multitest

A Xcerra Company

Silicon Valley TEST Conference & Expo San Jose Nov. 8-9, 2010 www.svtest.com

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

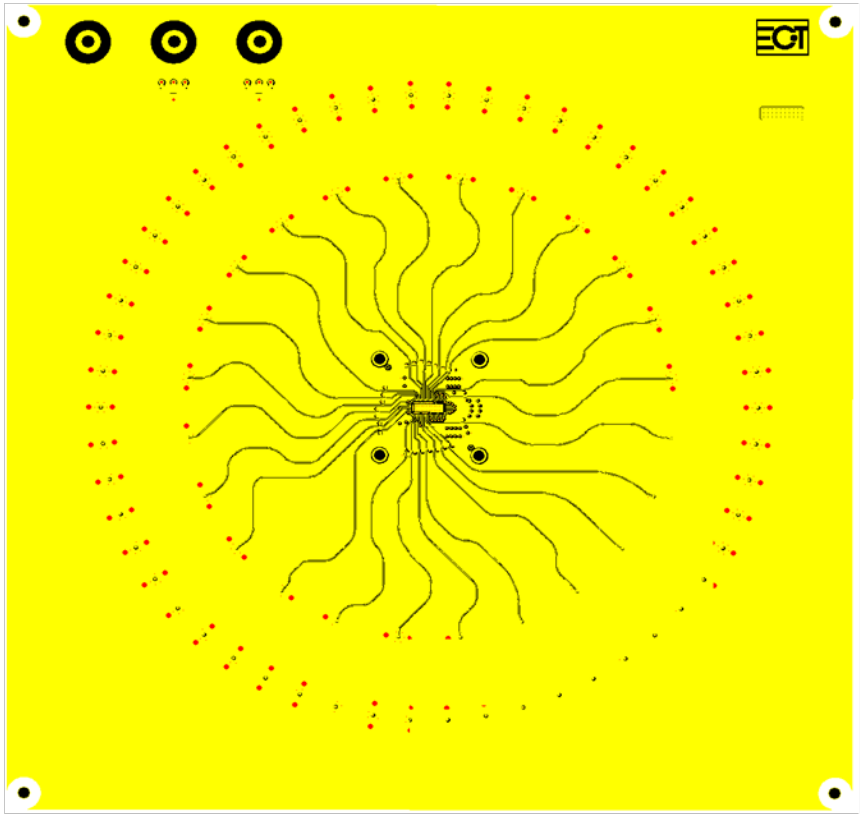
- This presentation describes the process of meeting a 10 GHz interface requirement:
  - Original design review and interface modifications
  - Simulation-to-measurement correlation

Structures Optimized

>Original design failed due to insufficient bandwidth, requiring a complete redesign

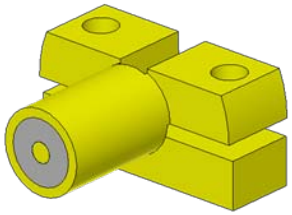
>Several structures were analyzed and modified to optimized performance:

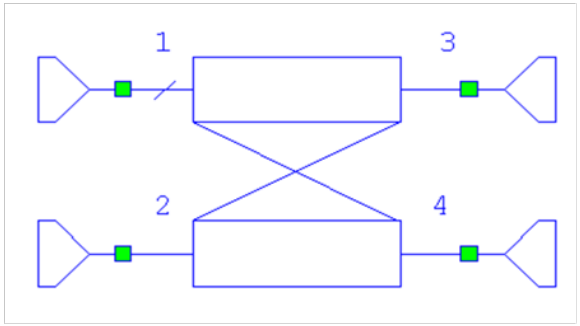
- >Connector
- >Trace
- >Contactor
- >Path Grounding

