same layer as the signal traces to avoid unnecessary via transitions.
- Maintain matched stub lengths between the trace and capacitor pad for both sides of the differential pair.
- Use inline or embedded capacitor placement to minimize discontinuity.
- Ensure identical orientation and geometry for each cap to preserve symmetry and differential balance.



Simulation of AC coupling caps

Why It Matters:

At high frequencies (particularly at 16 GT/s (Gen4), 32 GT/s (Gen5), and 64 GT/s (Gen6)) even minor impedance mismatches caused by misaligned or asymmetric capacitors can produce reflections strong enough to collapse the eye diagram or increase bit error rates. Just 20 mils of stub length between the trace and a capacitor pad can reflect high frequency energy back into the channel, appearing as ISI or common-mode noise. Poor capacitor placement can also shift the channel's return loss profile, pushing it out of compliance even if the rest of the routing is pristine.

Engineering Tip:

When routing space is limited, consider using embedded capacitors (e.g., embedded passives or inner-layer buried caps) to eliminate stubs entirely. If discrete capacitors must be used, place them symmetrically, avoid sharp bends, and locate them within 100 mils of the receiver to ensure optimal return path and minimal latency. Never rotate one capacitor 90° relative to its pair, as this introduces field asymmetry and differential-mode imbalance. For Gen6 designs, validate capacitor placement using 3D EM simulation to confirm minimal impact on S11 and mode conversion.

13. What layout priorities are critical for PCIe Gen6?

Gen6 uses PAM4 signaling and operates at 64 GT/s, making layout constraints tighter than ever. Key priorities include:

1. Low insertion loss: Use ultra-low-loss laminates (e.g., Megtron 7, Tachyon 100G)
2. Tight skew control: Match lengths within <0.5 ps between differential pairs
3. Eliminate stubs: Mandatory backdrilling or VIPPO (via-in-pad plated over)
4. Strict impedance control: Target ±5% across entire channel
5. PAM4-specific symmetry: Layout must preserve equal rise/fall symmetry and common-mode balance
6. Isolation from aggressors: Keep away from clocks, switchers, and power vias

Why It Matters:

With PAM4, each eye has only 1/3 the vertical margin of NRZ. Even minor discontinuities collapse the signal. Equalization and FEC can't compensate for fundamental layout errors.

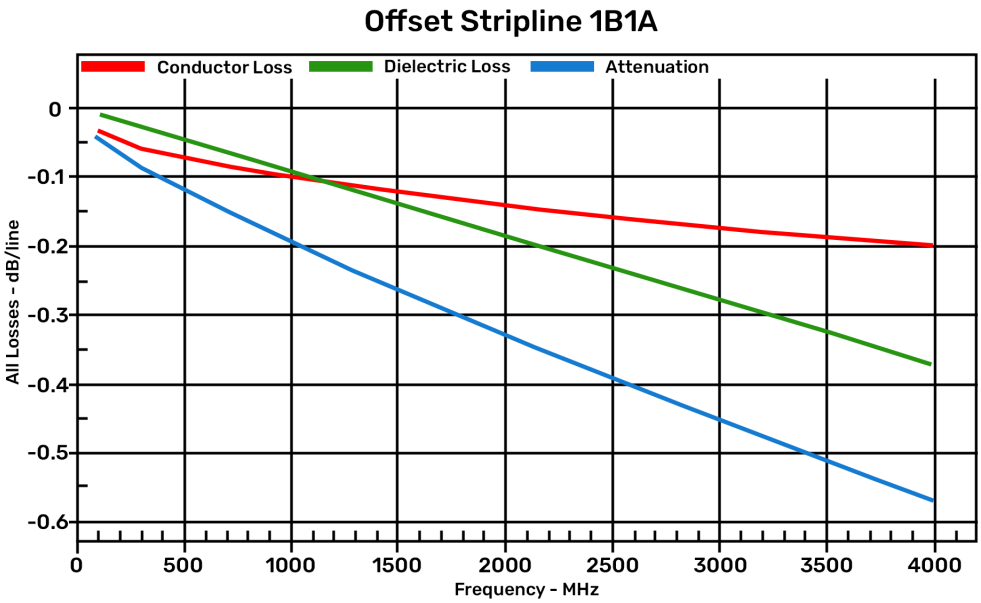
Engineering Tip:

Use tight routing constraints (≤10 mil skew, ≤1 mil mismatch). Keep trace widths and spacing uniform, especially in breakout and connector regions. Run full Gen6 COM simulations with measured S-parameters pre-fab.

Loss & Materials

14. What is insertion loss, and what mechanisms cause signal attenuation in PCIe?

Insertion loss (IL) is the reduction in signal amplitude as it travels through a PCB trace, connector, or via. It is measured in decibels (dB) and increases with both frequency and channel length, making it a critical factor in high-speed PCIe links - especially from Gen4 onward. As bandwidth demands increase, insertion loss becomes the dominant constraint in determining whether a channel can meet eye margin and BER targets.



