224G Package and PCB Investigations and COM Reference Model

Mike Li, Jenny Xiaohong Jiang, Yee Lun Ong, Jianming Huang, Qian Ding, Lei Hua, Hsinho Wu, Masashi Shimanouchi, Zhiguo Qian, Jiwei Sun, Kemal Aygun, Stas Litski, Itamar Levin

Intel

Mar 16, 2022

OUTLINE

- 224G package design considerations
- Package technology enablement
- 224G package and PCB design practices
- Correlation
- Proposed 224G Package T-Line Reference Model
- Summary

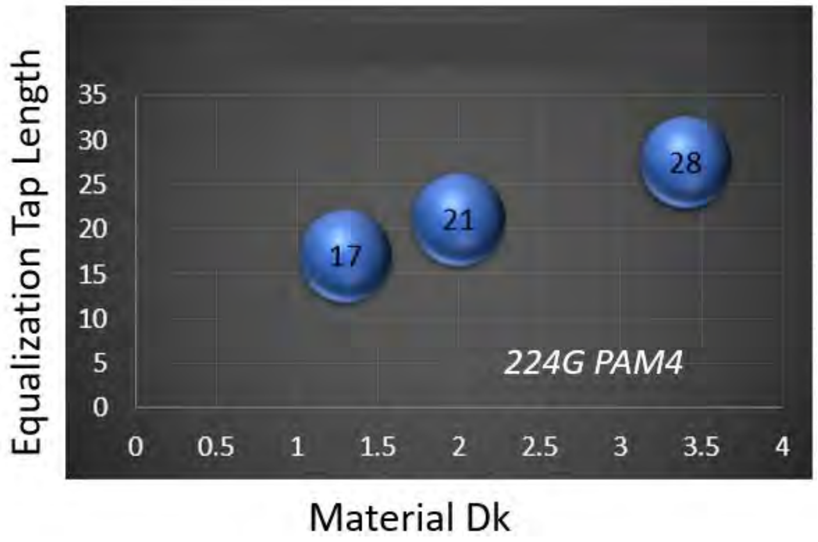
224G Package Design Considerations

- **Higher-order mode propagation and dispersion**
  - Small BGA ball pitch to eliminate the higher-order mode propagation
- **Plane resonance**
  - Adequate ground plane stitching to suppress the plane resonance
- **Transmission loss**
  - Skip-layer trace routing to mitigate the dielectric loss
- **Vertical transition and cross talk**
  - Optimized ball/PTH pattern/voiding and ball size for proper shielding and mitigation of discontinuities
- **PCB breakout adoption**
  - Smart BGA ball pattern for facilitating the board breakout

Package Technology Enablement

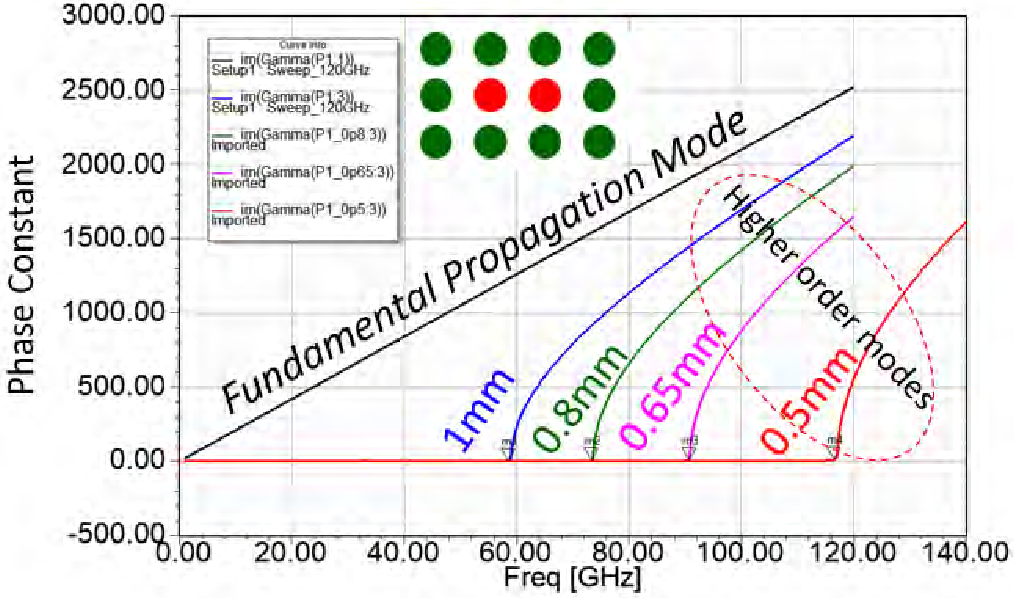
- **Low loss dielectric material**
  - Dielectric loss $\propto f$
- **Advanced copper surface treatment**
  - Metal loss $\propto \sqrt{f}$
- **Skip-layer technology**
  - $\text{Loss} \propto E^2 \; (E = {}^{V}/_{d})$ where d is the distance from signal to ground
- **Lower Dk dielectric material**
  - Less capacitive effects → More relaxed voiding requirement
  - Lower propagation delay ($\propto \sqrt{\varepsilon_r}$) → Less equalization tap length
  - Dk<2.0

0.12 dB/mm @ 56GHz

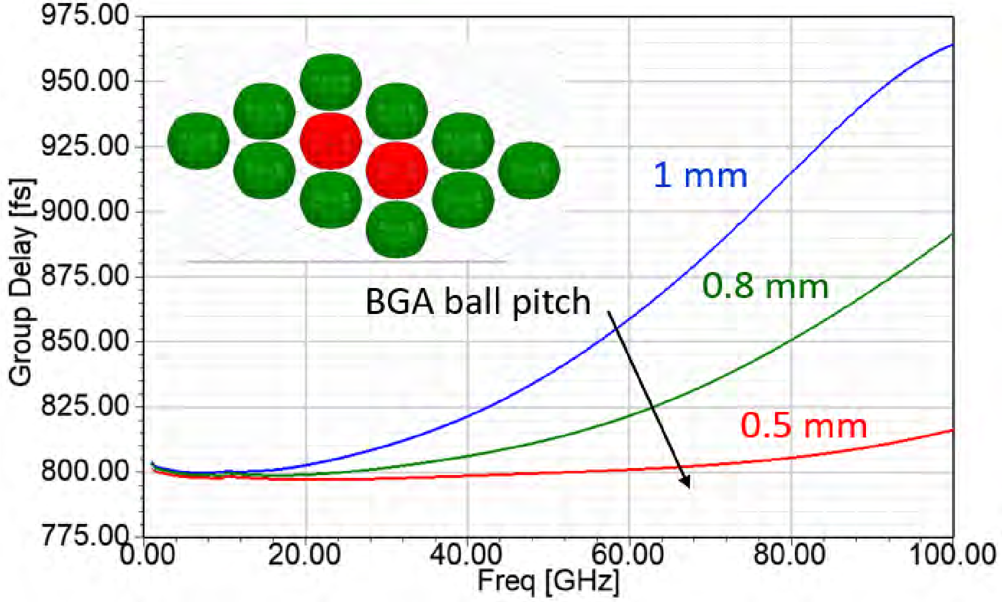


Package Ball Pitch Design: Higher-Order Mode and Dispersion

BGA



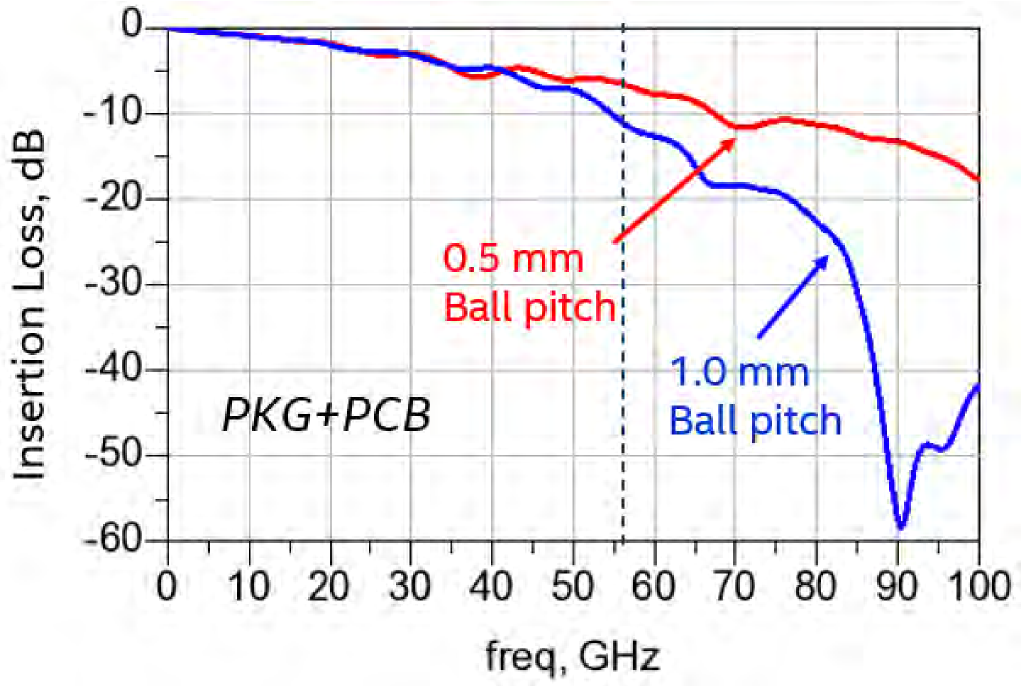
Cutoff frequency vs. BGA ball pitch



Normalized BGA group delay

Takeaway: The smaller the BGA ball pitch, the higher the cutoff frequency of higher-order modes; and the smaller the group delay variation hence the less dispersion (ISI).
1mm ball pitch cutoff frequency is ~ 58GHz, 0.8mm or smaller ball pitch is recommended for 224G PAM4.