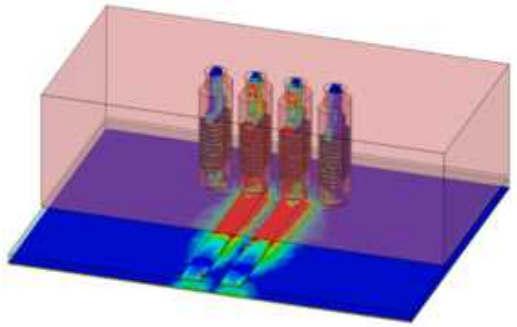


Structures Optimized

- 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

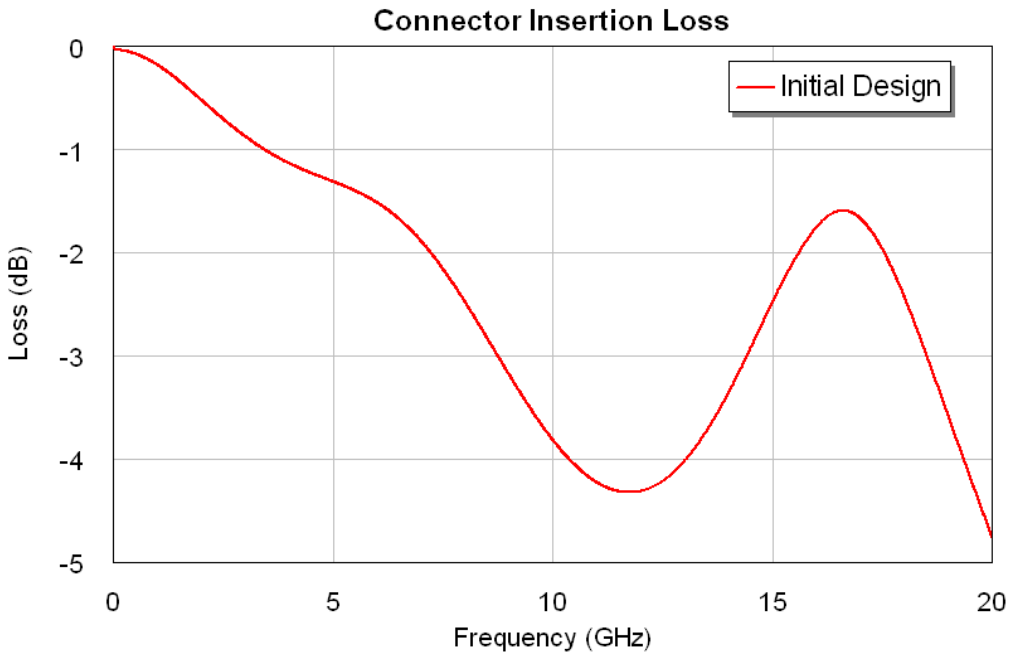


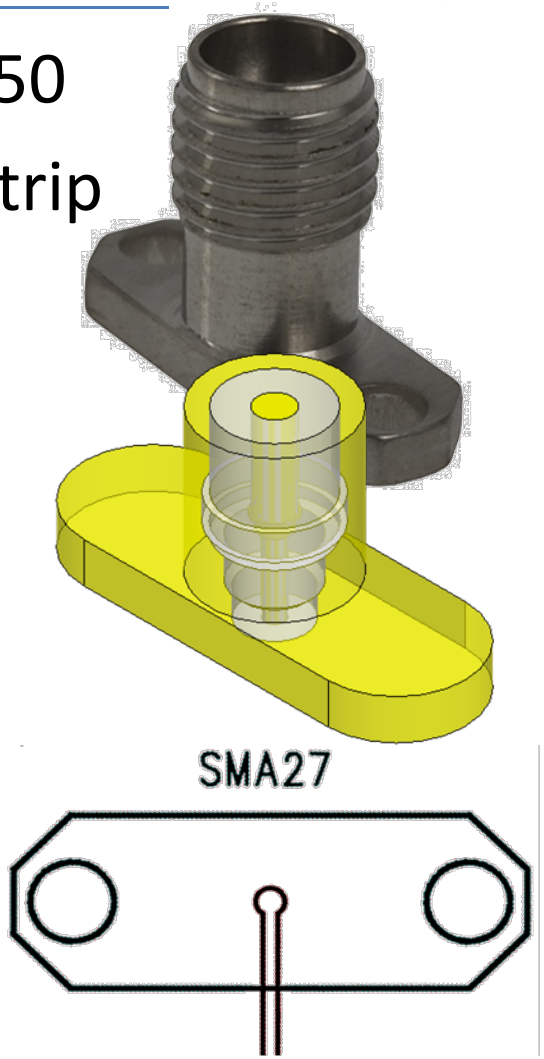




Original Design - Connector

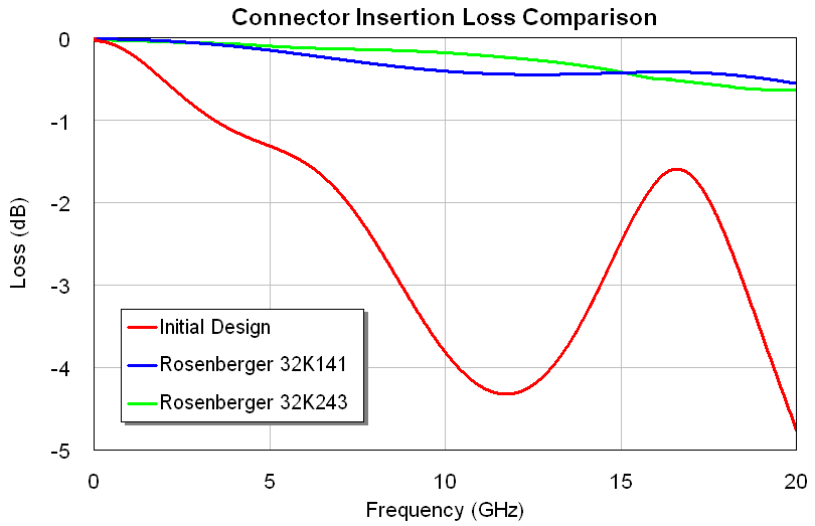
- Original Connector – Molex 73251-1850
- Poor high-speed connector for microstrip
- Footprint not optimized

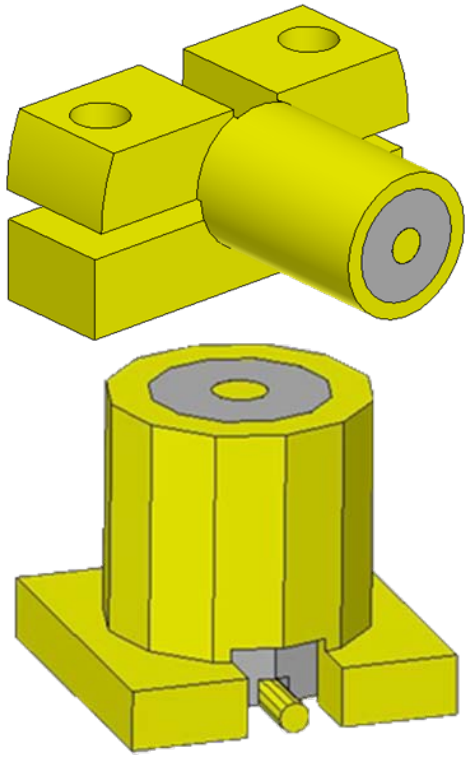




Redesign - Connector

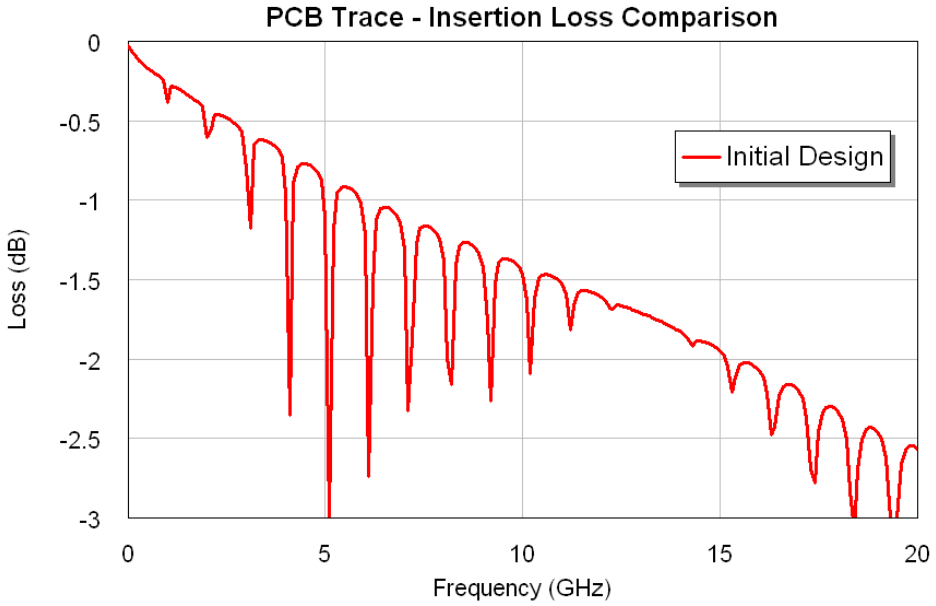
- Optimized connector possibilities
  - Rosenberger 32K243, Rosenberger 32K141
- Connector footprints optimized through simulation
- Rosenberger 32K243 chosen