Conductor and dielectric loss graph

Four key physical mechanisms contribute to insertion loss (IL):

- Dielectric loss: Caused by energy dissipation in the PCB substrate, directly proportional to the dielectric loss tangent (Df) and signal frequency.
- Skin effect: At high frequencies, current flows only on the outer surface of the conductor, increasing effective resistance as frequency rises.
- Surface roughness: Scattering at the conductor's uneven surface increases resistance and disrupts high-frequency current flow.
- Discontinuities: Vias, pads, and connector transitions introduce frequency-dependent reflections and absorption, adding localized IL and degrading return loss.

Typical Insertion Loss Budgets (Differential, at Nyquist Frequency):

Gen	Nyquist Freq	Max IL
3	4 GHz	<20 dB
4	8 GHz	<28 dB
5	16 GHz	<36 dB
6	32 GHz	<32 dB (PAM4, eye height limited)

Note: Gen6 uses PAM4 encoding, which makes the eye height far smaller than in NRZ systems. This lowers the effective tolerance for insertion loss (IL) even though signaling rate is higher.

Why It Matters:

At high frequencies, insertion loss increases nonlinearly and rapidly consumes the available signal margin. Even 12 inches of standard FR4 can exceed the Gen5 loss budget, especially when combined with connector or via transitions. For Gen6, where the signal operates at 64 GT/s, the tolerable IL drops despite the faster rate because PAM4 signaling relies on narrow eye openings, which is often just 30-40 mV per level. If loss is too high, the receiver's equalizer cannot fully restore the waveform, leading to eye closure, ISI, and training failure. Equalization alone cannot fix poor physical channels.

Engineering Tip:

Always use low-loss laminates such as Tachyon 100G, Megtron 7, or equivalent for long PCIe runs. Simulate the full interconnect using post-layout extracted S-parameters and validate against the full loss budget. In modular systems or long backplanes, retimers are strongly preferred over redrivers, retimers re-clock and re-equalize the signal, whereas redrivers merely amplify existing noise and do not correct timing jitter. Retimers can restore signal integrity and extend PCIe reach, especially for Gen5/Gen6.

15. What PCB materials are suitable for PCIe Gen4, Gen5, and Gen6?

As data rates increase, PCB material selection becomes critical to signal integrity. Loss tangent (Df) and dielectric constant (Dk) govern how signals degrade as they propagate.

- Standard FR4 (Df ~0.020) is only suitable for Gen1/Gen2 or very short Gen3 traces.
- Gen3/Gen4/Gen5/Gen6 require low-loss laminates, such as:
  - Gen3: FR408HR, I-Speed (Df < 0.010)
  - Gen4: Megtron 6, Tachyon 100G (Df < 0.005)
  - Gen5: Megtron 6N, Tachyon 100G, Megtron 7 (Df < 0.002)
  - Gen6: Megatron 8, Panasonic R-5785(N) or R-5795, Isola I-Speed Plus, Tachyon 110G, Panasonic Megatron GX (Df ≤ 0.0015 or better)

Key Material Properties:

- Low Df: Reduces dielectric attenuation.
- Stable Dk: Ensures consistent impedance and propagation delay, minimizing reflections and skew.
- Thermal stability: Critical for maintaining performance during reflow cycles and long-term heating.
- Surface roughness compatibility: Important for Gen6+; use HVLP or VLP copper to reduce conductor loss at 32+ GHz.
- Loss modeling accuracy: At Gen6 speeds, ensure 3D field solvers account for anisotropic Dk/Df behavior and resin/glass composition.

Why It Matters:

Gen5 at 32 GT/s and Gen6 at 64 GT/s offer extremely tight signal margins. At these speeds, insertion loss, dielectric loss, and even minor impedance discontinuities can severely degrade eye openings and equalization effectiveness. Materials with high loss (Df) or inconsistent dielectric properties (Dk) can absorb or distort high-frequency energy, leading to inter-symbol interference (ISI), timing skew, or even modal conversion.

At Gen6, channel loss budgets are tighter than ever (typically <32 dB at 32 GHz) with no room for uncontrolled material variation, surface roughness, or anisotropic resin/glass effects. Even slight variations in material weave, copper profile, or prepreg flow can disrupt channel compliance.

Engineering Tip:

Use symmetrical stackups to prevent imbalance-induced mode conversion. Simulate material performance across the channel and specify actual laminate part numbers (not just generic FR4) in fab drawings.