Package Ball Pitch Design: Impact on PCB Transition Loss

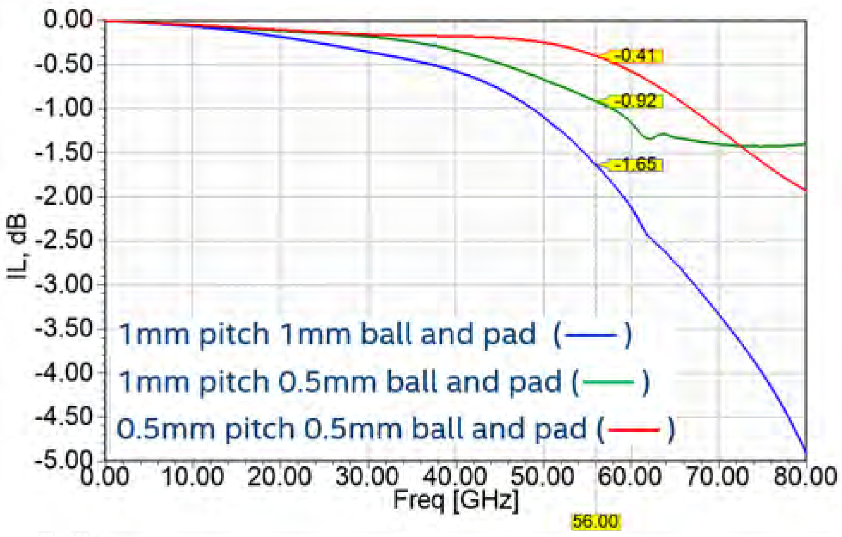


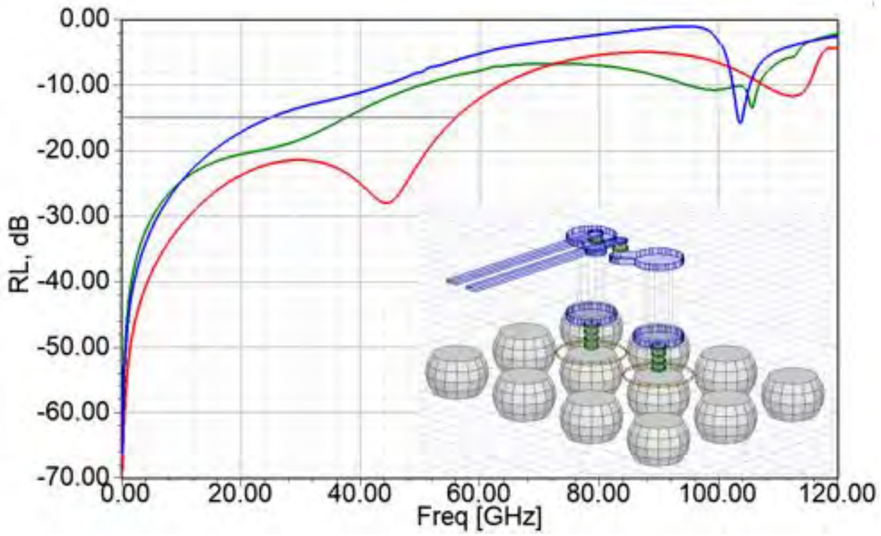
Package+PCB with 1mm vs. 0.5mm BGA ball pitch

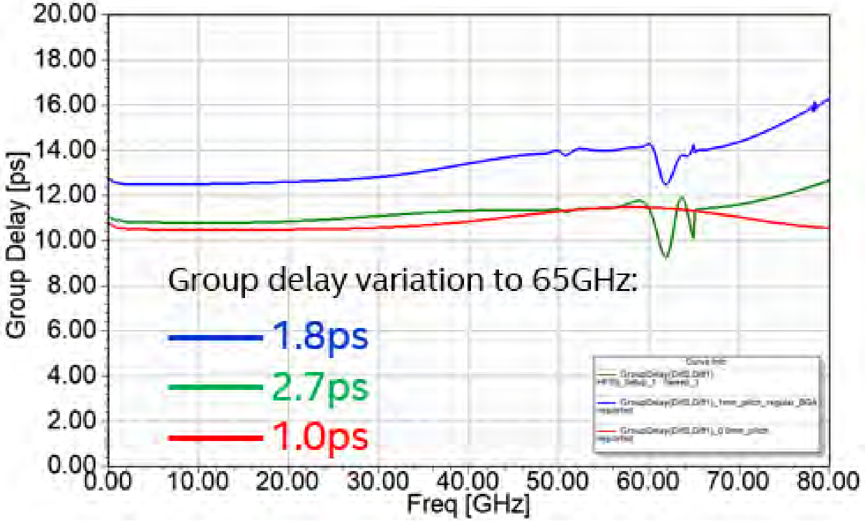
- PCB via-in-pad (VIPPO) pattern/pitch follows package BGA ball pattern/pitch
- More confined via configuration (from small package ball pitch) showed much smaller PCB transition loss (5 dB difference here) when using the same package ball pattern
- A properly selected ball pitch to suppress the higher-order mode propagation is the first step to designing a healthy 224G package

Package Ball Pitch and Ball Size Impact

- **Ball pitch – dispersion and loss**
- **Ball size – discontinuities and loss**
- **Comparison of**
  - o 1mm ball pitch + 1mm ball/pad size
  - o 1mm ball pitch + 0.5mm ball/pad size
  - o 0.5mm ball pitch + 0.5mm ball/pad size

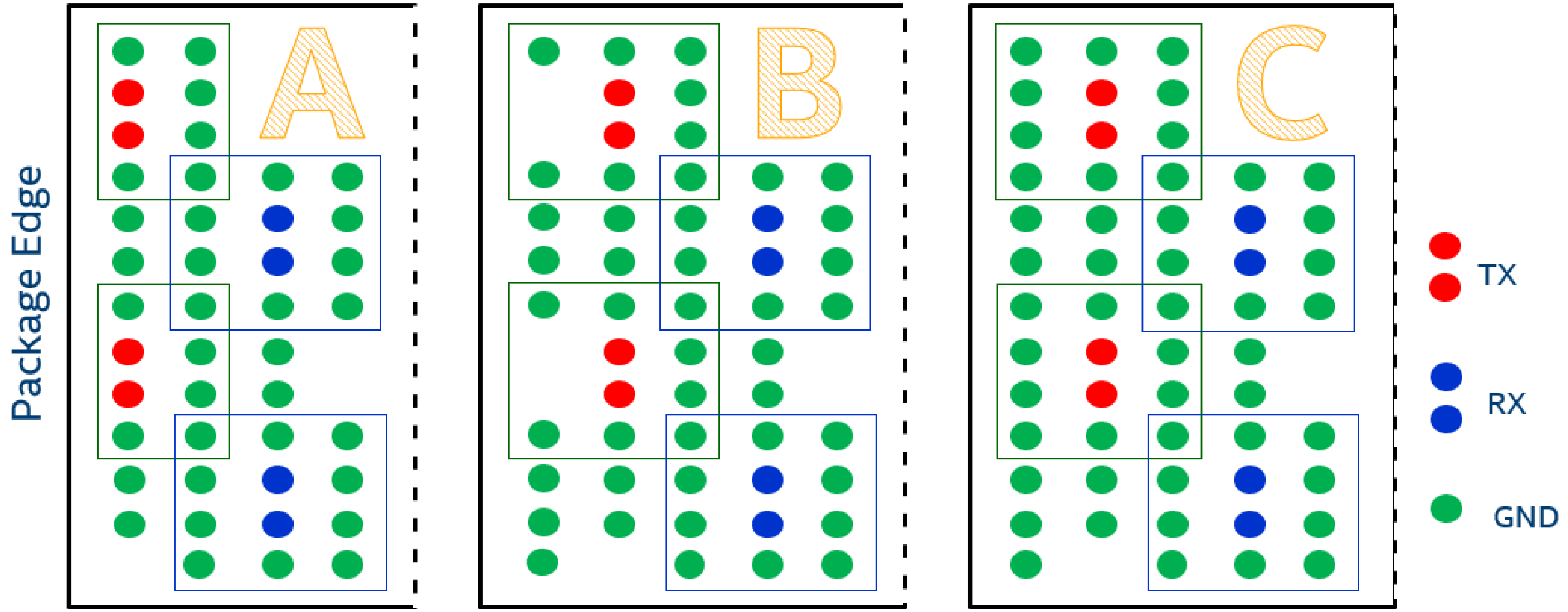






224G Package Ball Pattern Design

- **Case study of three BGA ball patterns**
  - Comparable return loss and insertion loss
  - 5-10 dB package cross talk improvement from A to C for TX



Package Localized Skip Layer Trace Design

- **Regular routing layer for**
  - GPIO/DDR
    - *Density and crosstalk control*
  - PDN
    - *Close ground reference*
- **Localized skip-layer routing for**
  - 224G Channels
    - *Loss control*

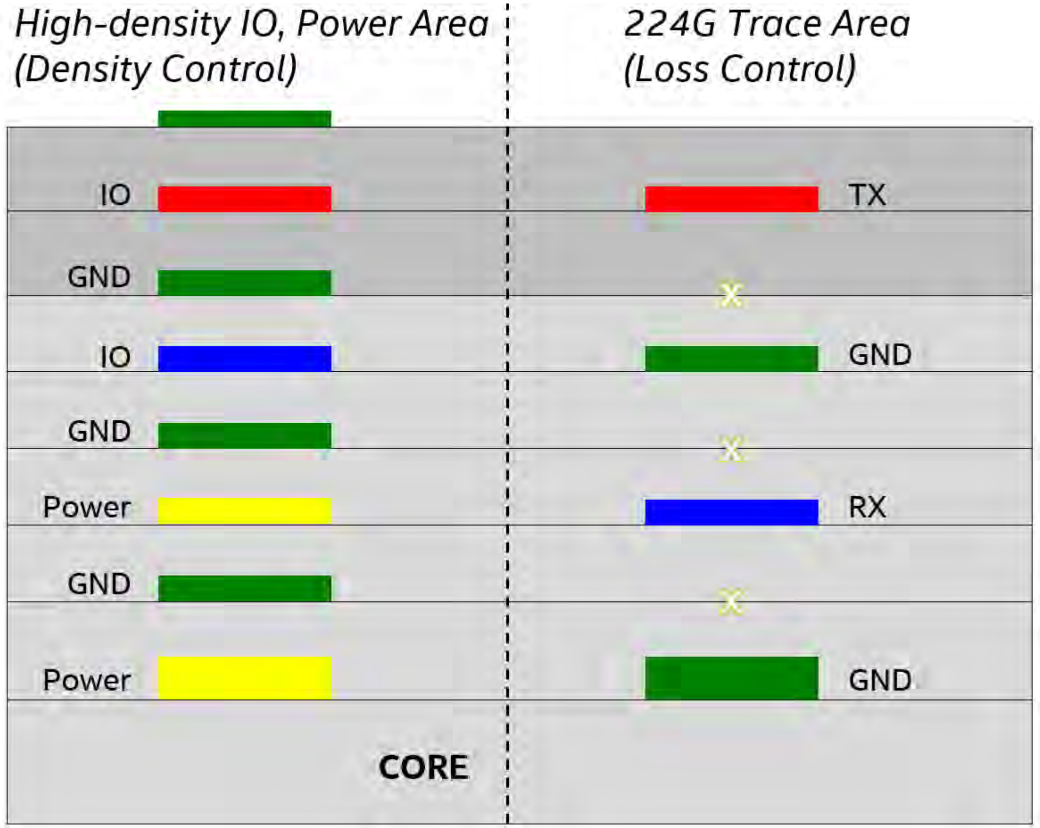
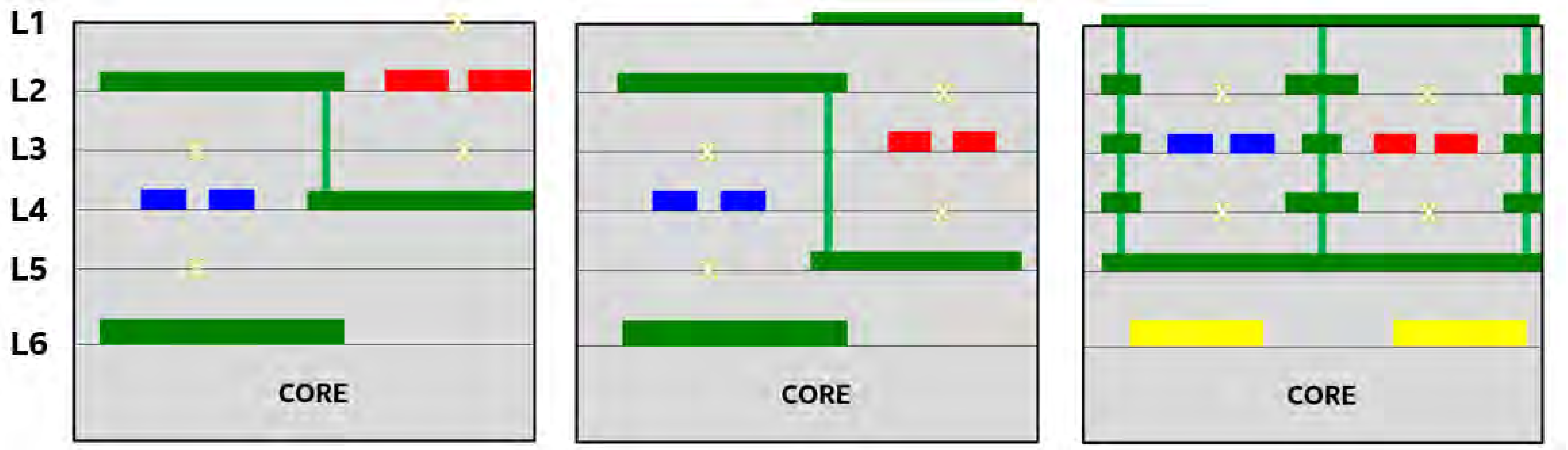


