



Using Simulation to Optimize an Interface for 10GHz

Ryan Satrom | Multitest | r.satrom@multitest.com

Introduction

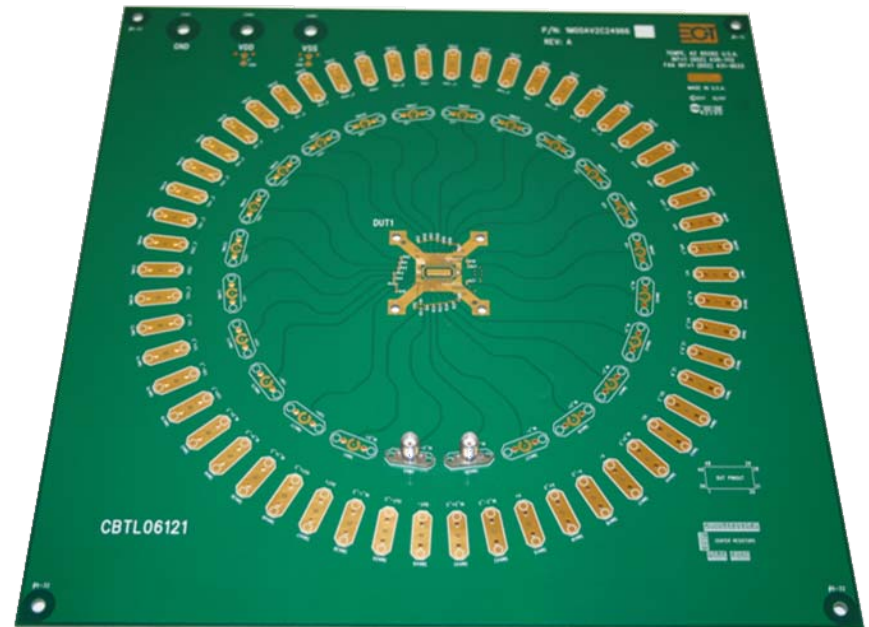
- Interface design for high-frequency devices often requires detailed analysis
- Current design standards are often inadequate for signals with multi-GHz frequencies
- 10+ GHz interfaces that are designed without careful attention to signal integrity will fail

Agenda

- This presentation describes the process of meeting a 10 GHz interface requirement:
 - Original design review and interface modifications
 - Simulation-to-measurement correlation

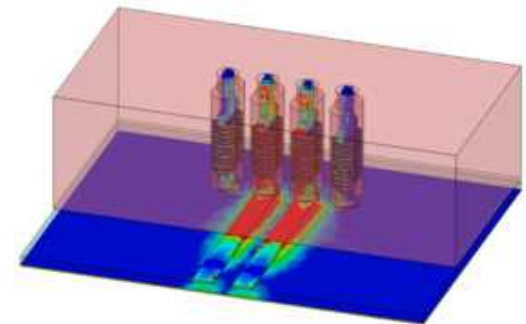
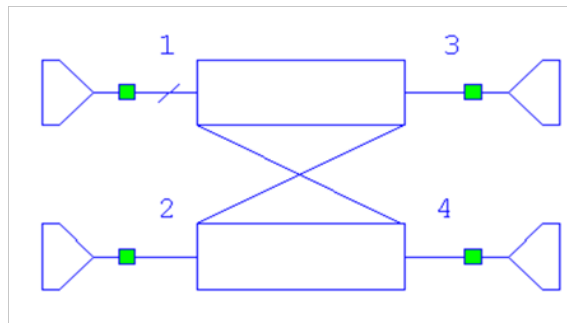
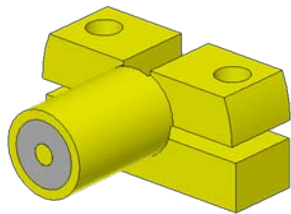
Structures Optimized

- >Original design failed due to insufficient bandwidth, requiring a complete redesign
- >Several structures were analyzed and modified to optimized performance:
 - >Connector
 - >Trace
 - >Contactor
 - >Path Grounding



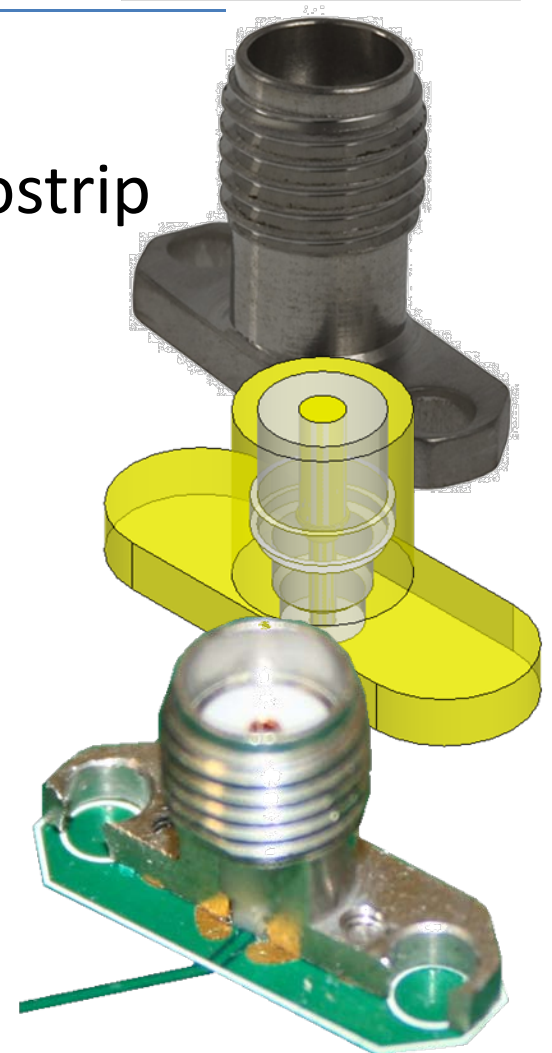
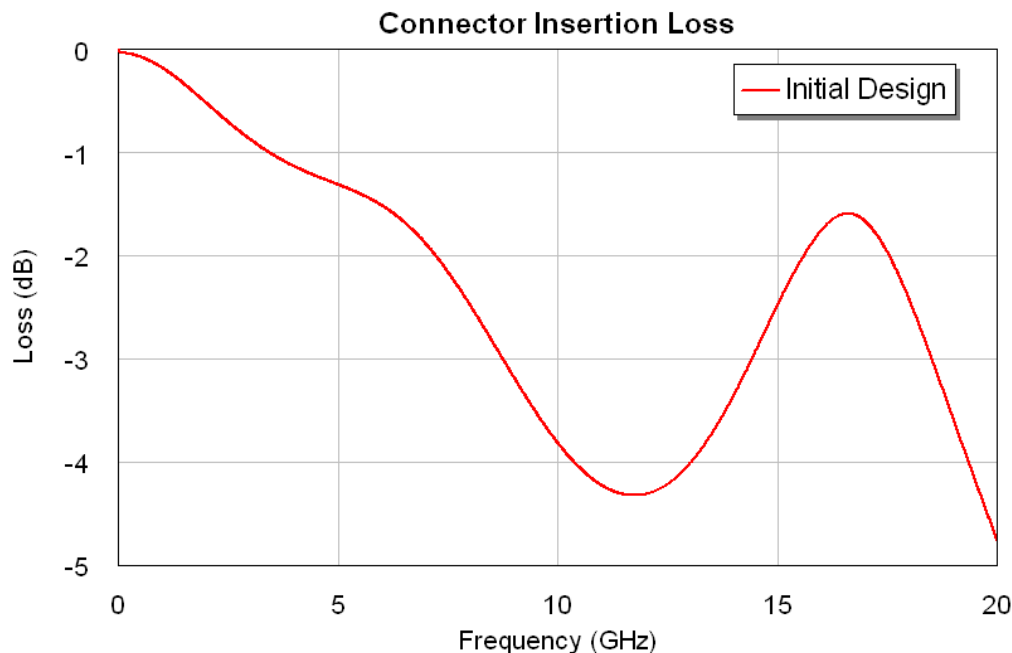
Structures Optimized