16. How does copper roughness affect insertion loss at Gen6 speeds?

Copper roughness increases insertion loss by elevating the effective surface resistance of traces at high frequencies due to the skin effect, which confines current to the outermost microns of the conductor. As signal frequency increases, current flows through a smaller cross-sectional area. Rough copper surfaces increase the path length of current flow and introduce microscopic scattering losses. At 32 GHz (Nyquist for PCIe Gen6), this effect is severe, especially over long trace lengths or in vias where surface area dominates.

This loss is not linear. At Gen6 bandwidths, roughness can increase total insertion loss by 10–30% compared to flat foil, depending on trace length and stackup geometry. Without accounting for it in modeling, you risk underestimating attenuation and overestimating eye height in IBIS-AMI simulations or COM calculations.

Why It Matters:

Gen6 PAM4 signaling offers only ~30-40 mV of vertical eye margin per eye. Even small amounts of unexpected insertion loss, from copper foil selection alone, can cause the signal to fall below receiver detection thresholds. This results in:

- Failed COM compliance (margin < 0 dB)
- FEC uncorrectable symbol errors
- Eye diagram collapse in lab validation

Standard copper roughness used in FR4 (e.g., Rz = 5-8 µm) is unacceptable for Gen6 signal layers. Failing to specify copper roughness in fab drawings can make a compliant simulation fail in hardware.

Engineering Tip:

- Use HVLP (Hyper Very Low Profile) or reverse-treated copper with Rz < 1.5 µm on all Gen6 signal layers.
- Specify foil type and roughness class (e.g., VLP2) in the fabrication notes.
- Simulate insertion loss using a roughness-aware model (e.g., Huray or Hammerstad) in your field solver or channel simulator.
- For backdrilled vias, consider roughness of the barrel and plating layer in via IL modeling.

17. How do you manage material transitions between different boards (e.g., FR4 low-loss)?

In PCIe systems using modular board architectures (e.g., baseboard + riser, or add-in card + carrier), it's common for high performance sections (like the baseboard) to use low-loss laminates (e.g., Megtron 6/7, Tachyon 100G) while the mating board (e.g., a low cost riser) uses standard FR4. Each material has different dielectric constant (Dk), loss tangent (Df), and copper surface roughness - which impacts impedance, propagation delay, and insertion loss.

When a Gen6 signal crosses from one PCB to another, typically through a connector, the dielectric mismatch at the boundary can result in:

- Localized reflections (impedance discontinuities)
- Group delay distortion (velocity mismatch)
- Step-like IL degradation (loss mismatch)
- Modal conversion (if asymmetry exists)

The connector interface acts as a transition point, and if not modeled and compensated properly, it becomes a dominant cause of return loss violations and eye distortion, especially at 32-40 GHz.

Why It Matters:

Gen6 PAM4 signaling allocates a total channel loss budget of ~32 dB, with very tight return loss (S11) constraints (≥15 dB from 10 MHz to 32 GHz). Material transitions can create impedance mismatches of 10-15 Ω and introduce multiple ps of skew. This is enough to cause sampling edge misalignment, incorrect equalizer adaptation, or FEC uncorrectable errors.

If the transition is not simulated across the actual material boundary, designs may pass IBIS-AMI or COM using idealized stackups but fail in the real system due to impedance shifts and loss spikes near the interface.

Engineering Tip:

- Simulate the full channel, including connector, via transitions, and stackup properties for both boards. Use vendor 3D models and include copper roughness and surface finish (e.g., ENIG vs. immersion silver).
- Use identical surface finish and foil roughness across both boards when possible to minimize surface discontinuity loss.
- Design for impedance matching at the lower performance board's Dk/Df, not the ideal stackup.
- Backdrill vias on both boards under connector pads to eliminate stub resonance.
- If routing Gen6 lanes on FR4 is unavoidable, keep trace lengths short (<1-1.5") and push EQ complexity to the receiver with properly tuned CTLE and DFE stages.

Equalization & Channel Modeling

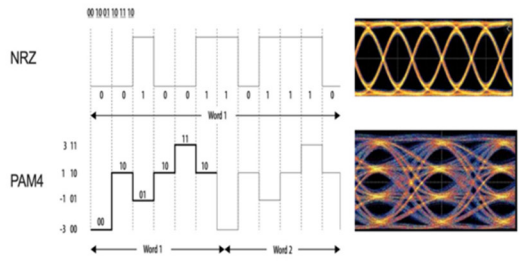
18. How does PCIe receiver equalization work?

PCIe receivers rely on adaptive equalization to compensate for channel impairments such as loss, reflections, and inter-symbol interference (ISI). Equalization occurs both at the transmitter (Tx) and receiver (Rx) and is negotiated during link training (via the LTSSM state machine).