Illustration of a localized skip-layer configuration for 224G trace routing

Package Skip Layer Design: 12L Configurations



A

Embedded MSL
and STL

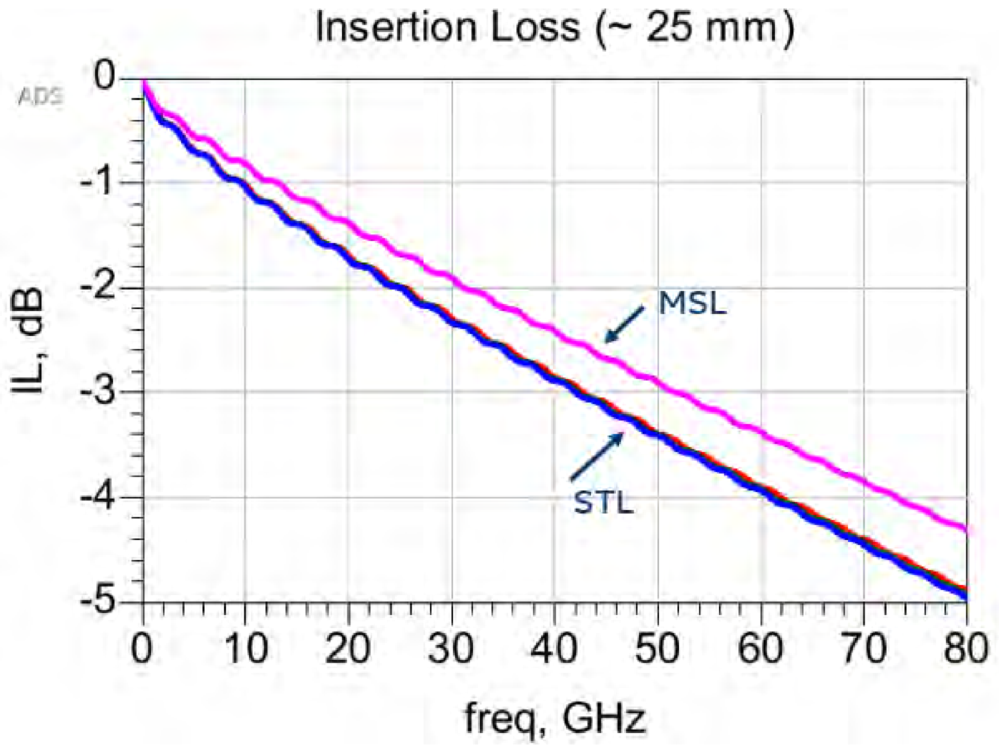
B

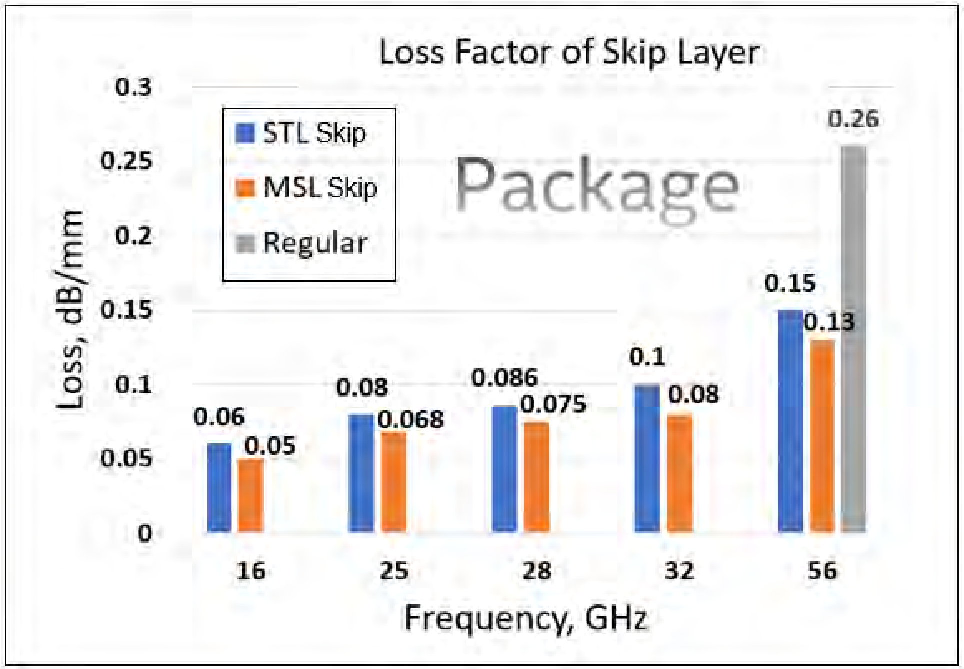
Shielded STL on
adjacent layers

C

Shielded STL on
same layers

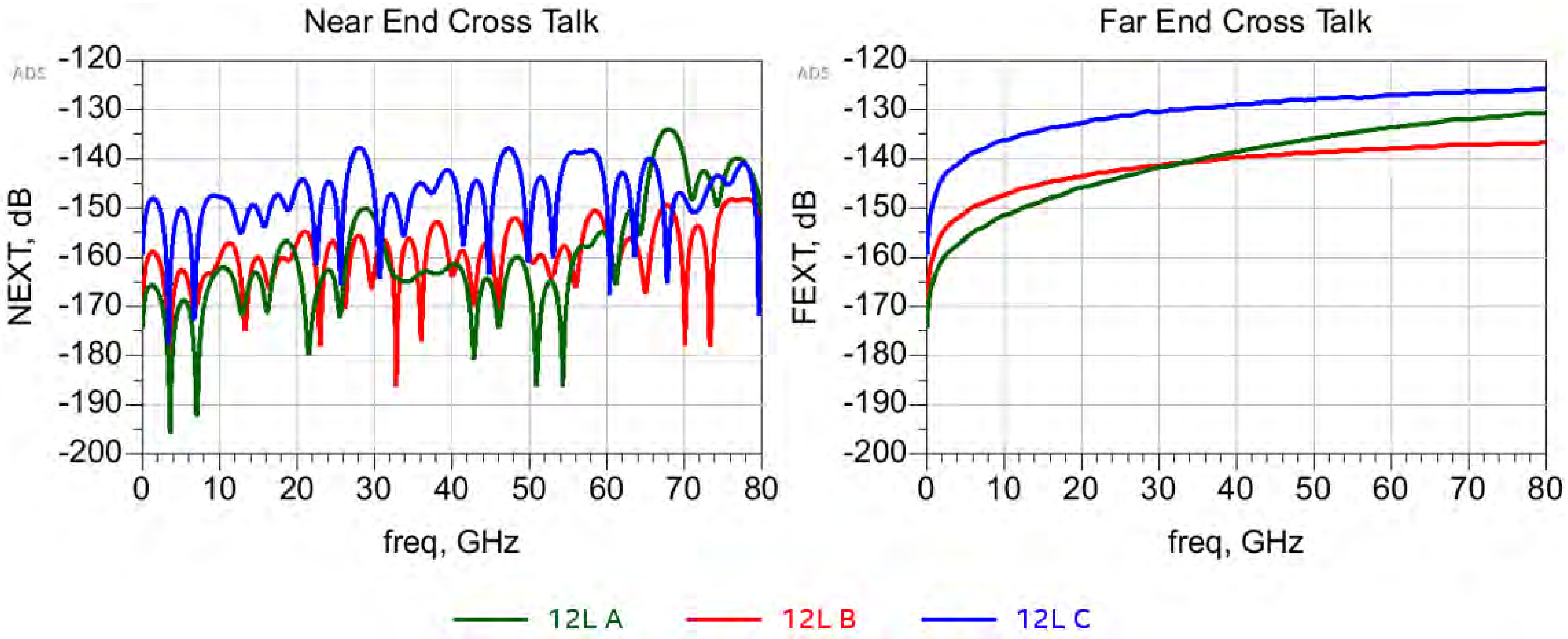
Package Skip Layer Trace Loss





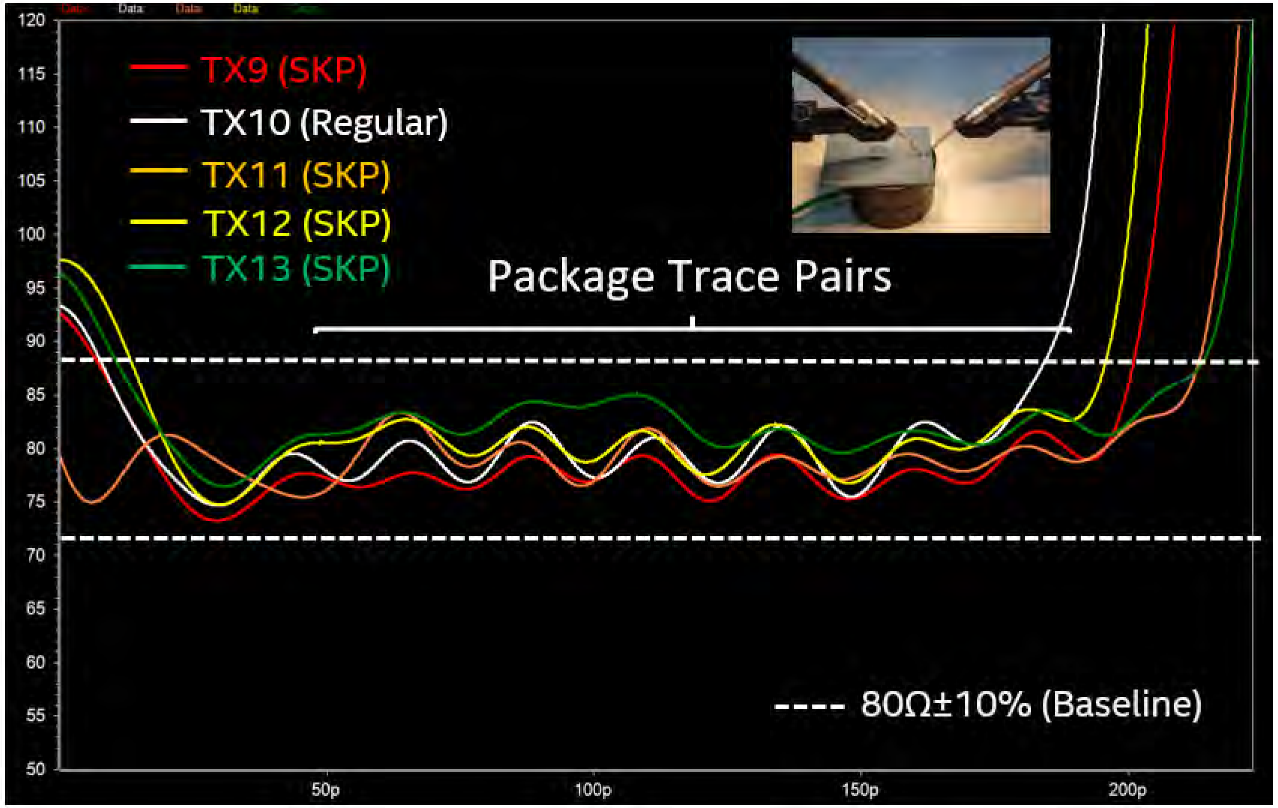
Notes: The trace loss was simulated based on current low loss material and copper surface treatment;
More advanced substrate material and copper surface treatment will further improve the package trace loss.

Package Skip Layer Trace Crosstalk



Takeaway: 25mm trace coupling is below -125dB to 56GHz following the suggested design guides.

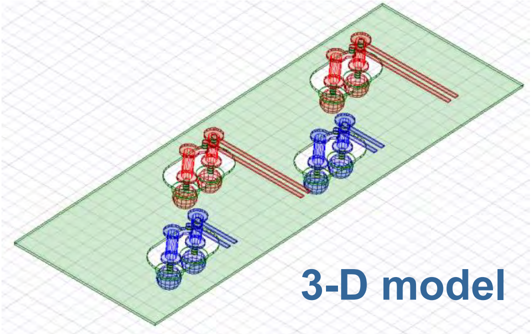
Package Trace Impedance Verification



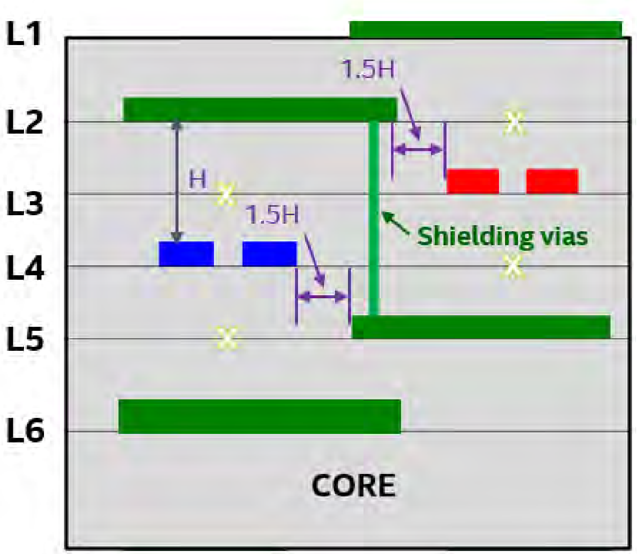
Notes: The TDR measurement verified the 80Ω differential impedance with ±10% tolerance for the skip-layer and the regular trace design;
No significant difference was seen between the skip-layer and the regular trace design from the TDR measurement.

224G Package Design Practice

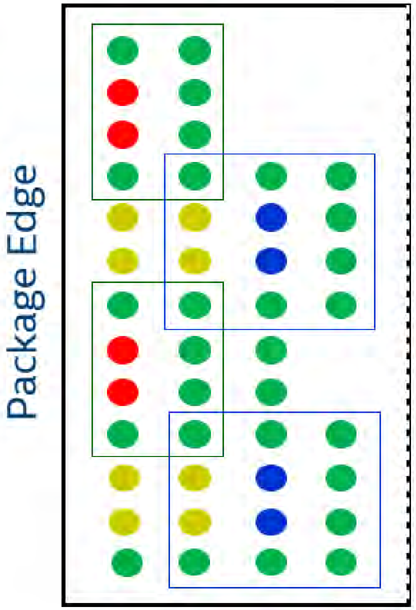
- **12L package with low loss material**
- **0.5mm ball pitch (ball pattern A)**
- **Skip-layer configuration B**
- **400 um core**
- **Trace-PTH-BGA optimization**
  - o Discontinuities
  - o Loss
  - o Crosstalk
  - o PCB breakout