- Methods for optimization differ based on structure:
 - **Connector** - 3D electromagnetic (EM) simulation to optimize footprints, including ground via locations, via diameter, ground cutouts
 - **Trace** – 2D simulation to optimize trace width and ground location (for CPW)
 - **Contactor** - 3D EM simulation to optimize probe choice, contactor design
 - **Path Grounding** – No simulation; design optimized by minimizing ground loops



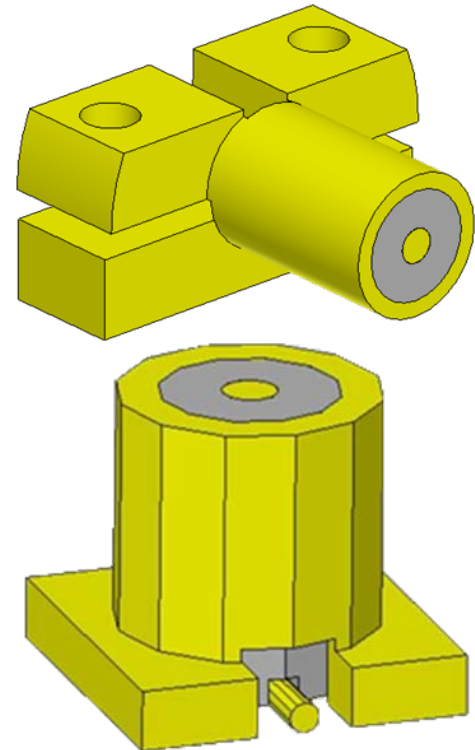
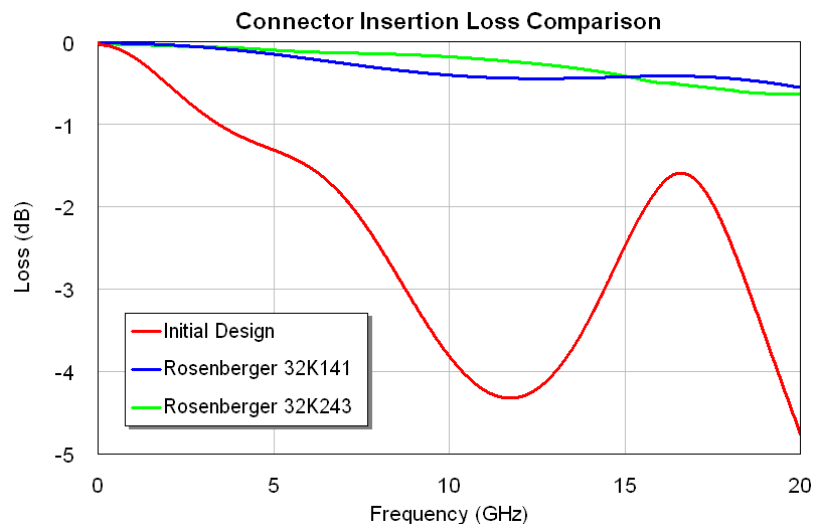
Original Design - Connector

- Original Connector – Molex SMA
- Poor high-speed connector for microstrip
- Footprint not optimized



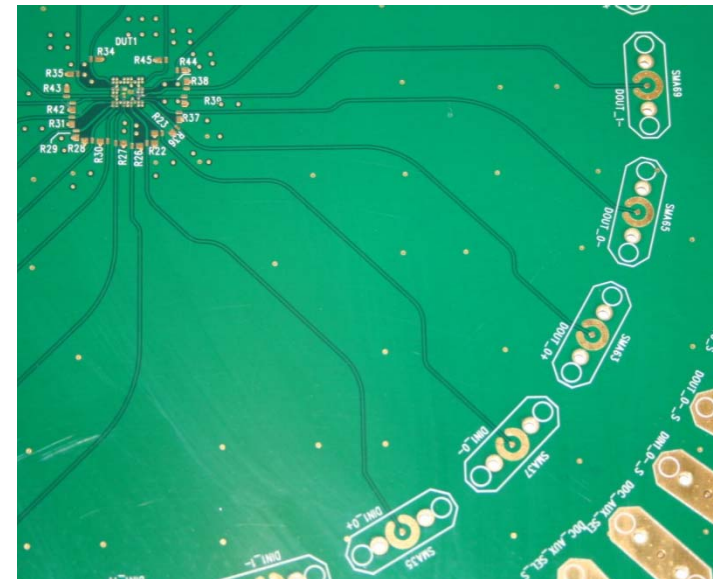
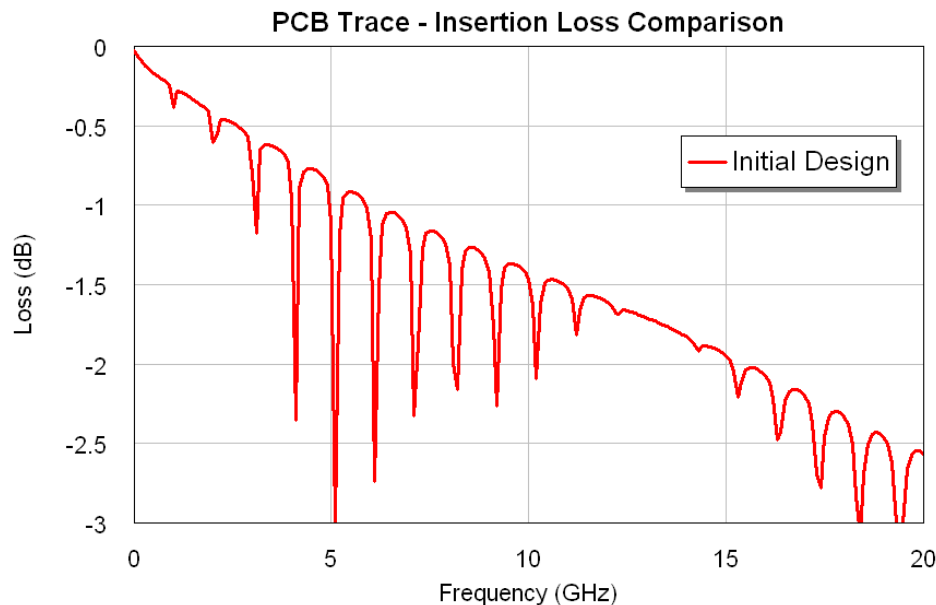
Redesign - Connector