Key equalization components:

- CTLE (Continuous-Time Linear Equalizer): Boosts high-frequency content attenuated by dielectric and skin effect losses.
- DFE (Decision Feedback Equalizer): A digital adaptive filter that cancels post-cursor ISI based on prior bit decisions.
- CDR (Clock/Data Recovery): Aligns the sampling clock to the data stream, dynamically tracking jitter and frequency offsets.
- LEQ (Linear Equalization): Pre-emphasis/de-emphasis from the transmitter to counteract channel low-pass behavior.

These components work together during link initialization to select optimal settings, often iteratively adjusting to improve signal margin.



Data Rate	Encoding	Modulation
2.5 GT/s	8b/10b	NRZ
5.0 GT/s	8b/10b	NRZ
8.0 GT/s	128b/130b	NRZ
16.0 GT/s	128b/130b	NRZ
32 GT/s	128b/130b	NRZ
64 GT/s	1b/1b	PAM4

Image shows how PAM4 modulation significantly impacts signal quality with its four differential voltage levels, leading to a higher Bit Error Rate (BER)

Why It Matters:

Without equalization, a receiver cannot distinguish attenuated signals from noise, particularly after the signal has traveled through a lossy dispersive PCB channel. This limitation becomes critical at Gen4 and Gen5 speeds, where the differential eye opening at the receiver may shrink to just 50-70 mV. However, in PCIe Gen 6, which uses PAM4 signaling, the challenge becomes even more severe: each symbol is represented by one of four levels, and the vertical eye openings can be as small as 15-20 mV. At these levels, equalization is not just beneficial, it is essential. Advanced receiver equalizers must compensate for complex impairments such as multi-tap inter-symbol interference, high-frequency loss, mode conversion, and phase noise. Without robust equalization, the link simply cannot train or meet the required Bit Error Rate (BER), making compliance impossible even in physically short or low-loss channels.

Engineering Tip:

Design your physical channel to minimize the burden on receiver equalization. While adaptive CTLE and DFE can recover from a range of impairments, excessive insertion loss, discontinuities, or crosstalk will limit their effectiveness. Avoid abrupt impedance transitions (such as neck downs, mismatched via stubs, or poorly tuned pad/via geometries) that can induce post-cursor ISI and reduce DFE margin. Maintain consistent trace geometry and tightly coupled differential pairs across the full channel. Keep high-frequency components (e.g., connectors, vias, layer transitions) as short and symmetrical as possible to preserve signal symmetry. For Gen6, use ultra-low loss materials and tightly controlled stackups to ensure the receiver does not have to over equalize and introduce noise amplification. A cleaner Physical path always results in a more stable, lower power, and higher margin equalization response.

19. What are PCIe equalization presets and test modes?

PCIe defines a set of standardized transmitter equalization presets, each representing a different combination of pre-cursor and post-cursor emphasis levels. These presets are primarily used during link training and compliance testing to help the receiver evaluate channel quality and select the best setting for a given channel loss profile. The goal is to ensure the transmitter can shape the signal to open the eye at the receiver, even in the presence of high loss or distortion.

Purpose of Presets:

- Evaluate the receiver's adaptive equalization capability across various conditions
- Characterize how a channel responds to different levels of de-emphasis and pre-shoot
- Select the preset that maximizes eye opening at the receiver (Rx)

Presets are indexed numerically (e.g., Preset 0 through Preset 10 for Gen3/Gen4), with each defining a specific de-emphasis profile. In Gen5 and Gen6, equalization becomes even more critical due to tighter eye openings (especially with Gen6's PAM4 signaling), and presets are mapped to multilevel waveform shaping to compensate for more complex loss behaviors.

Why It Matters:

Preset based training allows PCIe devices to optimize signal integrity without manual tuning. During link training, the receiver sweeps through all transmitter presets and evaluates eye height and width for each configuration. This process ensures that the system can adapt dynamically to real-world loss profiles, crosstalk, and discontinuities. In Gen5 and Gen6 designs, this is especially important because high-frequency attenuation and inter-symbol interference (ISI) become dominant channel impairments. Without robust preset evaluation, the receiver may latch onto a suboptimal signal condition, leading to degraded bit error rate (BER), eye closure, or training failure. Preset compliance testing verifies that all standardized presets meet eye margin and BER requirements under worst-case loss conditions, ensuring interoperability across vendor platforms and cable or backplane configurations.

Engineering Tip:

During simulation, model your full PCIe channel using IBIS-AMI models across multiple equalization presets to evaluate sensitivity to loss and jitter. Pay close attention to how eye width and height change for each preset. In lab environments, use the Compliance Test Mode (CTM) to automatically sweep presets at the transmitter while observing the resulting eye diagrams at the receiver. This can be done using an oscilloscope with equalization support or a bit error rate tester (BERT). For Gen6 PAM4 links, verify all four eye levels and check that the selected preset achieves acceptable horizontal and vertical eye openings at all symbol thresholds.