3-D model



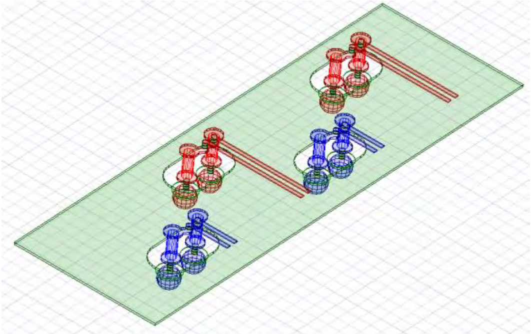
Skip-layer trace

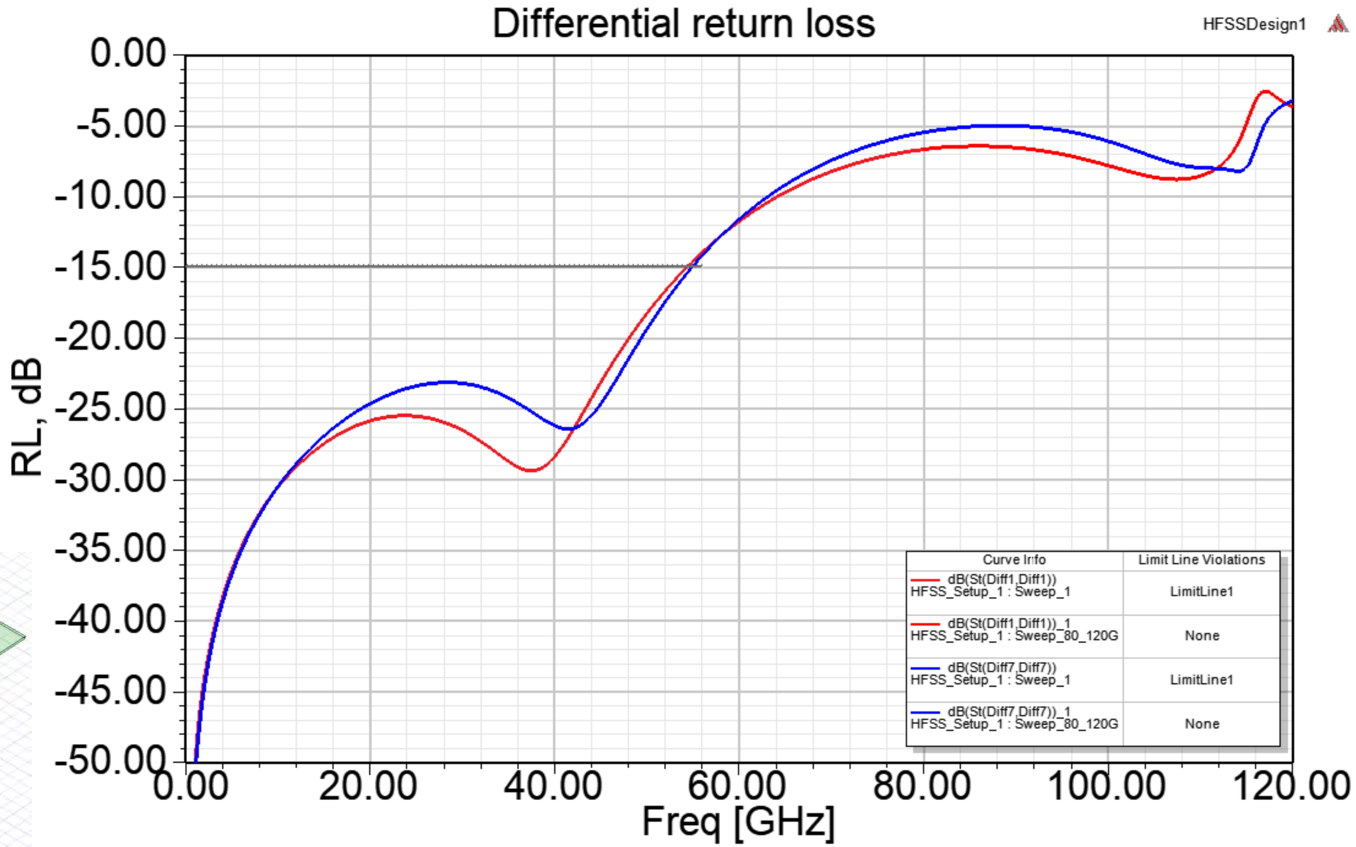


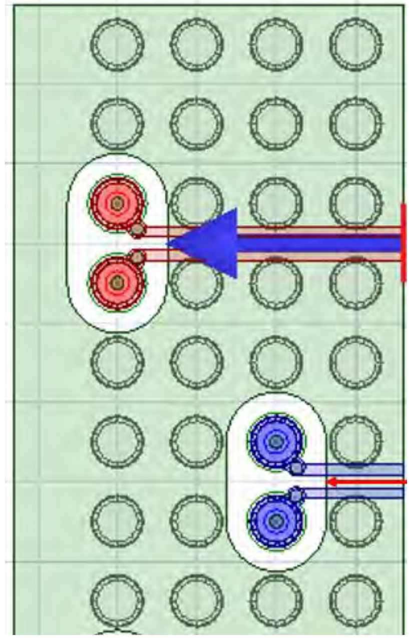
BGA ball pattern

Package Differential Return Loss

- **De-embed trace**
- **-15 dB to 56GHz**

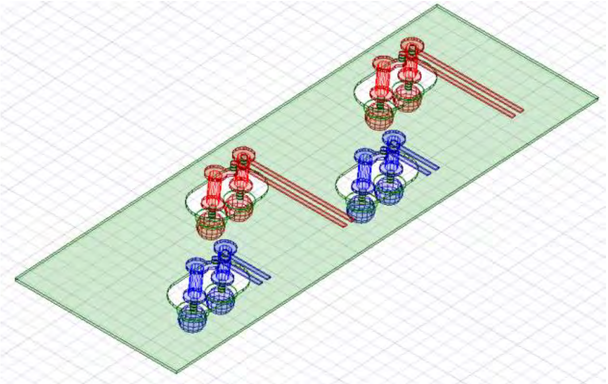


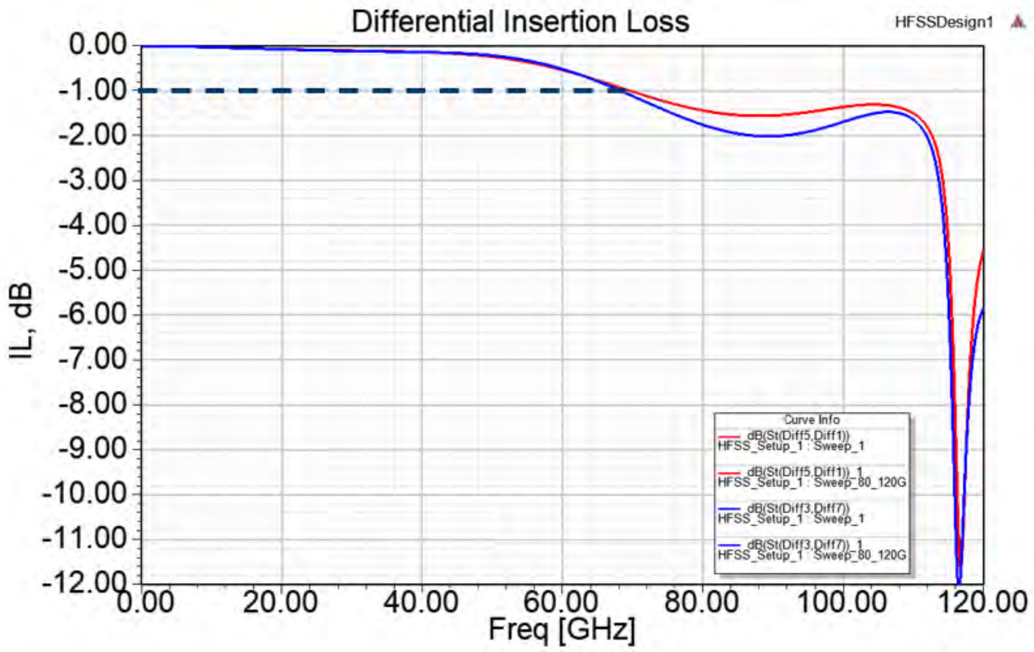


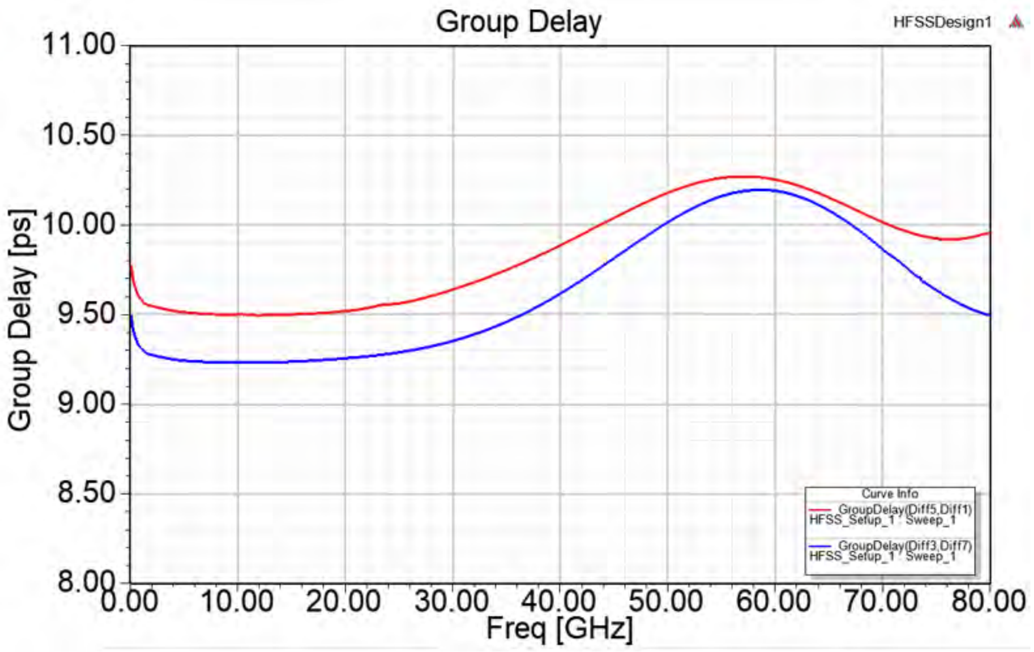


Package Differential Insertion Loss and Group Delay

- **1 dB BW: ~ 68GHz**
- **Group delay skew: < 1ps to 80GHz**