- Optimized connector possibilities
 - Rosenberger 32K243, Rosenberger 32K141
- Connector footprints optimized through simulation
- Rosenberger 32K243 chosen



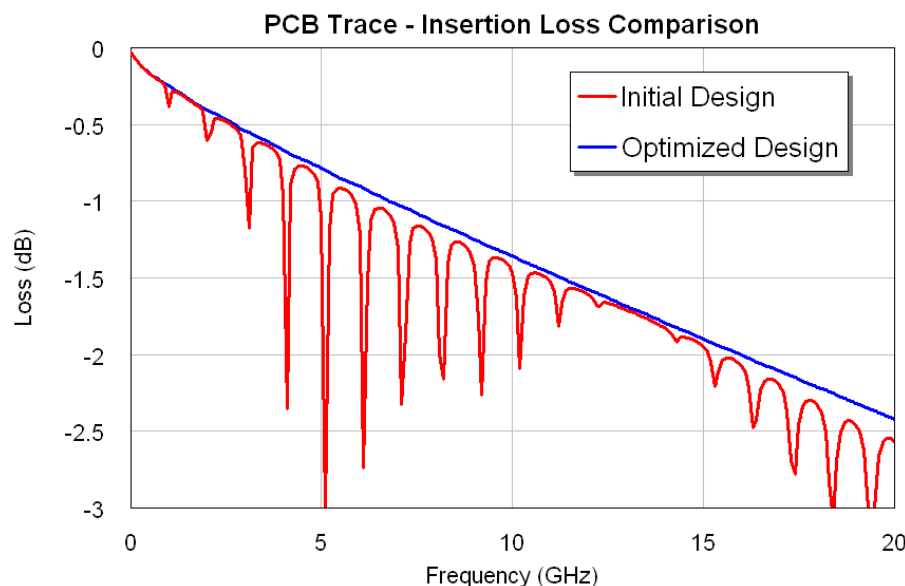
Original Design – Trace Vias

- Original design – ground via locations arbitrary
 - Coplanar waveguide (CPW) traces require vias in close proximity
 - CPW traces resonate if not properly designed
 - Resonances must be removed



Redesign – Trace Vias

- Coplanar waveguide traces
 - CPW traces resonate at $\frac{1}{2}$ -wavelength of distance between vias
 - 100mil ground via spacing (no resonances until $\sim 30\text{GHz}$)

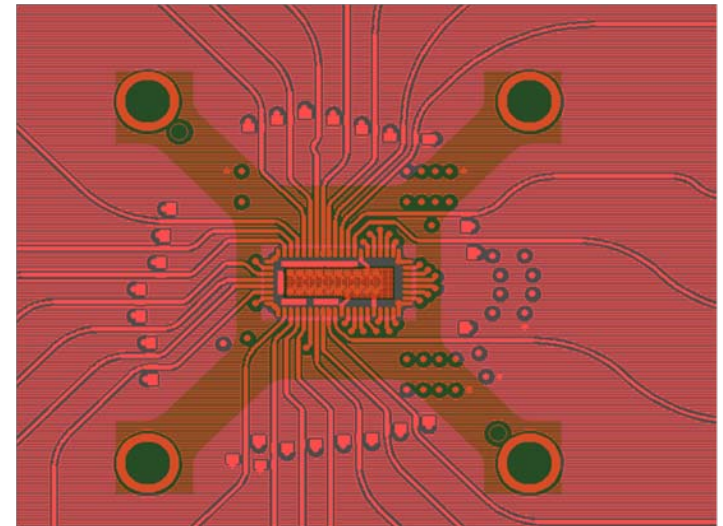


$$f_{RES} = \frac{c}{2\lambda\sqrt{\epsilon}}$$

*****CPW traces were required by customer**

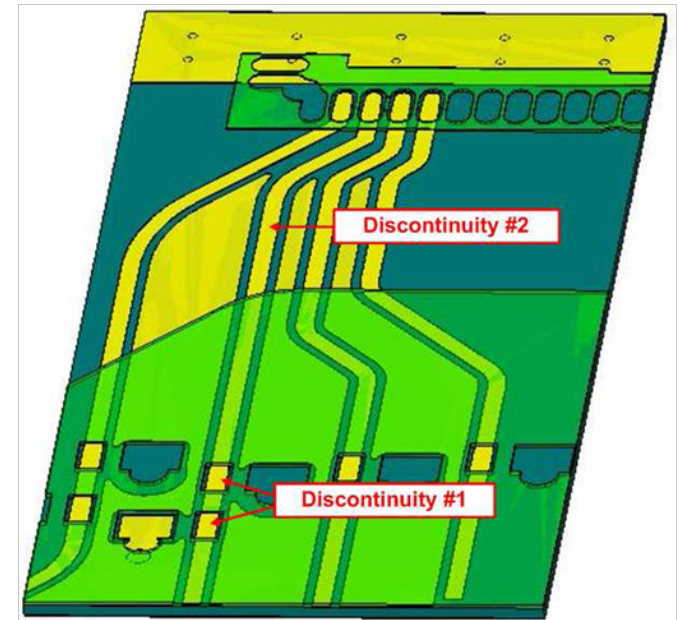
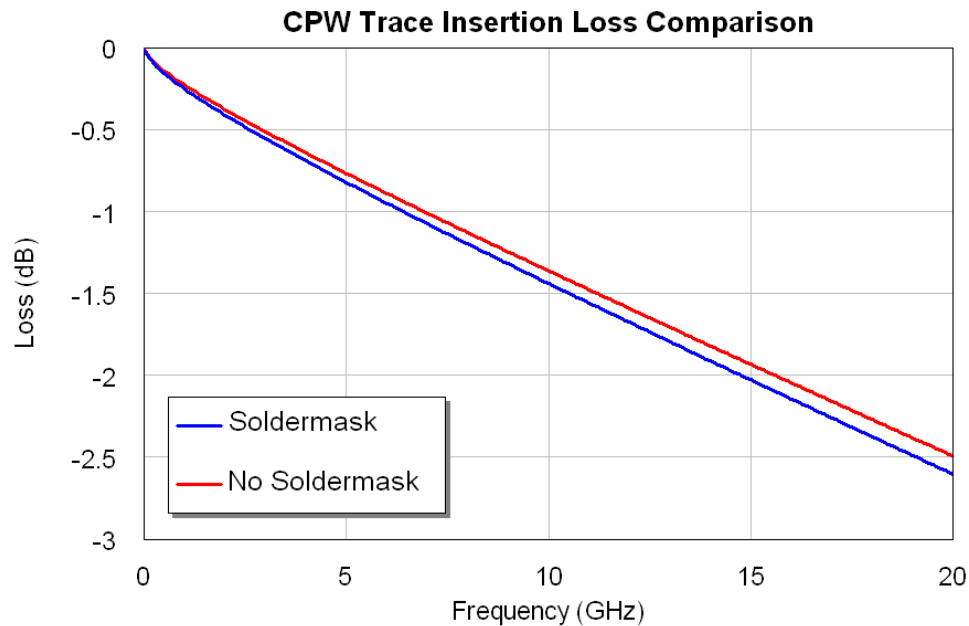
Original Design – Trace Z