20. What's the difference between CTLE and DFE equalization?

CTLE (Continuous-Time Linear Equalizer) and DFE (Decision Feedback Equalizer) are two complementary equalization mechanisms located at the receiver in PCIe links. Their job is to recover the eye opening after high-speed signals are degraded by loss, reflections, and inter-symbol interference (ISI) as they travel through PCB traces, vias, connectors, and other discontinuities.

CTLE is an analog filter that shapes the frequency response of the receiver front end. It boosts the high-frequency components of the signal that are attenuated by dielectric loss, skin effect, and surface roughness, especially over long traces and low-loss laminates. To maintain a flat channel response, CTLE also attenuates low-frequency content. Once selected during link training, typically via a set of predefined gain curves, CTLE operates in a fixed mode and does not adapt dynamically.

In contrast, DFE is a non-linear, adaptive digital filter that works by using previously decoded bits to cancel ISI caused by overlapping signal transitions. DFE is critical for recovering bits at high data rates, particularly when eye openings are small or severely distorted. It subtracts predictable error contributions based on historical decisions, enabling better timing recovery and cleaner bit boundaries. However, DFE is sensitive to noise and decision errors; once it makes a wrong prediction, that error can propagate forward. It requires a low BER environment to function correctly and converge reliably.

Why It Matters:

At Gen4 and Gen5 speeds, the dominant mechanism that closes the eye is ISI, especially for longer or lossier channels. DFE plays a central role in suppressing these bit overlap distortions after the fixed front-end CTLE has pre-shaped the incoming waveform. Without effective DFE, the receiver cannot distinguish between symbols, especially in the presence of residual jitter or reflections. With Gen6 and PAM4 signaling, the challenge becomes more severe: there are now four eye openings per UI (Unit Interval), and both CTLE and DFE must simultaneously restore signal margin across all three vertical eye thresholds. Subtle timing and amplitude distortions that would be manageable in NRZ signaling now require precise equalization at the receiver. If the CTLE gain is not properly matched to the channel loss profile, the DFE adaptation may fail to converge or may misidentify ISI contributions.

Engineering Tip:

Always use IBIS-AMI models that support both CTLE and DFE blocks and simulate both pre and post equalization eyes to understand where margin is gained or lost. Sweep CTLE gain curves in simulation to identify which setting best aligns with the channel's frequency dependent loss characteristics. Ensure that DFE convergence is stable across worst-case corners of the channel including channel length extremes, via transitions, and voltage ripple. In Gen6 designs, perform eye analysis across all PAM4 levels and verify that DFE decision thresholds adapt successfully under varying insertion loss, jitter, and crosstalk conditions.

21. How is equalization different in PCIe Gen6?

In Gen6, equalization becomes link wide and forward looking due to the use of PAM4 and Flow Control Unit (FLIT) encoding. Gen6 introduces:

- FEC (Forward Error Correction): Recovers bits lost due to PAM4 sensitivity
- DFE/CTLE/LEQ enhancements: Must compensate for lower eye height and higher ISI
- Training sequences (TS1/TS2) over FLITs: Requires full link layer coordination
- COM (Channel Operating Margin) metrics updated to evaluate PAM4 performance

Equalization still happens dynamically during link training, but now it must adapt to PAM4-specific impairments, including:

- Vertical non-linearity
- Multiple eyes
- Common-mode rejection limits

Why It Matters:

Gen6 equalization is more sensitive to layout issues, and traditional eye margin is no longer enough. Designers must ensure the entire link supports FEC decode and resolves all eyes simultaneously.

Engineering Tip:

Use updated COM and IBIS-AMI models that reflect PAM4 signal behavior. Design layout to avoid any asymmetry, including BGA breakout and capacitor pads. Ensure materials support at least 40 GHz insertion loss flatness.

22. What are redrivers and retimers, and how do they differ?

Both redrivers and retimers are used to extend PCIe reach, but they differ significantly in functionality and application:

Feature	Redriver	Retimer
Type	Analog	Digital (includes Clock and Data Recovery, CDR)
Function	Boosts signal, applies equalization	Fully re-clocks, re-times, and re-equalizes the signal
Jitter Removal	Limited (no CDR or clock alignment)	Strong (regenerates both timing and amplitude cleanly)
Latency	Low	Slightly higher (due to CDR and protocol processing)
PCIe Awareness	No protocol awareness (passive device)	Full PCIe protocol compliance (active link training)

- Redrivers apply analog gain and limited equalization to improve eye opening. They do not interpret the data stream.
- Retimers fully recover the embedded clock and regenerate the signal, essentially “retransmitting” a clean waveform.