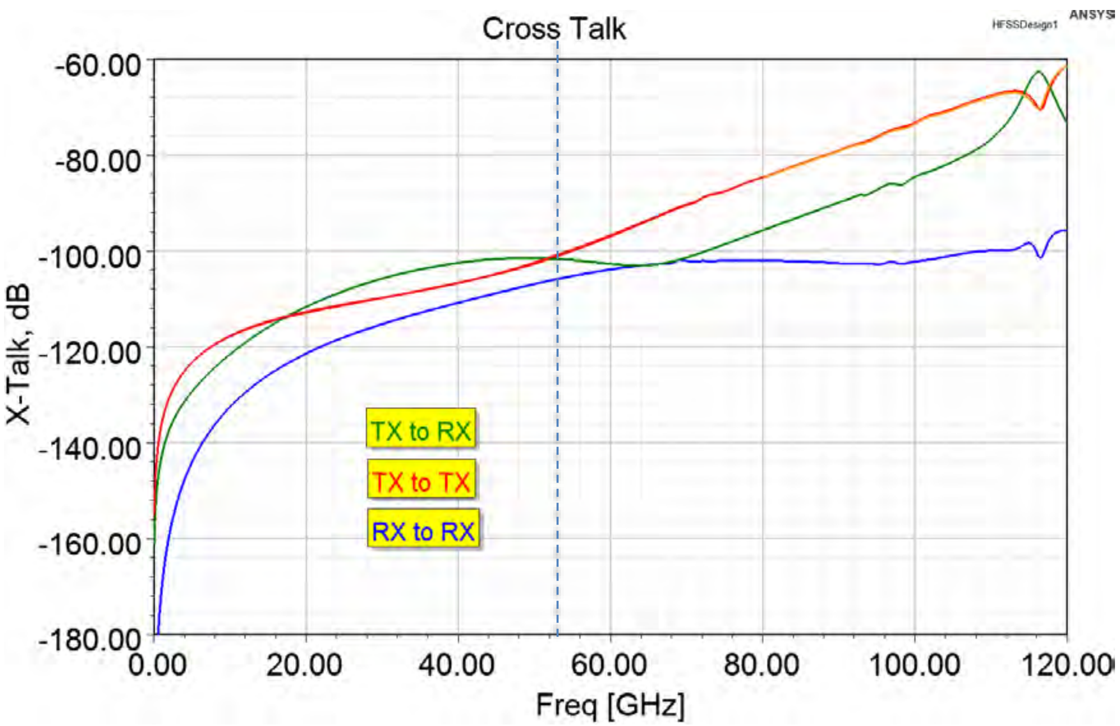


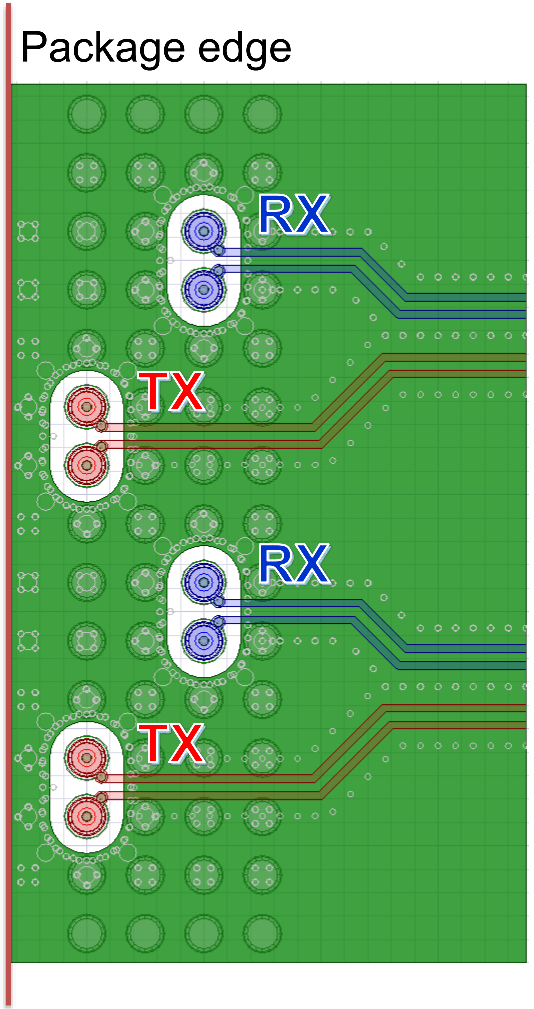




Package Differential Crosstalk

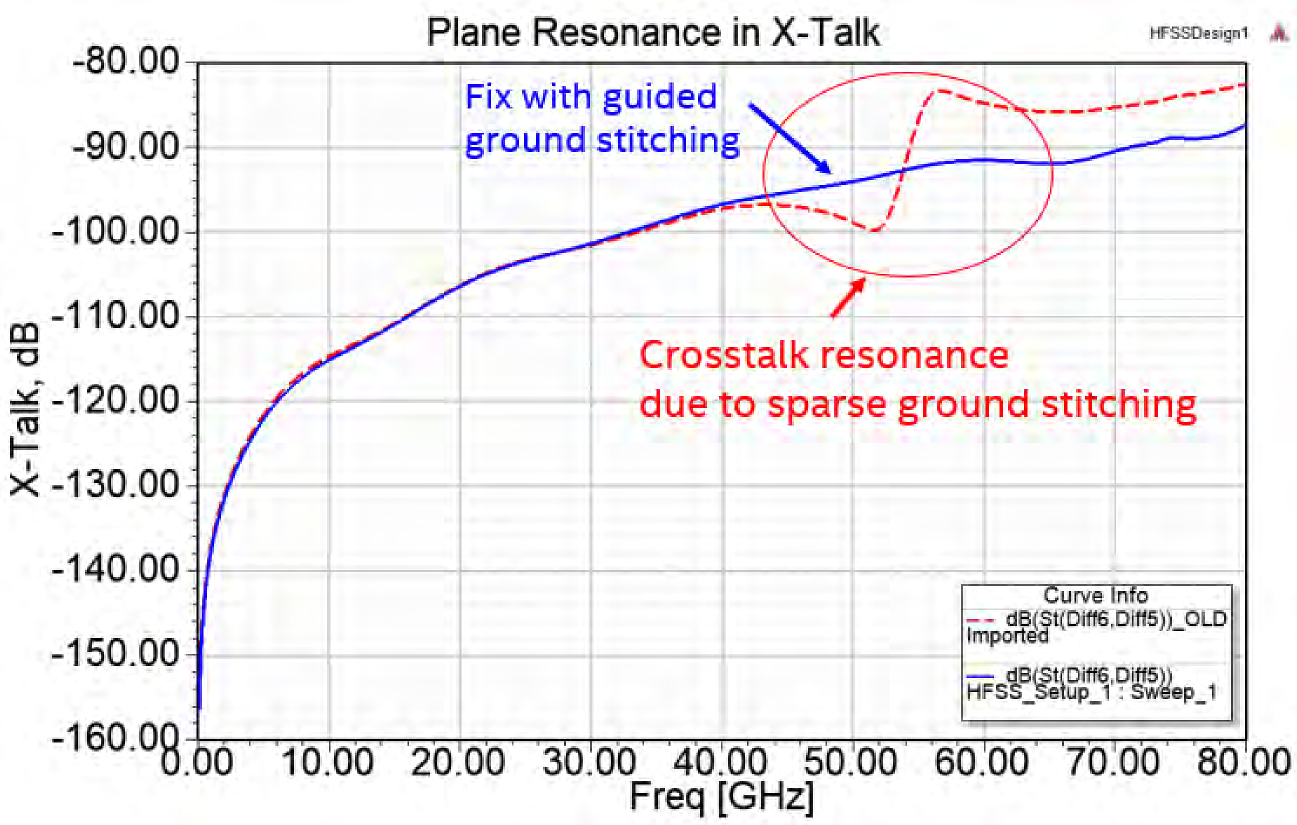
- Crosstalk below -98dB up to 56GHz
- Crosstalk to edge ball without full shielding increases dramatically with frequencies beyond 56GHz





Package Plane Resonance

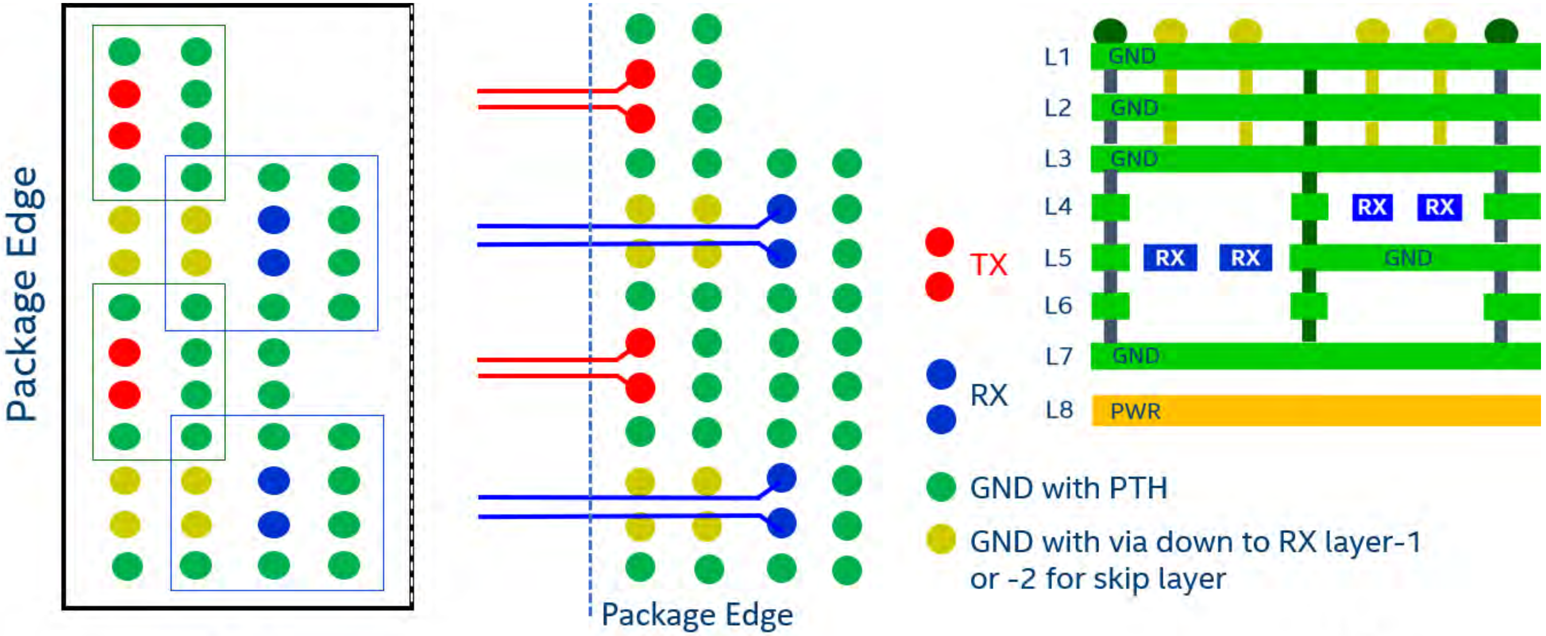
- **Ground plane stitching via guideline**
  - Via pitch < 1/10 wavelength along the TX/RX traces