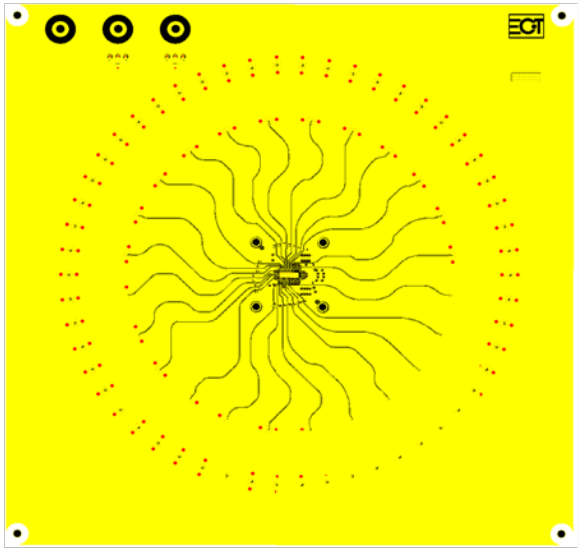




Original Design – Trace Vias

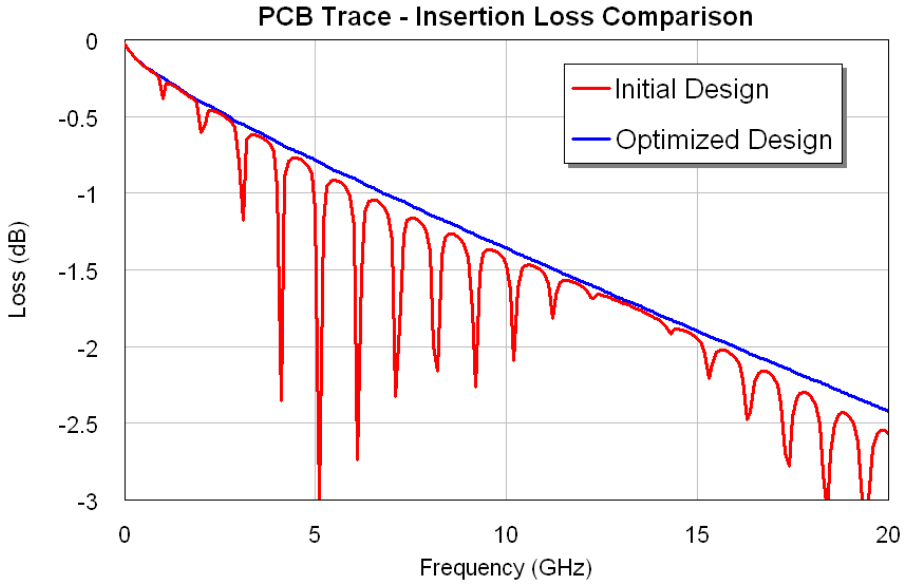
- Original design – ground via locations arbitrary
  - Coplanar waveguide (CPW) traces require vias in close proximity
  - CPW traces resonate if not properly designed
  - Resonances must be removed





Redesign – Trace Vias

- Coplanar waveguide traces
  - CPW traces resonate at ½-wavelength of distance between vias
  - 100mil ground via spacing (no resonances until ~30GHz)

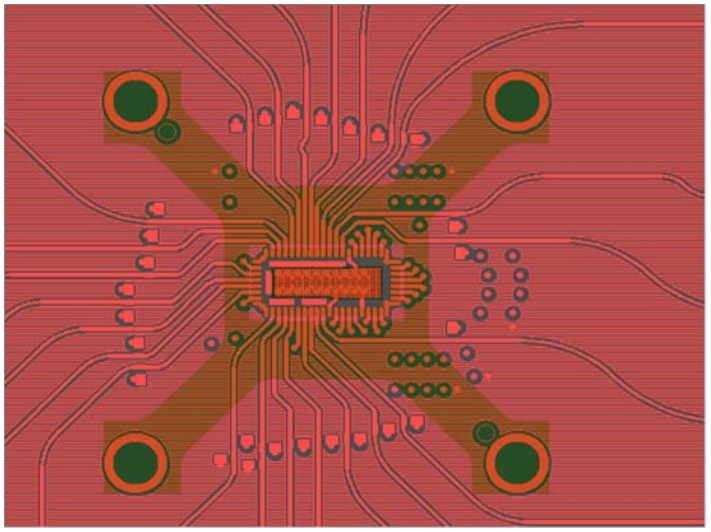


$$f_{RES} = \frac{c}{2\lambda\sqrt{\varepsilon}}$$

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

Original Design – Trace Z

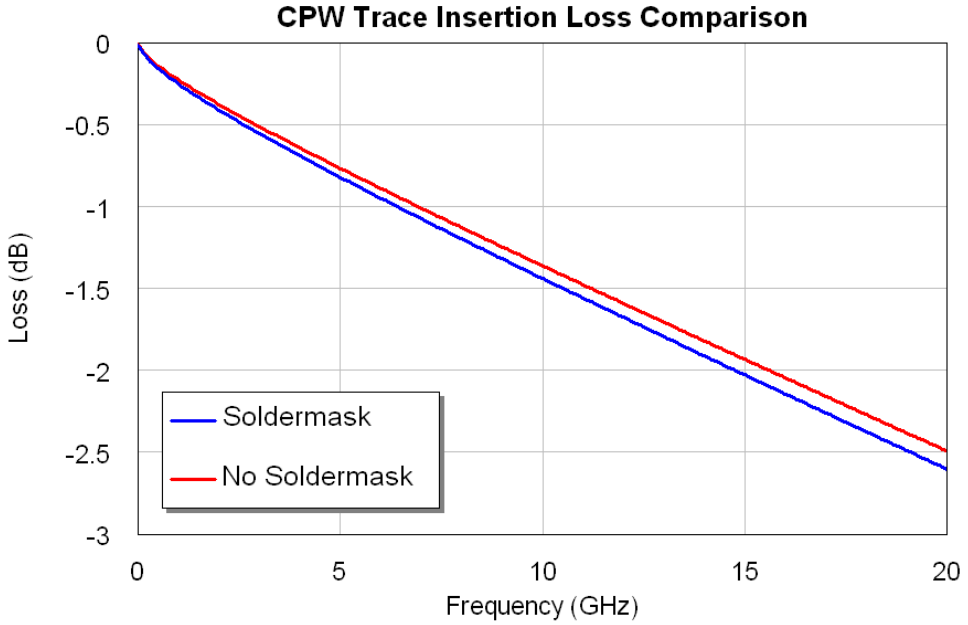
- Original Design – Trace width not adjusted as path changes
- Slight changes in trace geometry can have performance impact at multi-GHz frequencies