Placement Guidelines:

- **Redriver:** May be used in short Gen4 channels where insertion loss is borderline (~28 dB), and jitter accumulation is minimal. It should be placed near the point of highest loss. Not recommended for Gen5 or Gen6 unless validated in a controlled, low-jitter environment.
- **Retimer:** Required when insertion loss exceeds 36 dB, or when timing margin is compromised by multiple interconnects. For Gen5 and Gen6, retimers are the preferred method to re-establish channel compliance. It’s best placed at the midpoint of a long channel, at connector boundaries, or between backplane segments.

Why It Matters:

Redrivers amplify both signal and noise. If jitter is already present on the line, as is often the case in PCIe Gen5 and Gen6, it gets passed through or worsened. Retimers, on the other hand, fully regenerate the signal by recovering the clock, re-aligning bit edges, and retransmitting a clean waveform. This makes retimers essential for maintaining signal quality in Gen5 and Gen6 systems, especially over backplanes, risers, or other multi-board paths. In Gen6 PAM4 signaling, where vertical eye openings are only ~30-40 mV, jitter recovery is no longer optional, it's mandatory for reliable operation.

Engineering Tip:

- Always simulate both retimers and redrivers using IBIS-AMI models within your full-channel simulation flow.
- Use post-layout S-parameters to assess where jitter is accumulating and whether a retimer is needed to restore the eye.
- In multi-hop channels (e.g., host riser backplane endpoint), only retimers can ensure eye recovery and proper training.
- For Gen6 designs, avoid redrivers entirely unless the implementation has been qualified through full PAM4 compliance simulation.

23. When should you choose a redriver verse a retimer in PCIe systems?

Redrivers and retimers are used to extend PCIe reach and improve signal quality, but they operate fundamentally differently.

- Redrivers are analog devices that apply passive or semi-adaptive equalization (typically CTLE) and amplify signals without re-timing. They offer low latency and are transparent to protocol layers.
- Retimers are digital devices that fully recover and regenerate the PCIe signal using CDR, equalization (CTLE + DFE), and full Tx re-drive. They re-align the signal to a new clock domain and can break the timing correlation between Tx and Rx.

Selection guidelines:

- Use a redriver if:
  - Channel loss is marginal (e.g., 15-25 dB insertion loss)
  - Minimal latency is required
  - You only need to boost high-frequency content or flatten channel loss
- Use a retimer if:
  - Total channel loss exceeds 28 dB (Gen4/Gen5) or ~36 dB (Gen6)
  - There are multiple discontinuities (connectors, vias, risers)
  - You need to reset jitter and perform full clock/data recovery

Why It Matters:

Choosing the wrong component can result in non-compliance or unnecessary complexity. A redriver won't fix excessive jitter, while a retimer introduces extra power, complexity, and protocol implications (e.g., sideband configuration, LTSSM transparency). Retimers also require software configuration and SMBus/I2C integration in many designs.

Engineering Tip:

- Use retimers in systems with more than two connectors or where layout constraints prevent clean routing
- Simulate both redriver and retimer options using IBIS-AMI models before committing to hardware
- Check PCIe compliance limits: Gen5 Rx eye mask is tighter than Gen4; Gen6 PAM4 is even more sensitive to jitter accumulation
- If adding a retimer, verify it supports the correct PCIe version and lane bifurcation mode for your platform

24. How do you validate retimer operation and configuration in a PCIe link?

Retimers are not passive devices; they actively participate in link training, lane negotiation, and signal recovery. Misconfigurations (e.g., polarity inversion, bifurcation mismatch, or uninitialized SMBus settings) can prevent the PCIe link from initializing correctly or cause silent downtraining.

Validation involves confirming:

- Transparent forwarding of LTSSM states (Detect Polling Config L0)
- Correct lane mapping and polarity alignment
- Equalization adaptation success (Tx preset, Rx EQ training)
- No unexpected latency or framing errors through the retimer path

Best practices include:

- Use loopback modes (internal or external) to validate re-drive integrity
- Log and compare LTSSM transitions at upstream and downstream ends
- Use margining tools or BERTs to capture eye quality before and after retimer
- Verify retimer registers (via SMBus/I2C) for correct link speed, preset, and adaptive EQ lock
- Test at corner case temperature and voltage conditions to ensure stability

Why It Matters:

A retimer that incorrectly handles link training can silently cause Gen6 links to downtrain to Gen5 or Gen4, pass incorrectly in functional testing, or intermittently fail in the field. Without explicit validation, root causes are hard to isolate because the retimer becomes a “black box” in the signal chain.