  - Via pitch < ¼ wavelength everywhere else in the vicinity of 224G channel routing



PCB Design – BGA Ball Breakout

- Smart BGA ball pattern can ease the PCB breakout
- Properly separated ball pairs

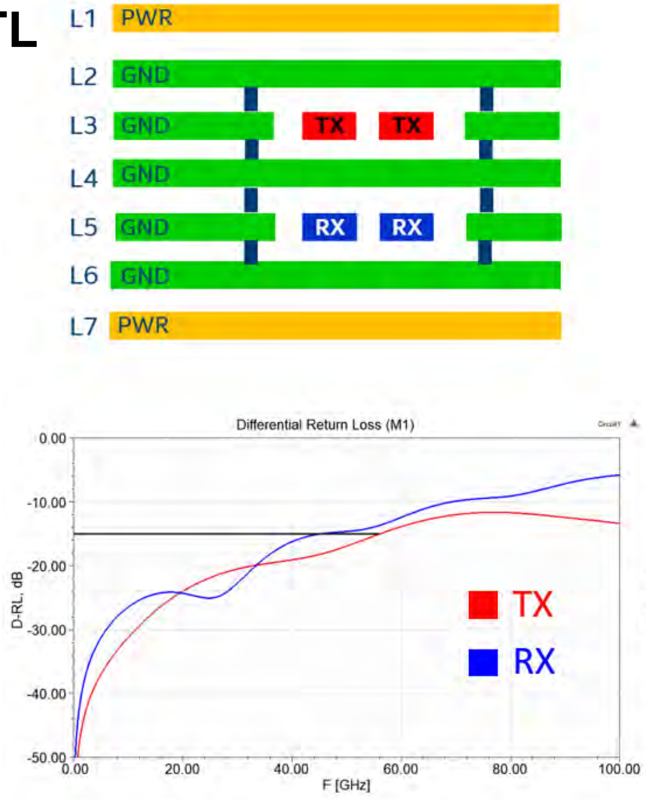
  can reduce trace to via coupling



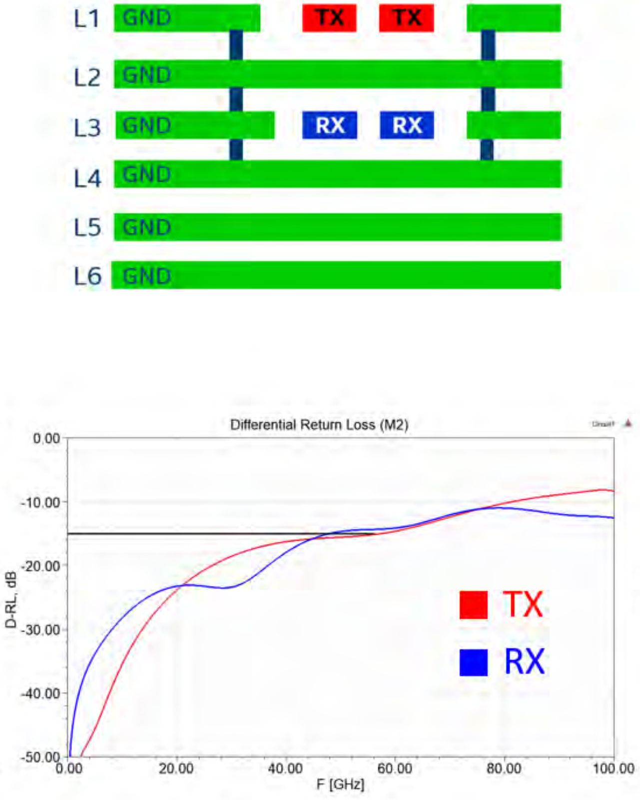
(a) (b) (c)

PCB Design – Trace Routing Configurations

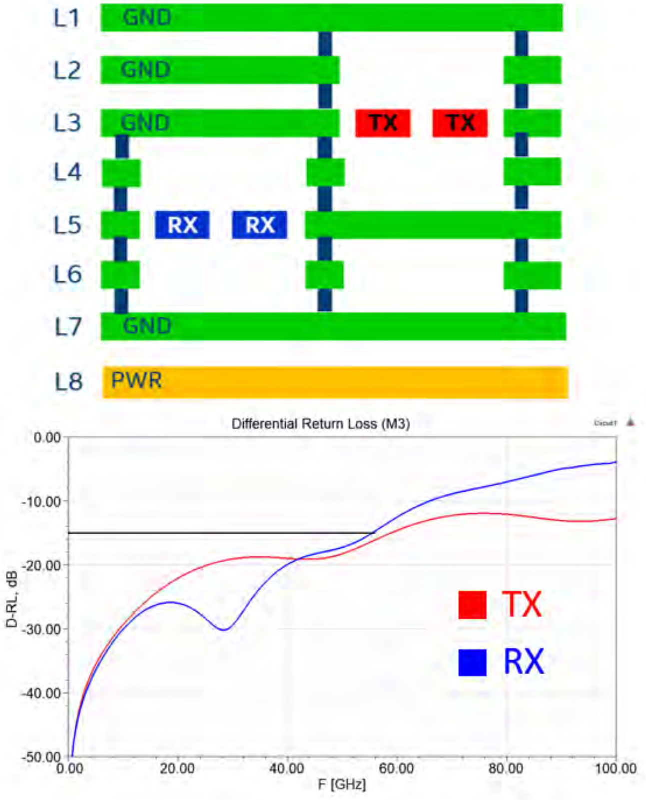
- **Regular STL**
- **Regular MSL + STL**
- **Skip-layer STL**