- Original Design – Trace width not adjusted as path changes
- Slight changes in trace geometry can have performance impact at multi-GHz frequencies
- Factors that may impact impedance
 - Soldermask
 - CPW coupling
 - Trace-to-Trace coupling
 - Trace traveling under contactor



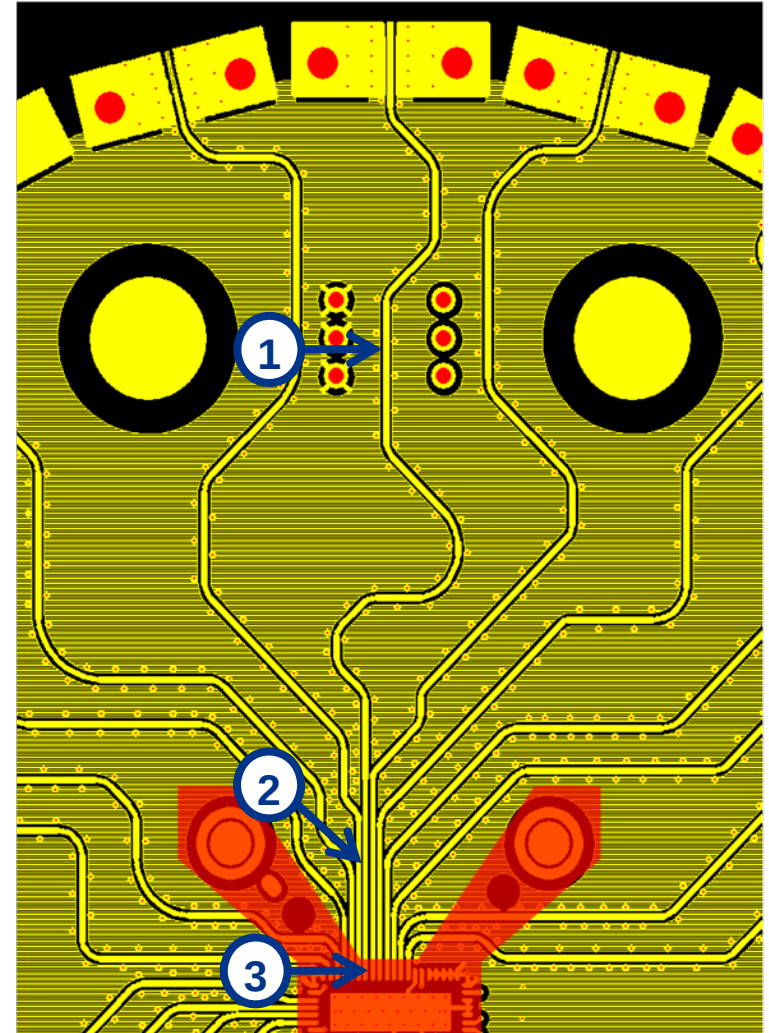
Redesign – Trace Z

- Remove Soldermask
 - Removes any discontinuities from soldermask/no-soldermask transition
 - Removal has minor impact on loss



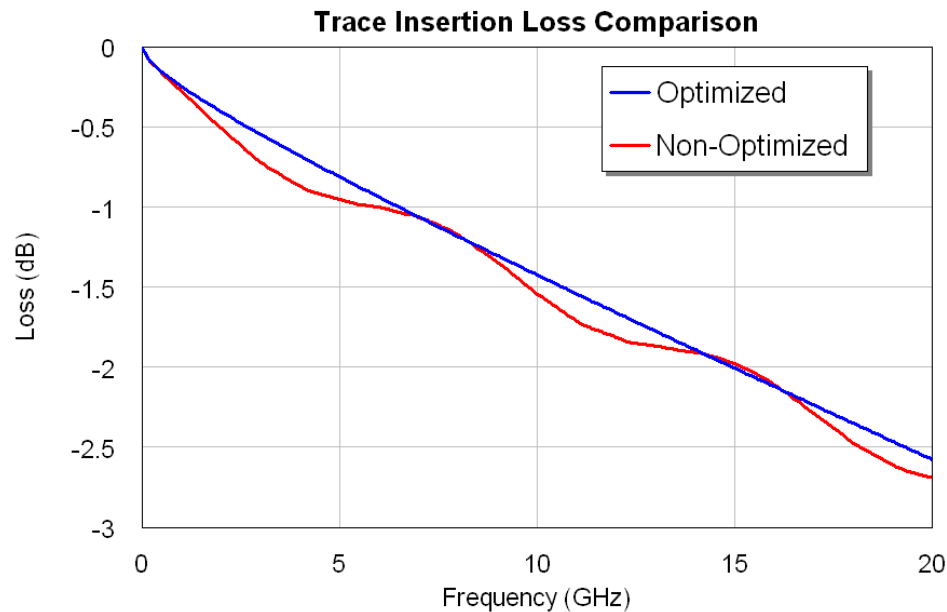
Redesign – Trace Z

- Trace Width Optimization
- Three different regions
 - 1) Uncoupled CPW trace
 - 2) Coupled trace under air
 - 3) Coupled trace under socket