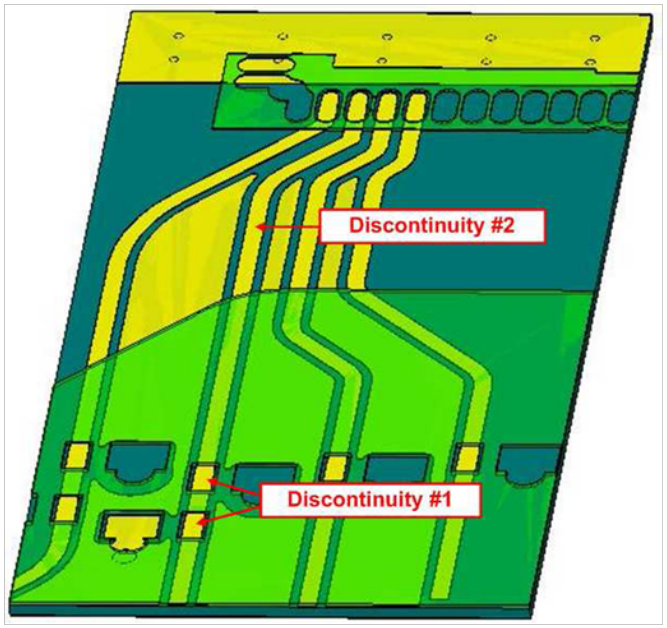


- Factors that may impact impedance
  - Soldermask
  - CPW coupling
  - Trace-to-Trace coupling
  - Trace traveling under contactor