(a) Regular STL



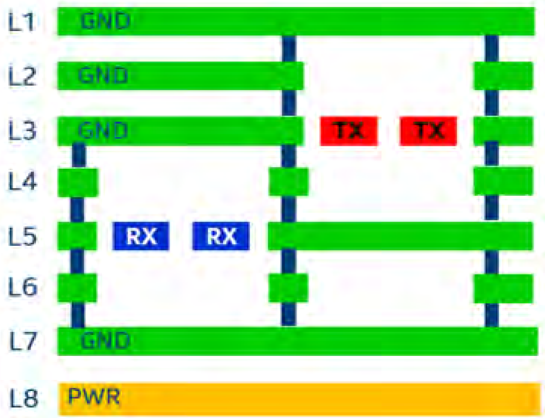
(b) Regular MSL

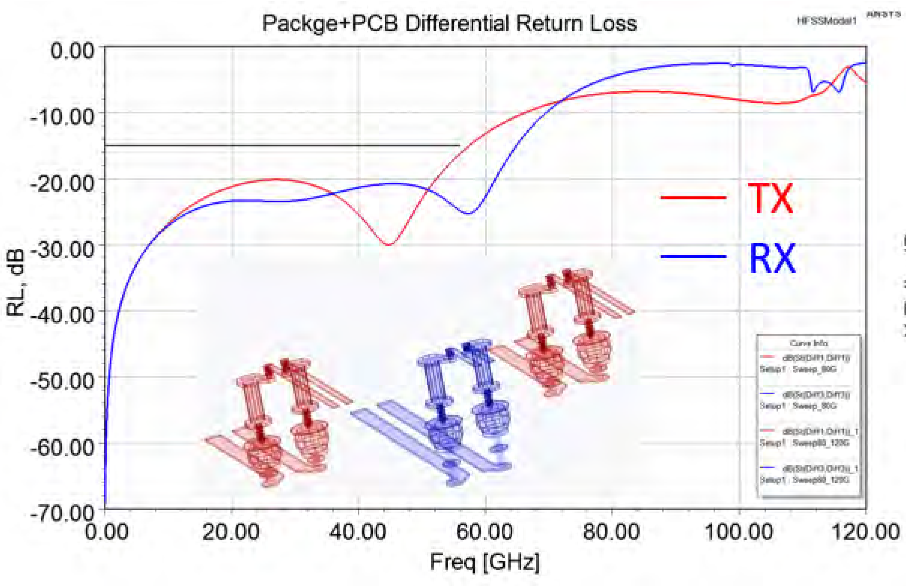


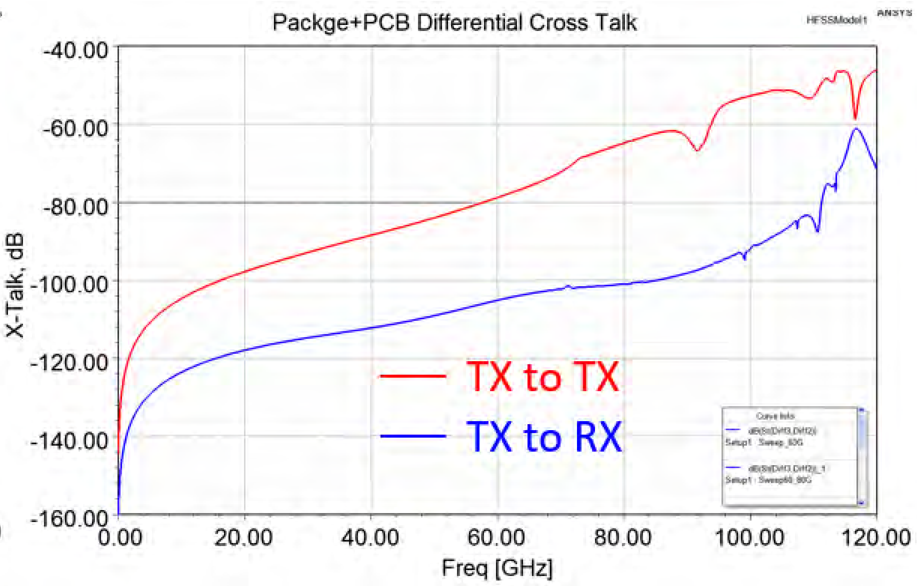
(c) Skip Layer STL

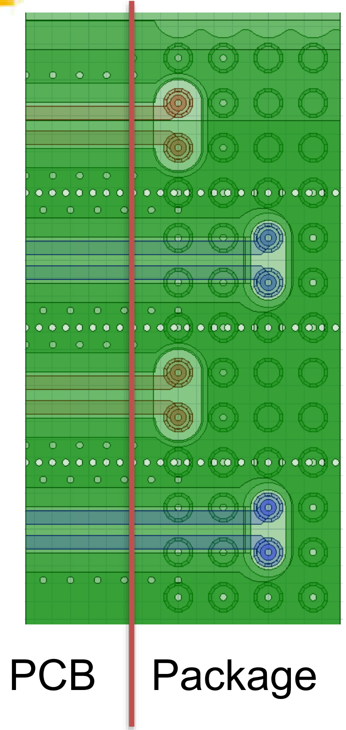
Package-PCB Co-modeling

- Package-PCB breakout return loss < -15 dB to 56GHz
- Edge ball/via coupling < -80 dB to 56GHz
- Inner ball/via coupling < -100 dB to 56GHz



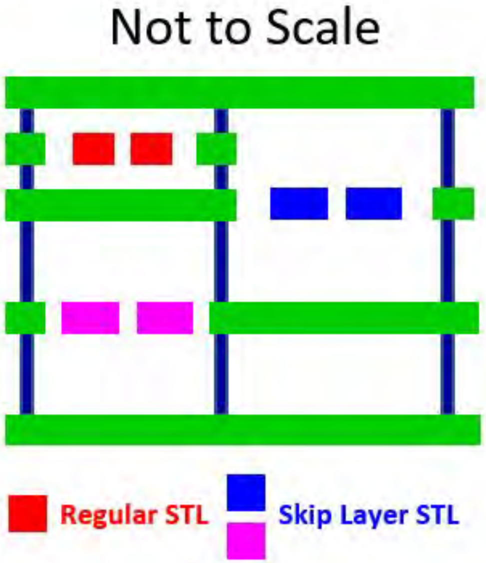


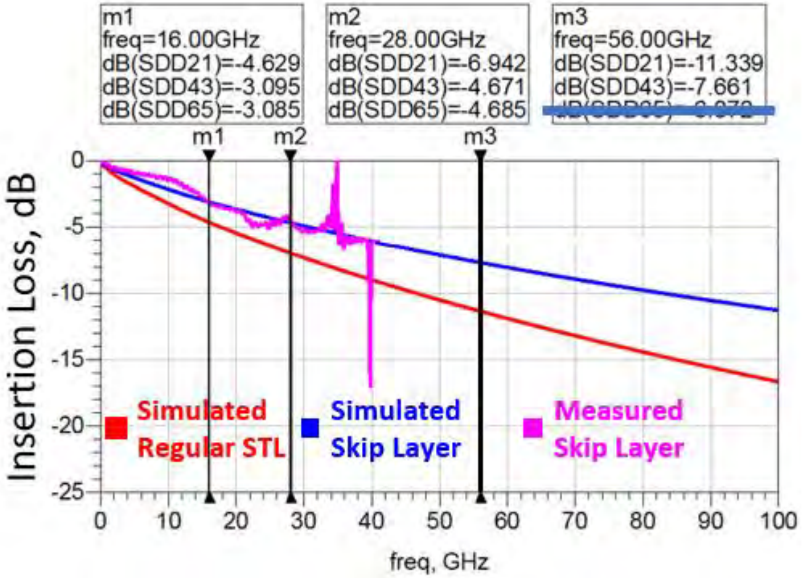




PCB Trace Loss Correlation

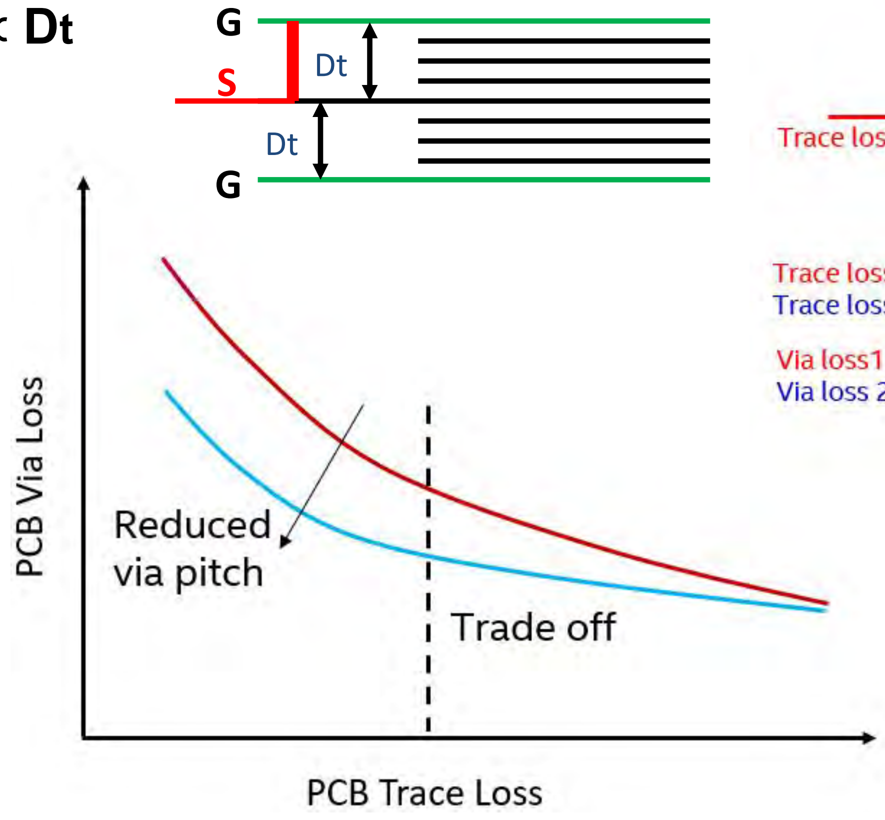
- **Standard PCIe AIC (Add-in-Card) stackup**
- **PCB material: M7N**
- **Trace loss at 56GHz**
  - o Regular STL: 2.8dB/inch
  - o Skip-layer STL: 1.9dB/inch