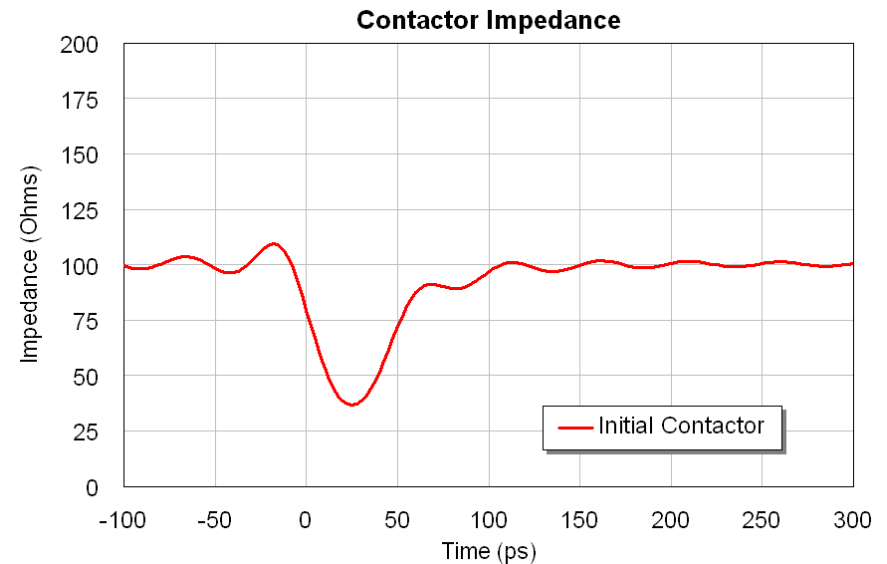
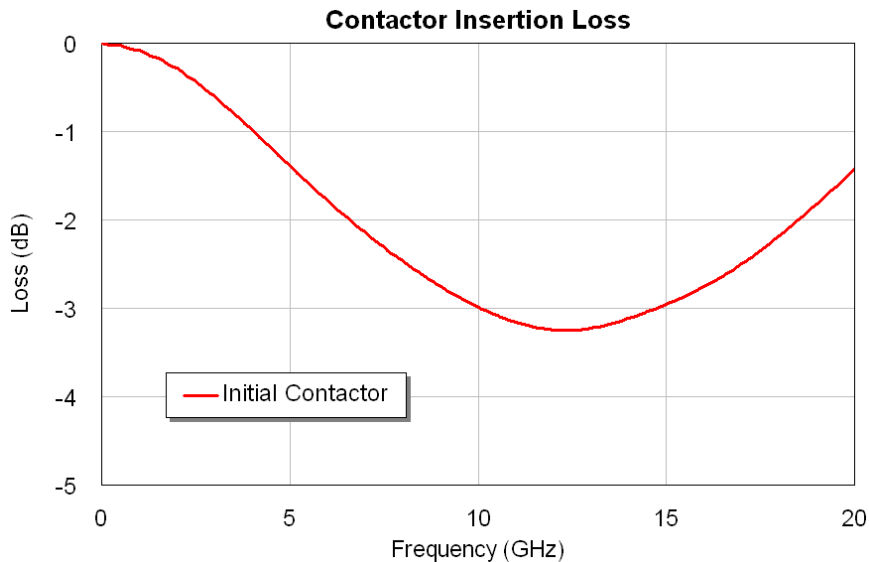
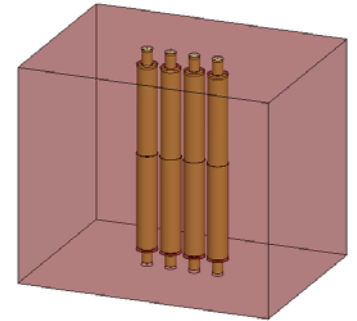
Redesign – Trace Z

- Trace Width Optimization
 - CPW Trace Width = 15.5mil; Sig-to-Gnd Spacing = 8mil
 - Differential Trace Width (under air) = 11mil
 - Differential Trace Width (under Vespel) = 10mil



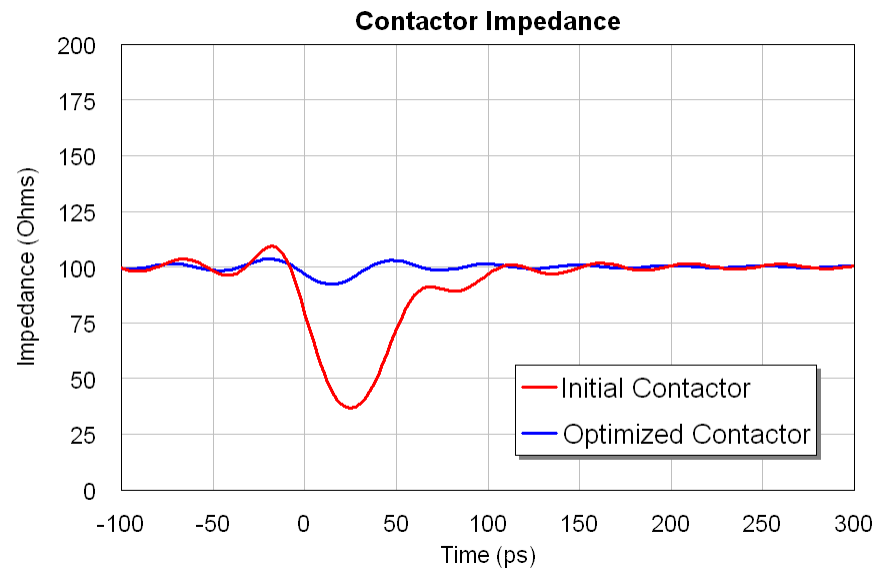
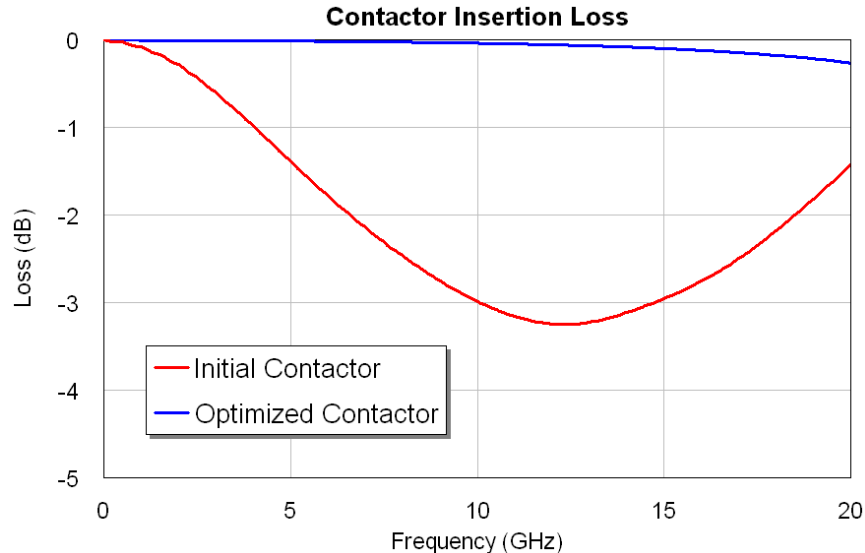
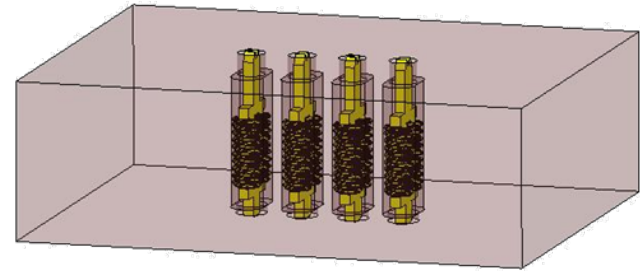
Original Design – Contactor