Engineering Tip:

- Use PCIe analyzers (e.g., Teledyne LeCroy, SerialTek) to monitor LTSSM state changes across the retimer
- Compare eye diagrams at input vs. output using compliance scopes or margining utilities
- Validate BER with and without the retimer enabled
- If sideband configuration is required, integrate register readback checks into system firmware
- Document and test each valid lane bifurcation and width (e.g., x8 2×x4) across the retimer link

25. How should redrivers and retimers be placed?

The effectiveness of redrivers and retimers in PCIe designs depends not just on their selection, but critically on where they are placed within the signal path. Improper placement can degrade eye openings, amplify jitter, or even prevent link training entirely; especially at Gen5 and Gen6 speeds where timing margins are tight and insertion loss budgets are strict.

Redrivers are best positioned immediately after high-loss sections, such as long PCB trace runs or multiple connector transitions. Their role is to boost signal amplitude and apply limited linear equalization to recover degraded edges. However, redrivers do not recover clock or data timing, so they will also amplify any jitter or noise already present on the signal. This makes their placement very sensitive, they must not follow a segment with severe signal distortion, or they risk reinforcing the noise rather than improving margin.

Retimers, on the other hand, should be placed wherever accumulated jitter threatens to exceed the receiver’s tolerance or where link training stability becomes questionable. Retimers contain full Clock and Data Recovery (CDR) circuits and regenerate a clean, protocol-compliant signal. They add latency but are protocol-aware, making them indispensable for long links, multi-board systems, or daisy-chained risers and backplanes. Proper placement ensures that jitter is reset before entering the final segment of the channel, greatly improving link reliability.

Best Practices:

- Limit total IL per segment to <20 dB before placing a redriver
- Place retimers at board boundaries or near backplane interfaces
- Avoid placing redrivers in long cascaded chains - they amplify residual jitter

Why It Matters:

Improper placement of these devices can be worse than not using them at all. A redriver placed after a noisy or highly reflective segment may actually reduce eye height further by amplifying jitter or boosting overshoot. Conversely, a mistimed retimer, such as one inserted just before a minimal-loss segment, may fail to lock or even introduce link training problems due to timing uncertainty. In high-speed PCIe systems (Gen5/Gen6), there is little room for placement error; each segment must be carefully engineered based on insertion loss (IL), jitter accumulation, and system timing budgets.

Engineering Tip:

Always map the full channel budget, including both loss and jitter, before inserting redrivers or retimers. For Gen5 and Gen6 designs, a good rule of thumb is to limit the IL per segment to <20 dB before placing a redriver, and to place retimers at major interface boundaries like backplane connectors or system board transitions. Avoid chaining multiple redrivers across the same link; residual jitter accumulates, and the final eye opening may fall below compliance margins. If unsure, default to retimers. They offer deterministic signal regeneration, are protocol compliant, and are far more robust across system variations.

26. How is margining performed in PCIe Gen4/Gen5/Gen6?

Margining is the process of systematically evaluating how much degradation a PCIe channel can tolerate before link failure occurs. It quantifies the robustness of the signal in the presence of eye closure, jitter, and environmental stress, and is an essential step in validating design compliance beyond the “pass/fail” of a static eye diagram. Margining includes both electrical-level margins (such as eye height, eye width, and jitter tolerance) and protocol-level margins, such as bit error rate (BER) and Forward Error Correction (FEC) headroom in Gen6.

For Gen4 and Gen5 designs, preset sweeping is commonly used, where the transmitter cycles through different combinations of de-emphasis and pre-shoot to test how the equalization affects the receiver’s eye opening. This allows designers to identify which transmitter preset results in the cleanest eye at the receiver. Additionally, receiver adaptation tracking tools can be used to observe how the receiver’s CTLE and DFE filters dynamically adjust to the channel loss profile. These adaptations help quantify how resilient the receiver is to different levels of insertion loss and ISI (inter-symbol interference).

Gen6 introduces more advanced margining mechanisms due to the use of PAM4 signaling and FEC. In this context, FEC threshold analysis is employed to measure symbol error rates over time and compare them against the FEC decoder’s correction limit. Instead of relying solely on eye diagrams, Gen6 receivers can monitor internal symbol error counters to assess whether a channel is drifting close to the decode failure threshold, even if the physical eye appears open.

In all generations, margining may also be conducted using external tools like BERTs (Bit Error Rate Testers) or PHY-embedded lane margining utilities, which provide precise control over amplitude, timing, and stress conditions to observe degradation over process-voltage-temperature (PVT) corners.

Why It Matters:

As PCIe speeds increase, margins shrink dramatically. A link that passes under nominal lab conditions may fail under aging, voltage drift, or temperature extremes. This is especially true at Gen5/Gen6 speeds, where signal quality can degrade quickly due to even minor variations in trace impedance, connector alignment, or equalization mismatch. Margining ensures that the system is not merely functional, but robust under worst case conditions, including production variability and field deployment scenarios.