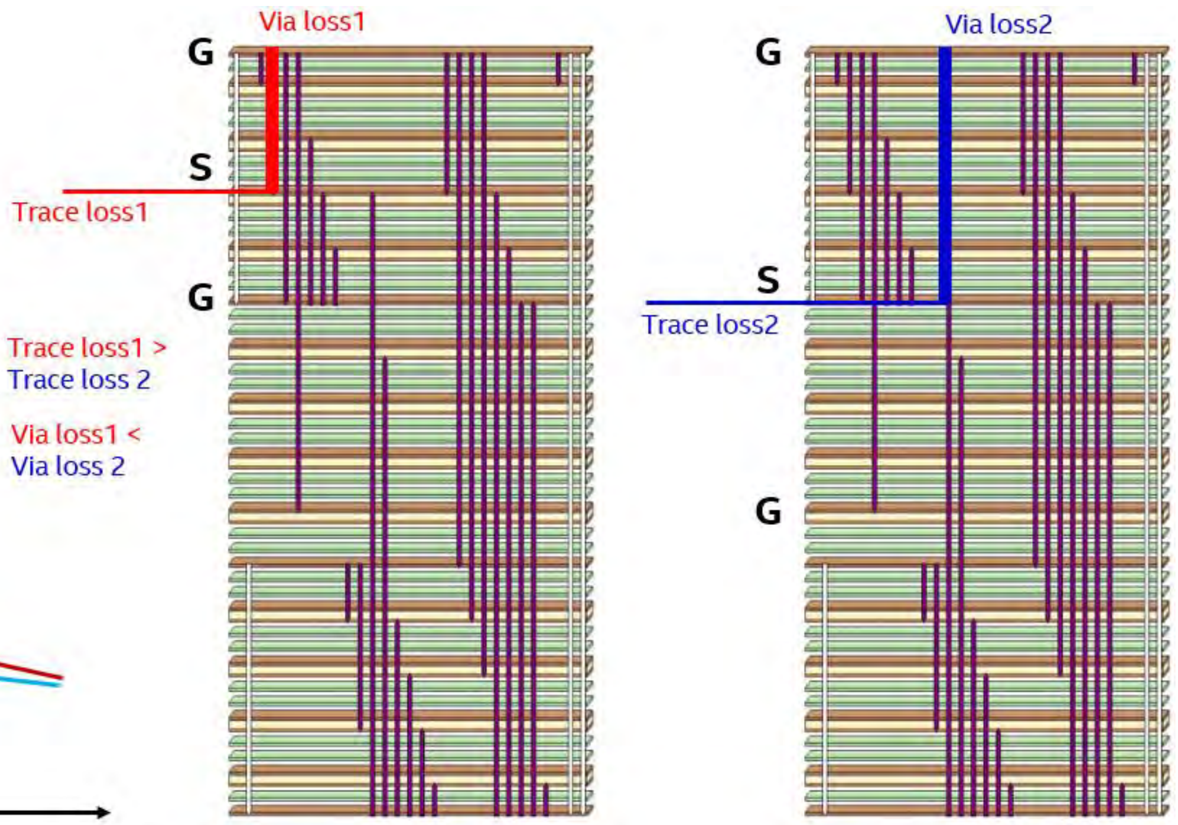




PCB Loss Improvement

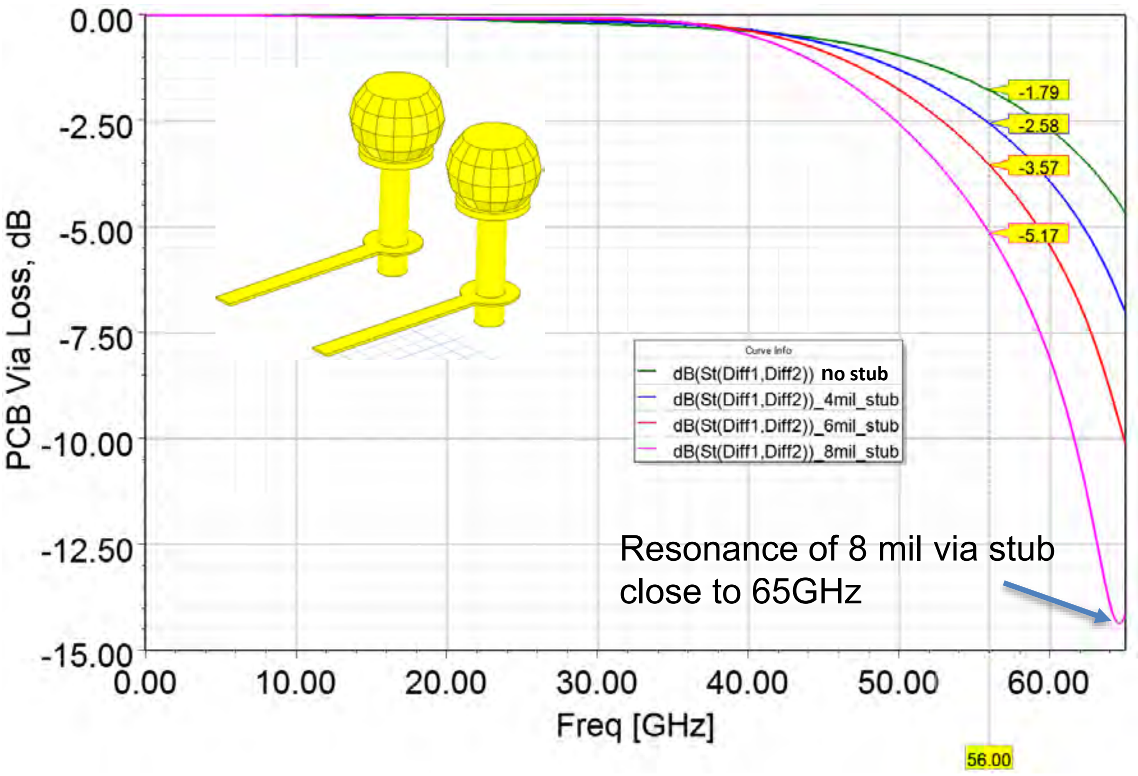
- **Skip-layer trace routing**
- **PCB trace loss ∝ 1/Dt**
- **PCB via loss ∝ Dt**
- **Trade-off of Dt**





PCB Via Stub Effects

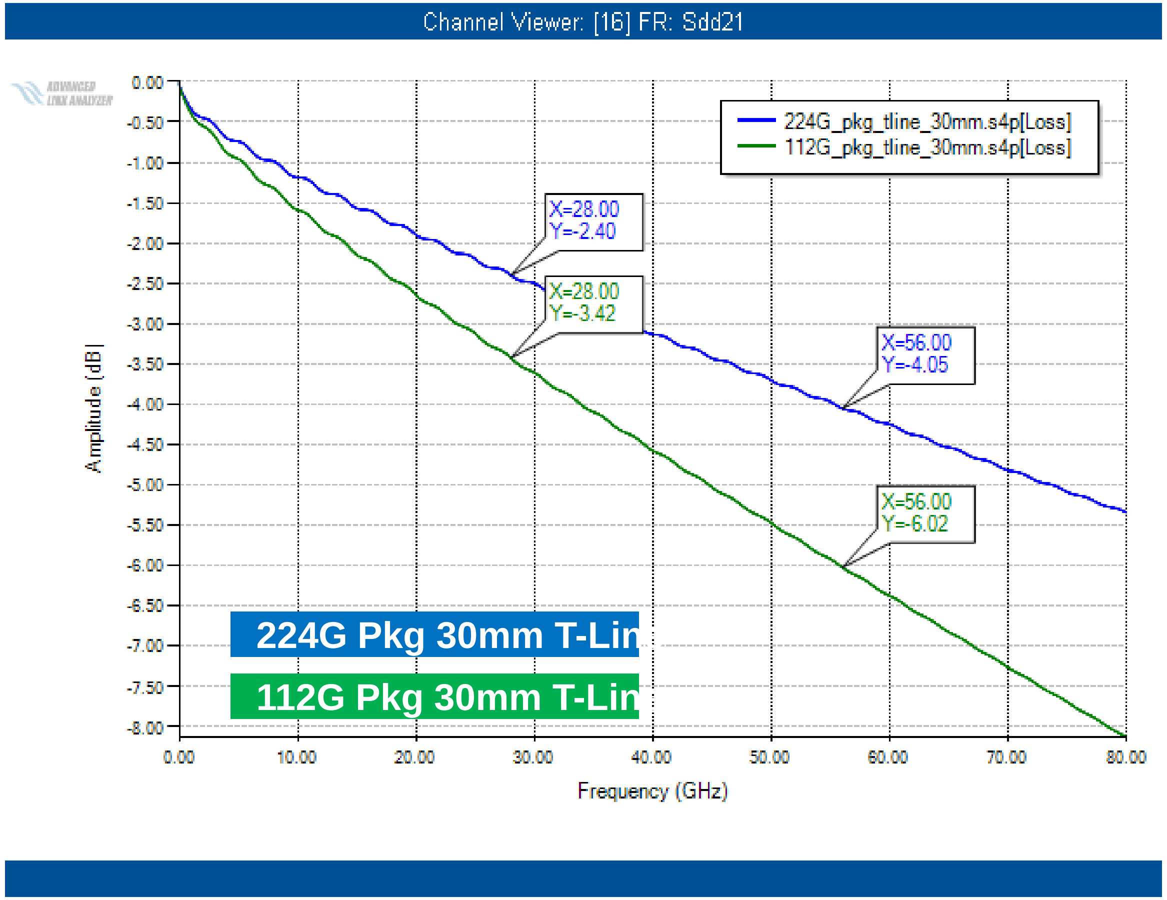
- PCB via loss increases with via stub length
- < 6 mil via stub length is recommended for 224G PAM4 design



PCB via loss with different via stub lengths

A Proposed Host Reference Package Model for 802.3df

Parameter	112G Package T-Line Model Parameters	Proposed 224G Package T-Line Model Parameters
Z_p	30 mm	30 mm
γ_0	0 /mm	**0 /mm**
τ	6.141e-3 ns/mm	**6.141e-3 ns/mm**
a_1	9.909e-4 ns$^{1/2}$/mm	**8.9e-4 ns$^{1/2}$/mm**
a_2	2.772e-4 ns/mm	**1.55e-4 ns/mm**
Z_c	87.5 Ω	**87.5 Ω**
R_o	50 Ω	50 Ω
C_p	87 fF	**40 fF**



224G PAM4 Package Design Summary

- **Desired next generation package trace loss target for interpretation flexibility: 0.1 dB/mm at Nyquist frequency**
  - Skip-layer trace routing is required for mitigating the transmission loss
  - Low loss material and advanced copper surface treatment are required
- **0.8mm ball pitch is recommended (0.65mm or smaller preferred)**
- **Smaller ball size can further reduce discontinuities and package loss**
- **BGA ball pattern needs to be PCB breakout friendly and fully shielded**
- **Ground stitching via pitch < 1/10 wavelength along TX/RX traces and < 1/4 wavelength everywhere else in the vicinity of the 224G channel routing are required**

224G PAM4 PCB Design Summary

- **Desired next generation PCB trace loss target for interpretation flexibility: 1 dB/inch at Nyquist frequency**
  - Skip-layer trace routing is required
  - Ultra low loss material is required
  - HVLP copper surface treatment is required
- **PCB via stub length < 8mil is required**
- **Well controlled process variation of Dk, Df and dielectric thickness is required**

References

[1] J. Jiang et al, “Designing 224G PAM4 High Performance FPGA Package and Board with Confidence”, *Designcon*, 2021.

Thank You!