- Original Design – non-RF Contactor
- Performance unacceptable at 10GHz
 - Pin Length = 4.5mm; Pin Diameter = 0.4mm
 - Creates large capacitive mismatch at 0.5mm pitch



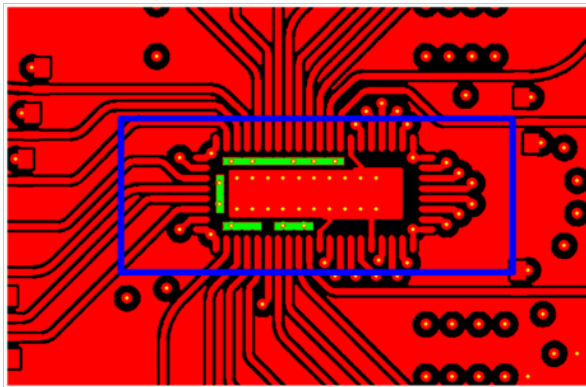
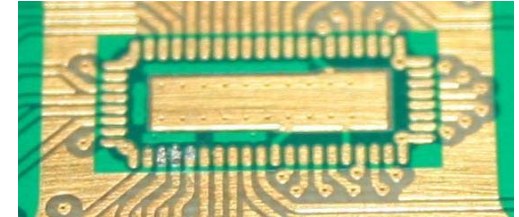
Redesign – Contactor

- Use RF Contactor
- GEM040
 - Pin Length = 1.54mm
 - Bandwidth (-1dB) = 40GHz

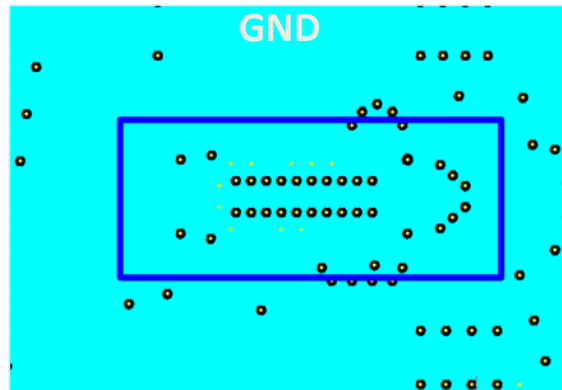


Original Design – Grounding

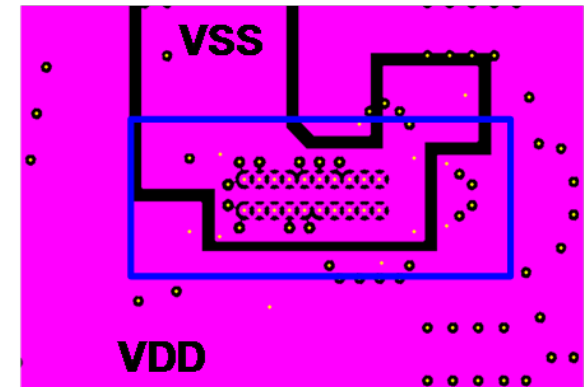
- Original Design – Poor grounding
 - Soldermask placed over only connections from DUT to GND
 - Two separate grounds: VSS (LYR07), GND (LYR02)
 - GND, VSS connected together through jumper on outside of PCB



LYR01 (TOP)



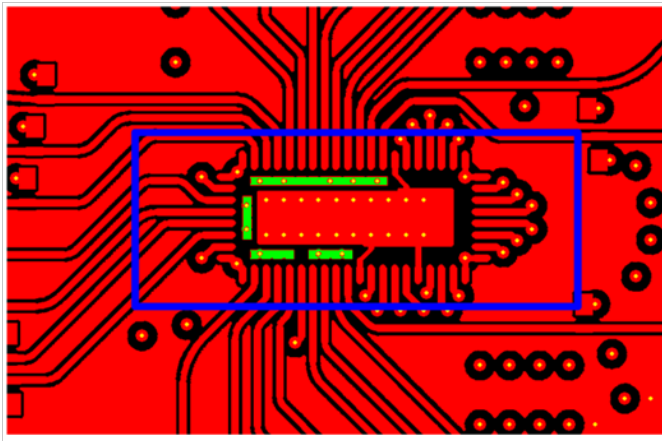
LYR02, 04, 06



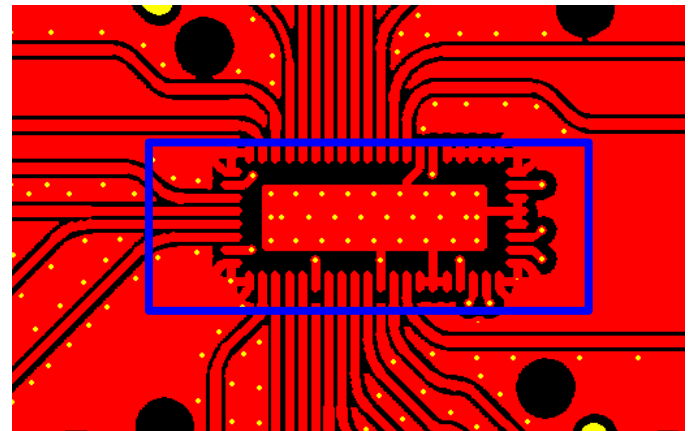
LYR07

Redesign – Grounding

- Soldermask removed on ground pads
 - Allows for probes to connected PCB GND to DUT GND
- Grounds merged into one
 - Provides continuity of ground path at DUT
 - Device did not require two grounds