Redesign – Trace Z

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

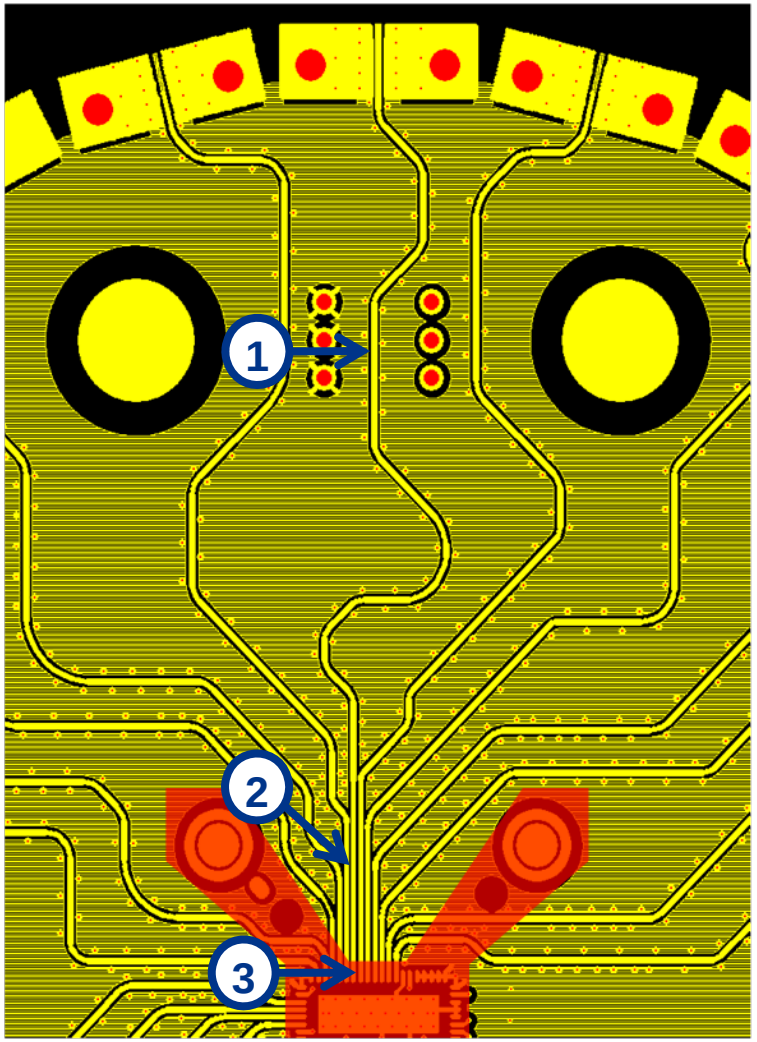




Redesign – Trace Z

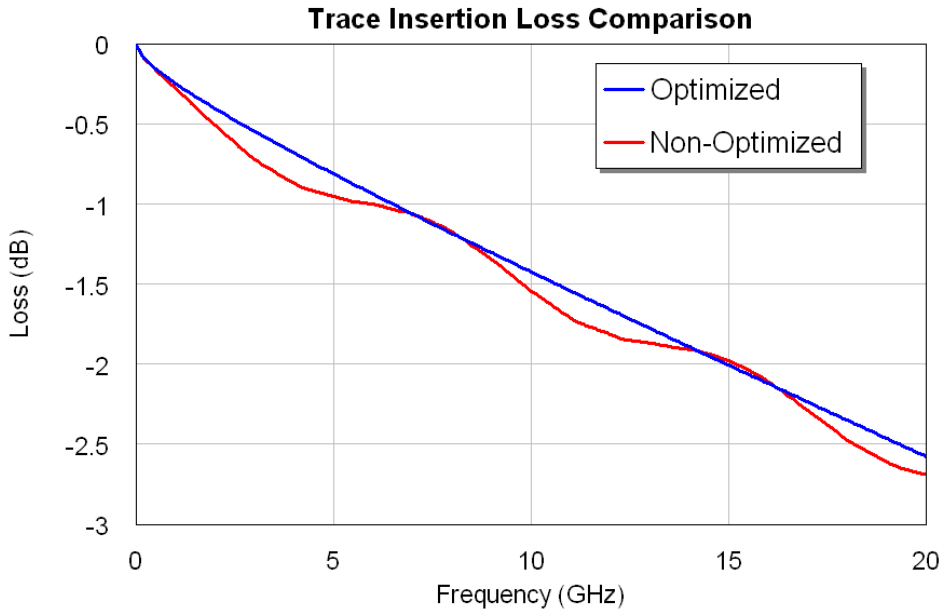
- Trace Width Optimization

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



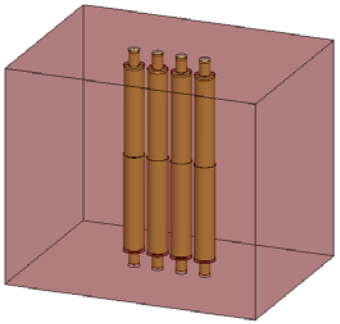
Redesign – Trace Z

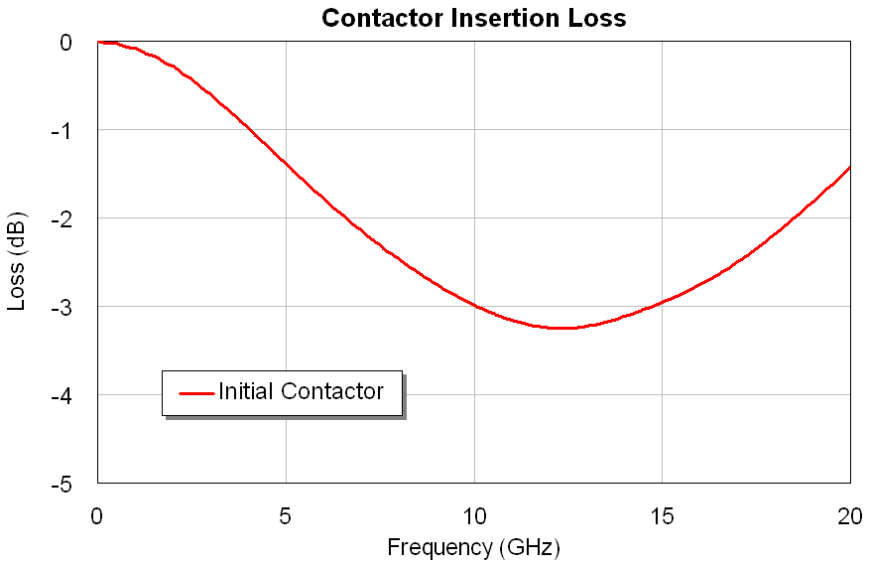
- Trace Width Optimization
  - CPW Trace Width = 15.5mil; Sig-to-Gnd Spacing = 8mil
  - Differential Trace Width (under air) = 11mil
  - Differential Trace Width (under Vespel) = 10mil