Engineering Tip:

For Gen4 and Gen5, perform IBIS-AMI margin analysis using multiple transmitter presets and observe receiver DFE/CTLE behavior over time. For Gen6, go beyond eye diagrams, validate that the raw error rate stays below the FEC correction threshold, even under elevated temperatures and voltage extremes. Stress tests should be performed in both simulation and silicon to ensure long-term reliability. If the system fails under stress but passes in a clean testbench, it's not margin compliant.

27. How is Channel Operating Margin (COM) calculated and interpreted for Gen6 compliance?

COM (Channel Operating Margin) is a standardized signal quality metric defined by PCI-SIG to predict whether a PCIe link can operate reliably under worst-case transmitter, receiver, and channel conditions. It is expressed in decibels (dB) and is computed using:

- Vendor-provided IBIS-AMI models (including Tx/Rx equalization, jitter behavior, and FEC)
- Full-channel S-parameter data representing IL, RL, crosstalk, and modal noise
- Defined jitter/noise profiles per PCIe base specification

For Gen6, COM calculations simulate PAM4 signaling behavior, where the link must maintain all three eye openings above a specified threshold after applying CTLE, DFE, and FEC. The simulation includes adaptive equalizer convergence and signal degradation due to reflections, dielectric loss, and impedance mismatches.

The minimum requirement is COM ≥ 0 dB, which means the channel has just enough margin to meet the specified BER target (typically 10^{-6} post-FEC).

Why It Matters:

COM replaces traditional eye mask or IL-only checks for Gen6 because PAM4 signaling requires understanding the cumulative impact of loss, jitter, and equalization behavior. A channel that "looks fine" in an eye diagram may actually fail Gen6 training if:

- The FEC can't converge due to excessive ISI
- Equalizers fail to open all three eyes
- Crosstalk or modal conversion distorts levels asymmetrically

Designers often assume Gen5 layout rules are sufficient, but PAM4 introduces multi-level thresholds that are far more sensitive to loss and asymmetry. COM analysis captures all of this in a single figure of merit and serves as the only compliance-approved simulation method for Gen6 channel evaluation.

Engineering Tip:

Use the PCI-SIG Gen6 COM tool along with vendor-supplied IBIS-AMI models for both Tx and Rx. Your S-parameters should:

- Extend to at least 40 GHz
- Include differential and common-mode data (Sdd, Sdc)
- Preserve causality and passivity (e.g., use passivity enforcement in simulation tools)

Run COM simulations at both typical and worst-case corners (process, voltage, temperature). A robust Gen6 channel design should target COM ≥ +2 dB for margin. Validate that eye height and jitter converge in the simulation, and cross-verify with IBIS-AMI time-domain simulations.

PCIe Design Guide – Content References

1. PCI Express Base Specifications

- PCI Express Base Specification Rev. 6.0
  - Source of definitions for PAM4 signaling, jitter tolerance, equalization methods, and COM metrics.
  - URL: https://members.pcisig.com/specifications/pciexpress
- PCI Express Card Electromechanical Specification (CEM) Rev 5.0 & 6.0
  - Connector layout, return loss, insertion loss, and compliance standards.
  - URL: https://members.pcisig.com/specifications/pciexpress

2. Texas Instruments Application Notes

- TI High-Speed Layout Guidelines for SERDES (SNLA224)
  - Covers skew, trace impedance, and high-frequency SI.
  - URL: https://www.ti.com/lit/an/snla224/snla224.pdf
- IBIS-AMI Modeling in High-Speed Serial Design (SLLA486)
  - URL: https://www.ti.com/lit/an/slla486/slla486.pdf

3. Intel High-Speed PCB Guidelines

- Intel: PCB Guidelines for High-Speed Signals
  - Recommended spacing, trace width, breakout routing, and stackups for PCIe Gen3/Gen4.
  - URL: https://www.intel.com/content/www/us/en/programmable/support/support-resources/design-guidelines/high-speed-board-design.html

4. Samtec and Molex High-Speed Connector Design Guides

- Samtec PCIe Connector Launch Optimization
  - Return loss and insertion loss characterization using S-parameters.
  - URL: https://www.samtec.com/technical-articles/pcie-connectors-launch-design
- Molex EdgeLine Series PCIe Interconnects
  - High-speed connector stackup and via breakout design.
  - URL: https://www.molex.com

5. Industry Educational Content

- Signal Integrity Journal by Eric Bogatin
  - High-speed layout principles, crosstalk, and reference plane continuity.
  - URL: https://www.signalintegrityjournal.com

6. Measurement & Validation Tools

- Tektronix: PCIe Receiver Testing Using BERTs
  - Eye diagram validation and jitter measurements.
  - URL: https://www.tek.com/en/documents/application-note/pcie-receiver-test-methodologies
- Anritsu: Return Loss and Insertion Loss Measurements
  - Using VNAs to validate PCIe high-speed channels.
  - URL: https://www.anritsu.com/en-US/test-measurement