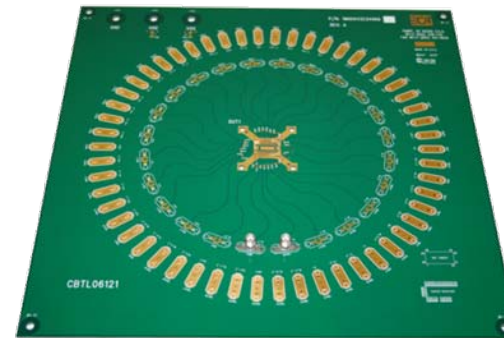
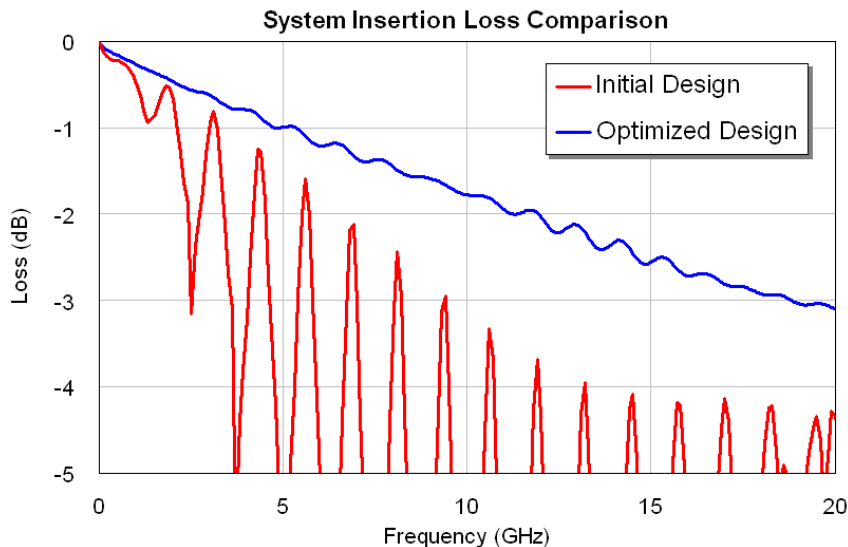
Original Design



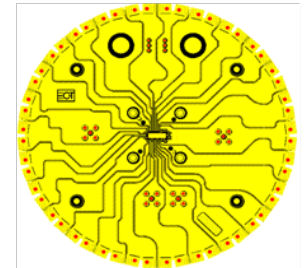
Board Redesign

Redesign – Final Results

- Bandwidth Results:
 - Original Design (-1dB) = 2.1 GHz; (-3dB) = 2.5 GHz
 - Board Redesign (-1dB) = 4.9 GHz; (-3dB) = 18.9 GHz



Original Design



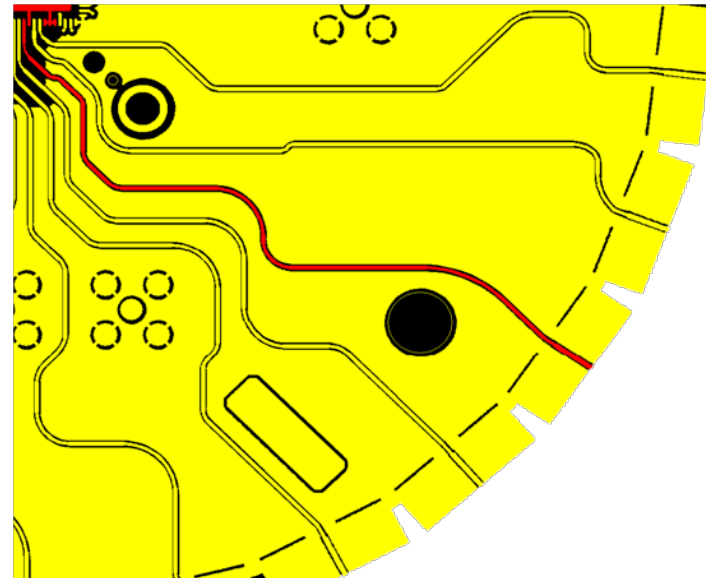
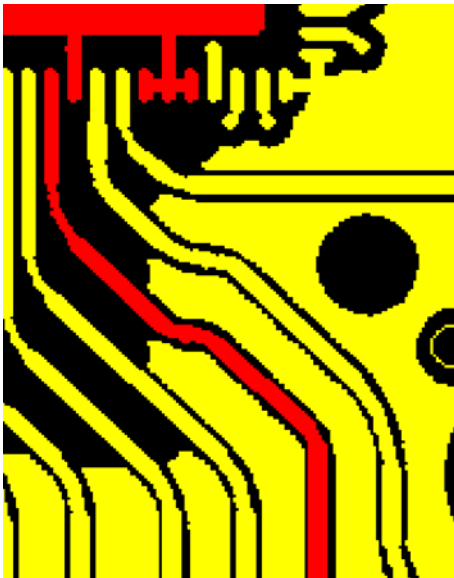
Board Redesign

Agenda

- This presentation describes the process of meeting a 10 GHz interface requirement:
 - Original design review and interface modifications
 - Simulation-to-measurement correlation

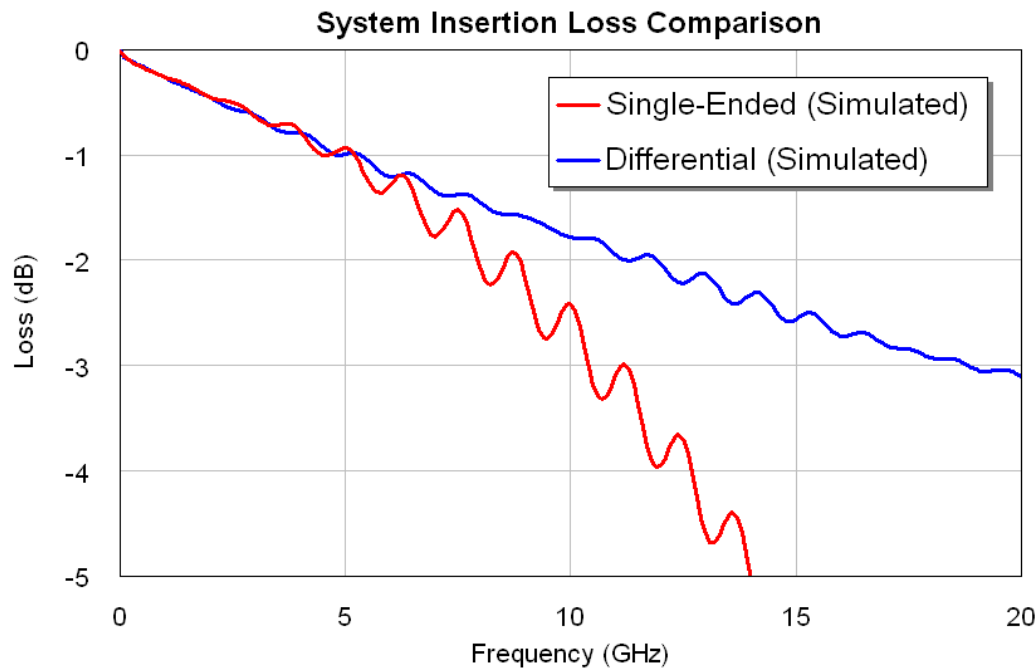
Results Correlation

- Correlation limitations
 - Measurement capabilities are limited to single-ended
 - Measured signal must have adjacent ground for probing