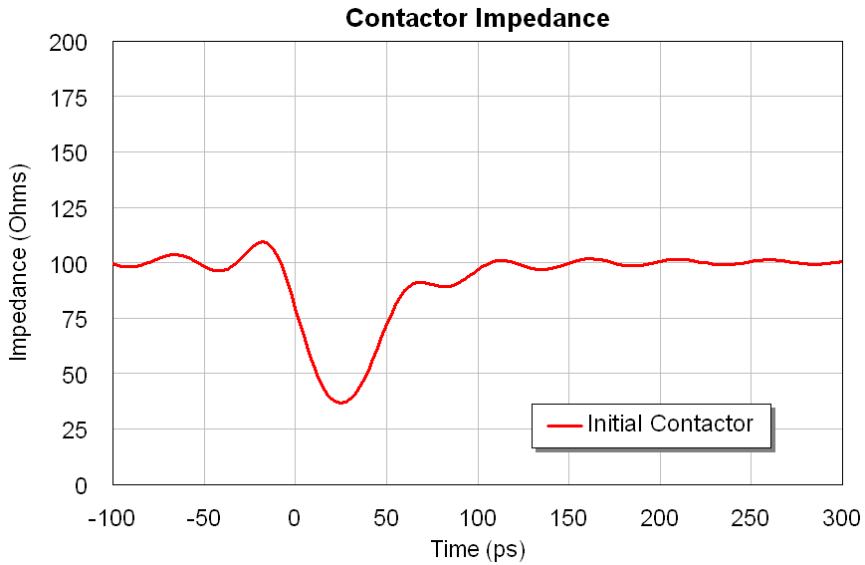


Original Design – Contactor

- 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

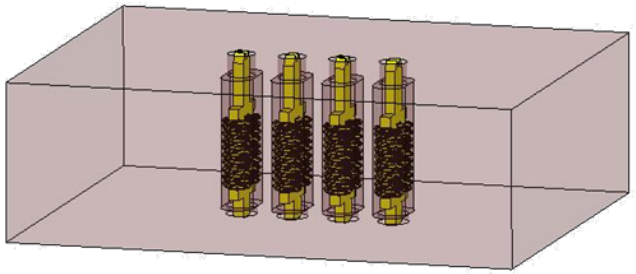


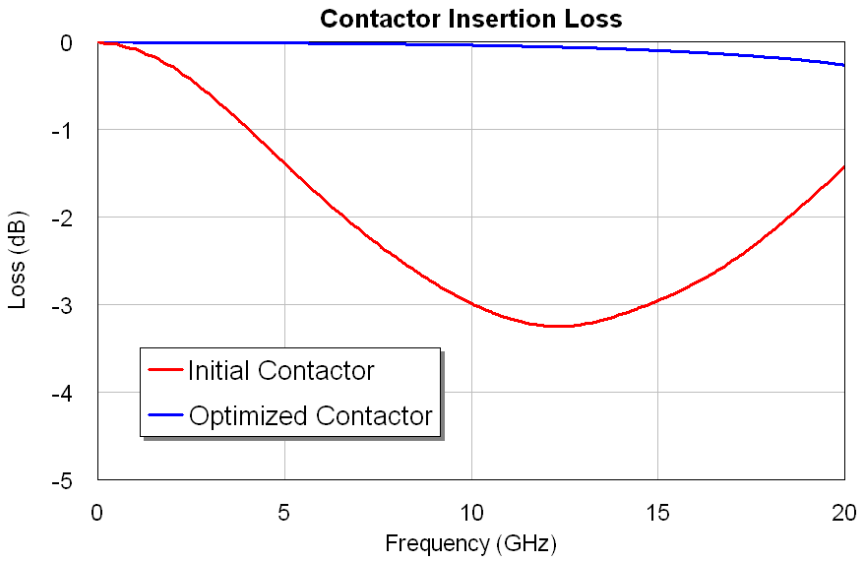


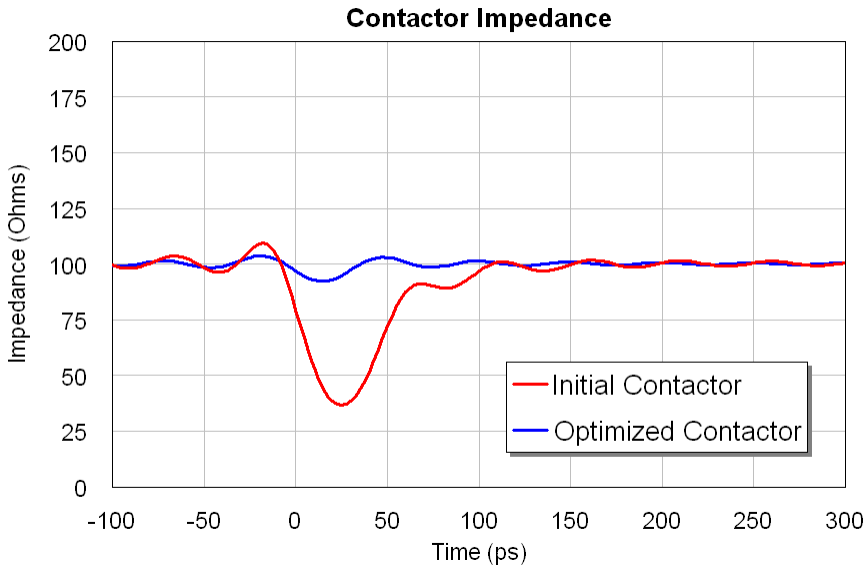


Redesign – Contactor

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

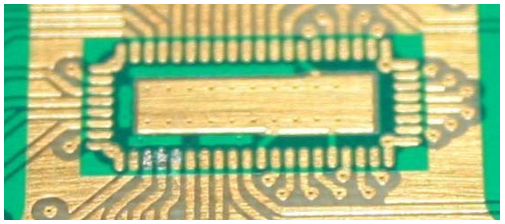


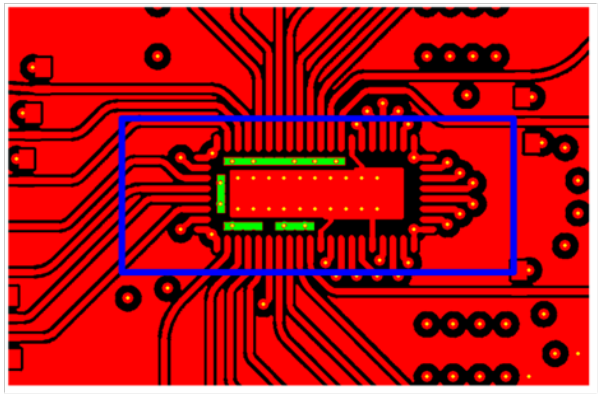




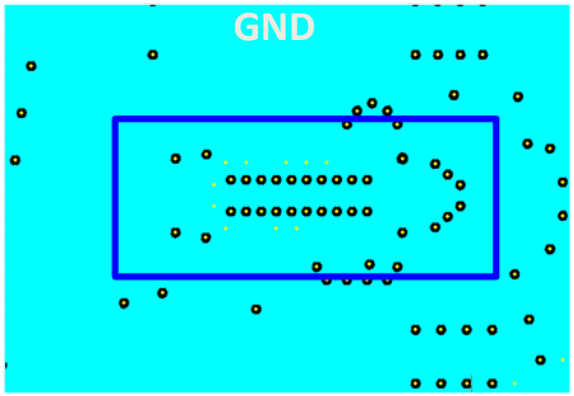
Original Design – Grounding

- 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

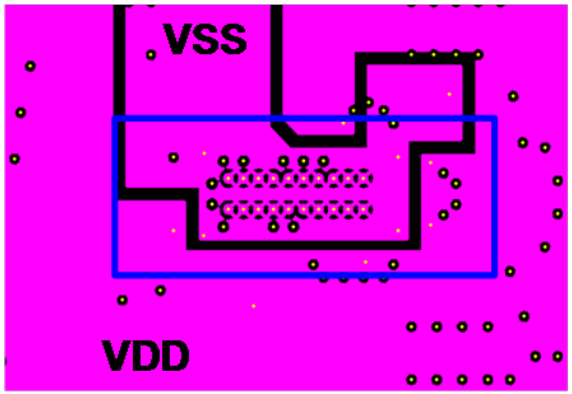




LYR01 (TOP)



LYR02, 04, 06

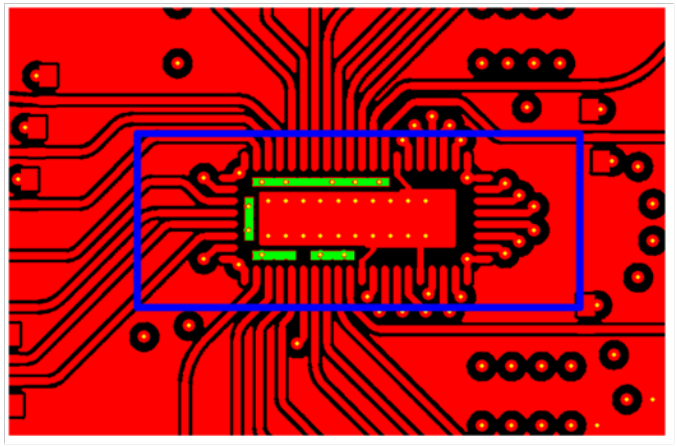


LYR07

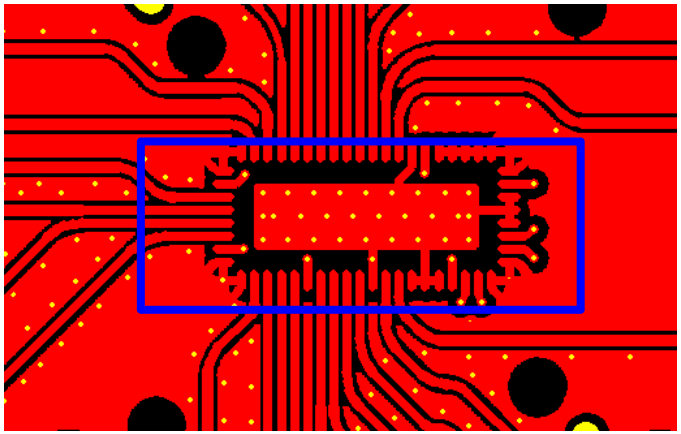
***Board is 8 layers**

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



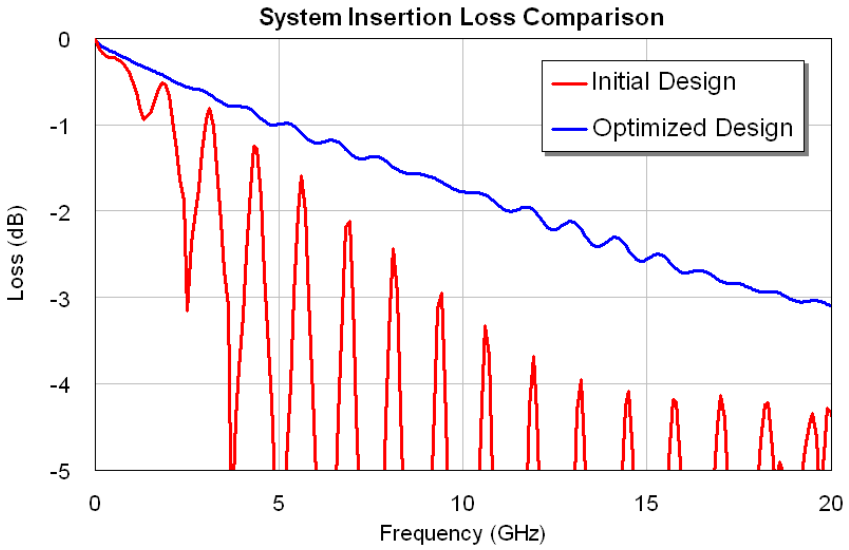
Original Design

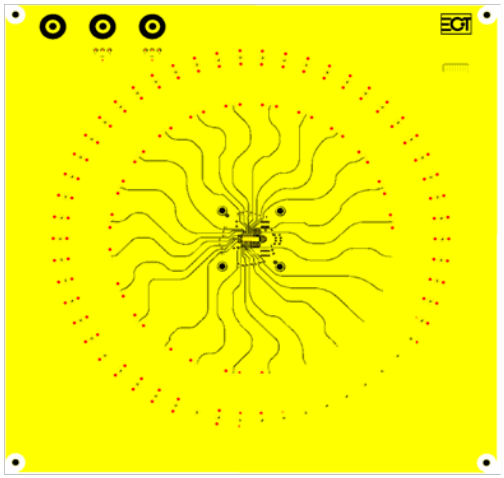


Board Redesign

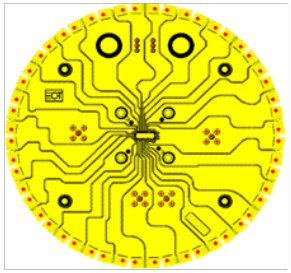
Redesign – Final Results

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





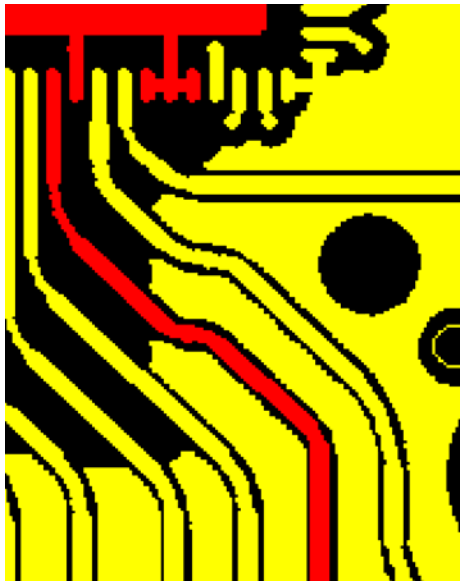
Original Design

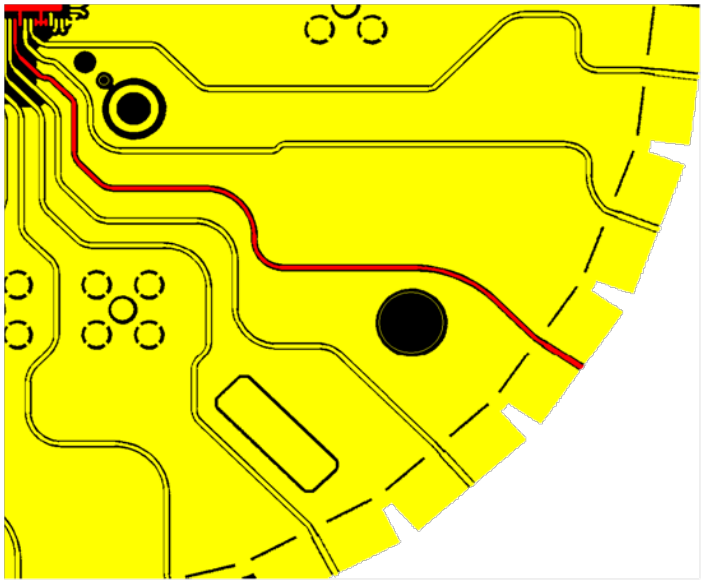


Board Redesign

Results Correlation

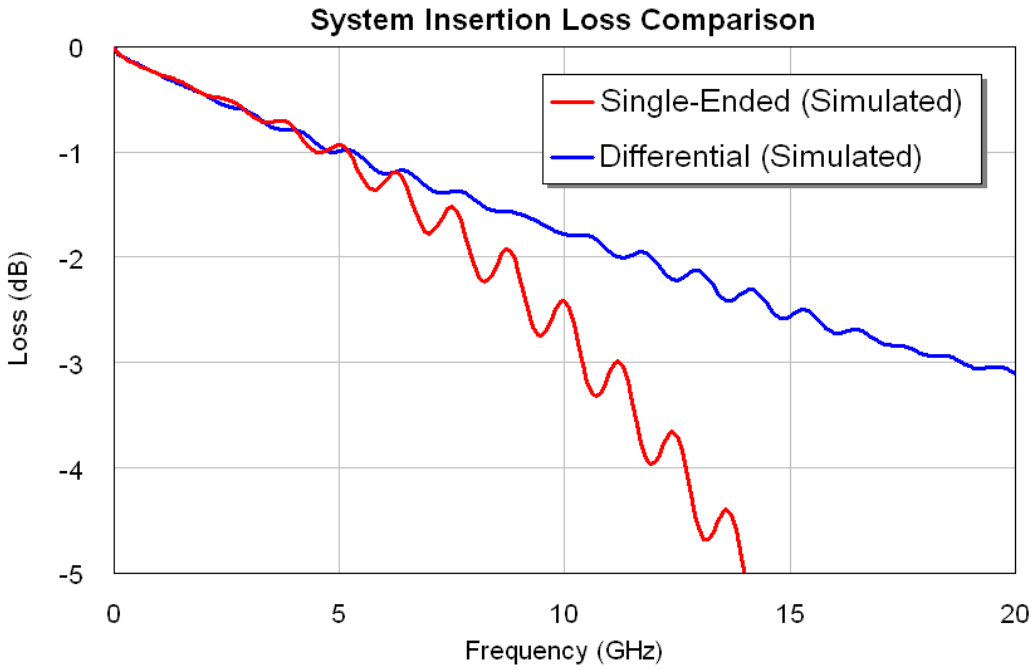
- Correlation limitations
  - Measurement capabilities are limited to single-ended
  - Measured signal must have adjacent ground for probing