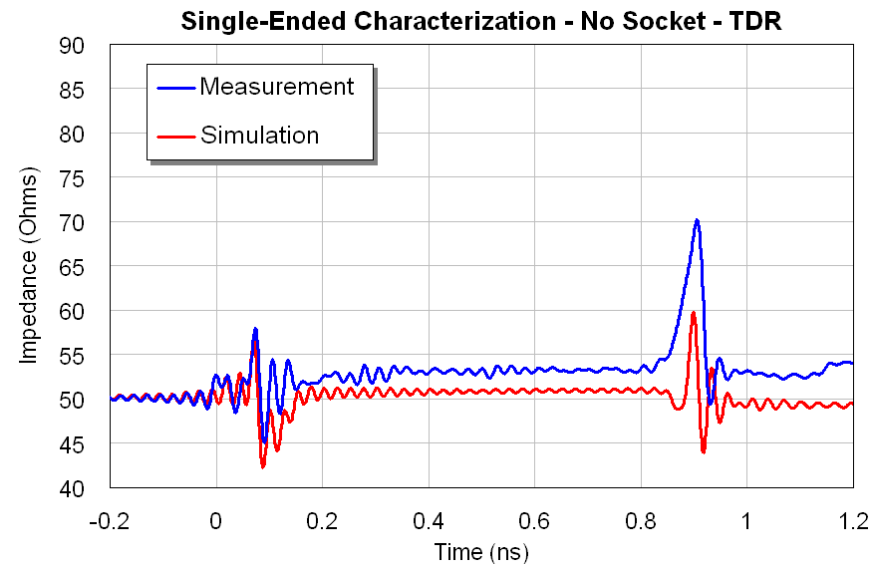
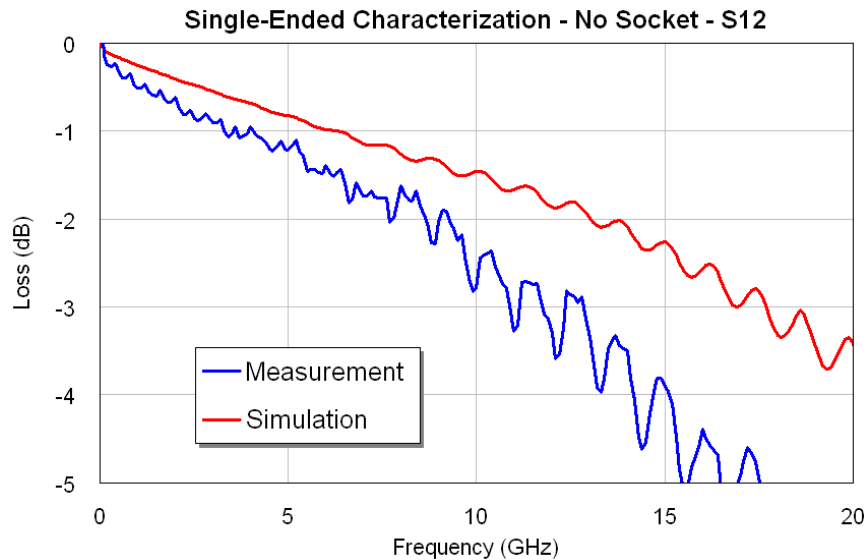
Results Correlation

- Measurements compared to single-ended simulation results
 - **BLUE** – Expected performance in application
 - **RED** – Expected correlation result



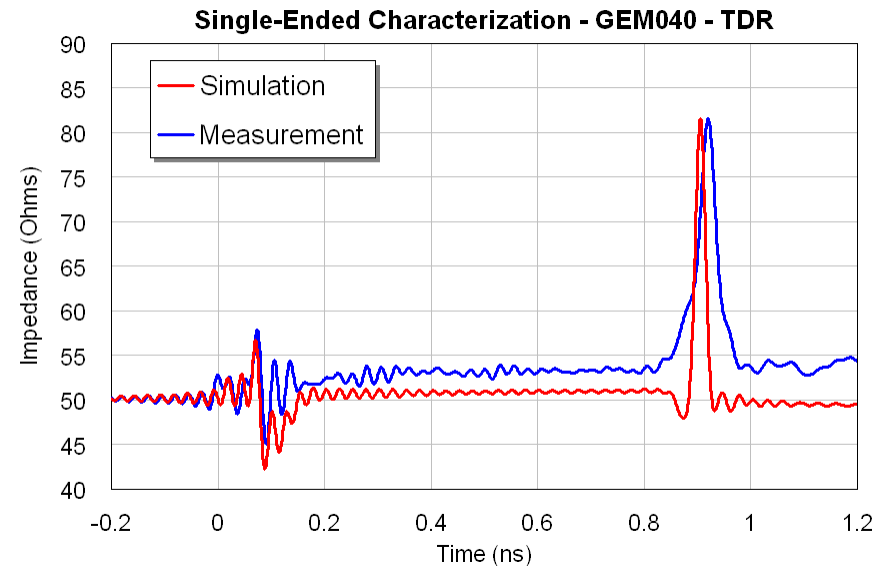
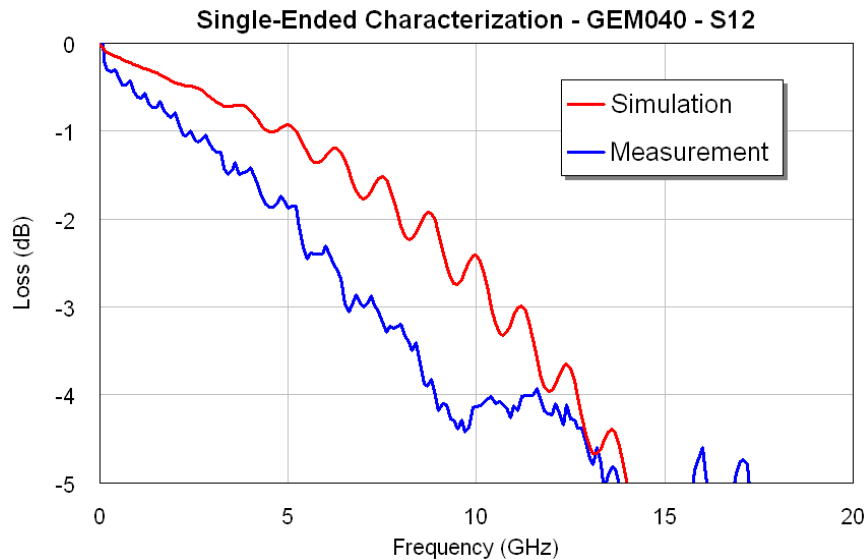
Results Correlation

- Results – No Socket
 - Path from SMA Connector to DUT (without a socket)



Results Correlation

- Results – with Socket
 - Path from SMA Connector to GEM socket



Conclusion

- Review of original design showed several shortcomings
- Limitations were overcome through 3D electromagnetic simulation
- Results were verified with measurement correlation
- This customer now incorporates simulation into the design process to achieve first pass success