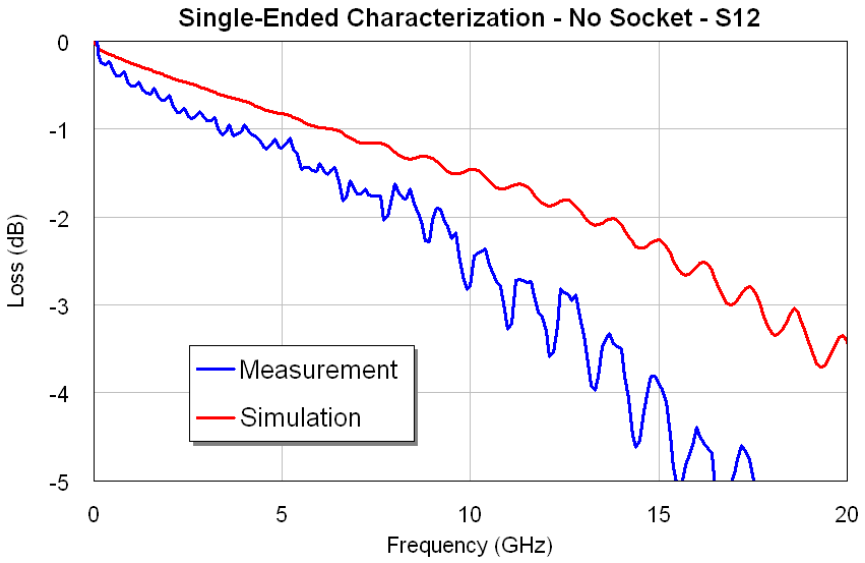
Results Correlation

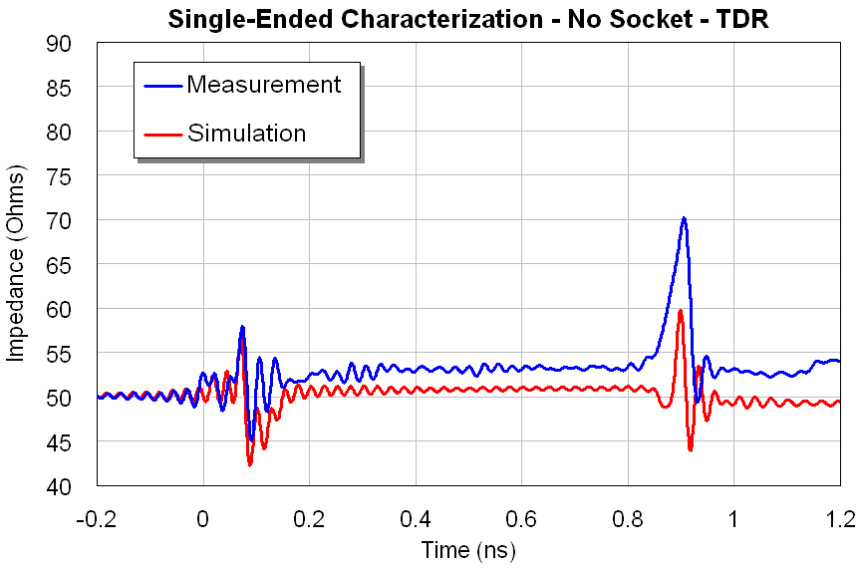
- Measurements compared to single-ended simulation results
  - **BLUE** – Expected performance in application
  - **RED** – Expected correlation result



Results Correlation

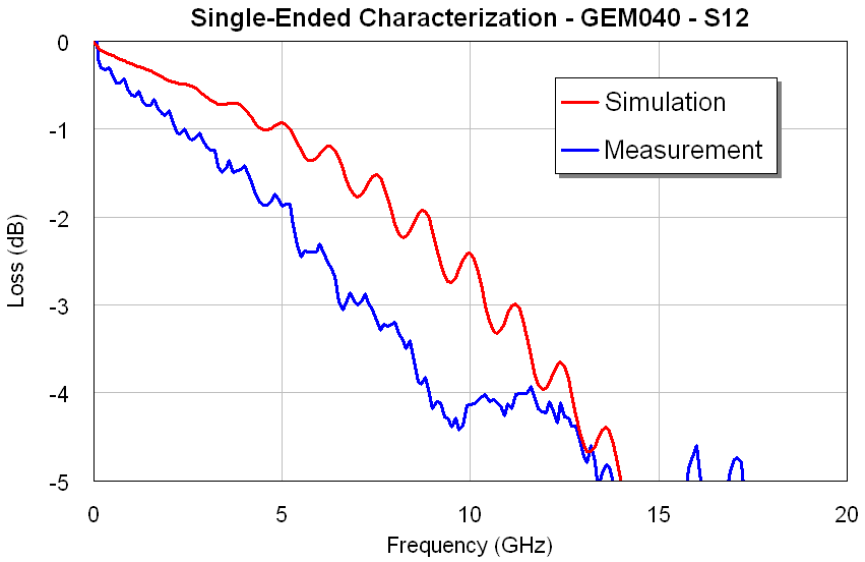
- Results – No Socket
  - Path from SMA Connector to DUT (without a socket)

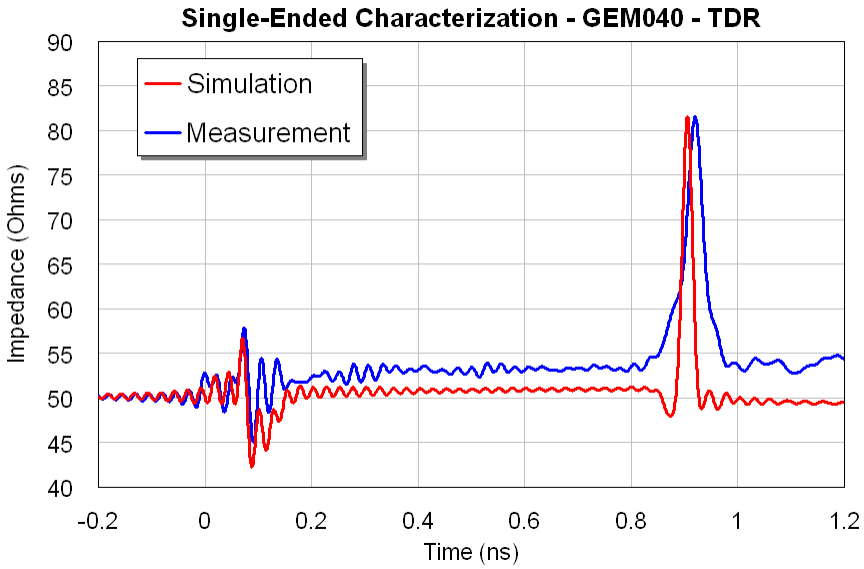




Results Correlation

- 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