TWENTY-FOURTH ANNUAL

TestConX™

Archive

DoubleTree by Hilton
Mesa, Arizona
March 5-8, 2023

With Thanks to Our Sponsors!

Premier

HiCon

Honored

GLOBAL INNOVATION ISC

WIN SOLUTION

LEENO Total Interface Solutions

TSE

With Thanks to Our Sponsors!

Distinguished

Johnstech™

smiths interconnect

MICRO CONTROL COMPANY SINCE 1972

TFE Align & Smart Contact

ADVANTEST®

Industry Partners

SWTEST

MEPTEC THE NEXT GENERATION

COPYRIGHT NOTICE

www.testconx.org

Into the PCB at 90 – high GHz signal launches

Gert Hohenwarter

GateWave Northern, Inc.

TestConX

Mesa, Arizona • March 5–8, 2023

GateWave Northern, Inc.

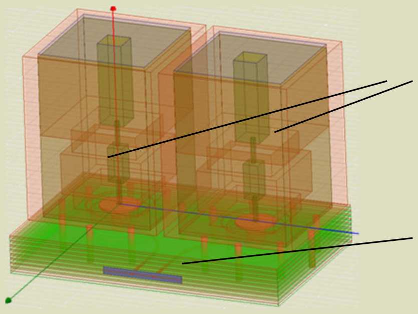
1

Problem

- High speed coaxial cable connectors and sockets for 5G and automotive radar circuits require adjustment of PCB parameters for (optimal) functionality.
- Interconnect architecture must be carefully defined and examined to avoid spurious resonances.

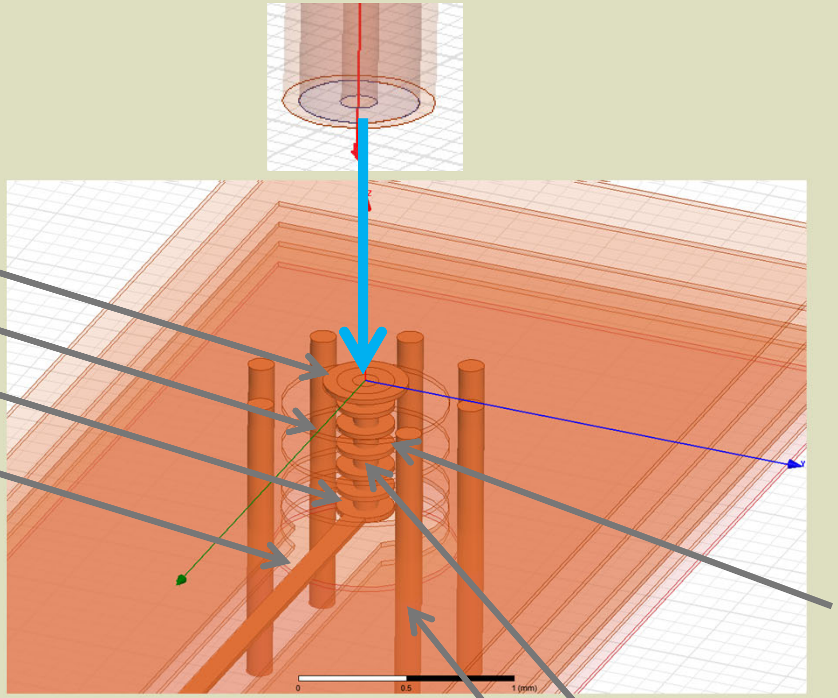
Approach

- Architecture example:

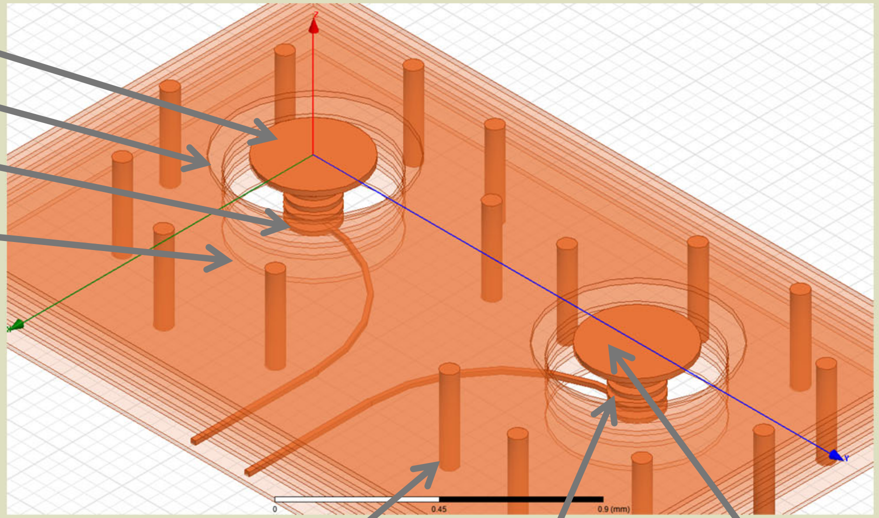


- Introduce problem set of launching into PCB
- Define parameters of interest
- Present results for various parameter changes
- Examine pitfalls and problem scenarios
- Outlook, comments and conclusions

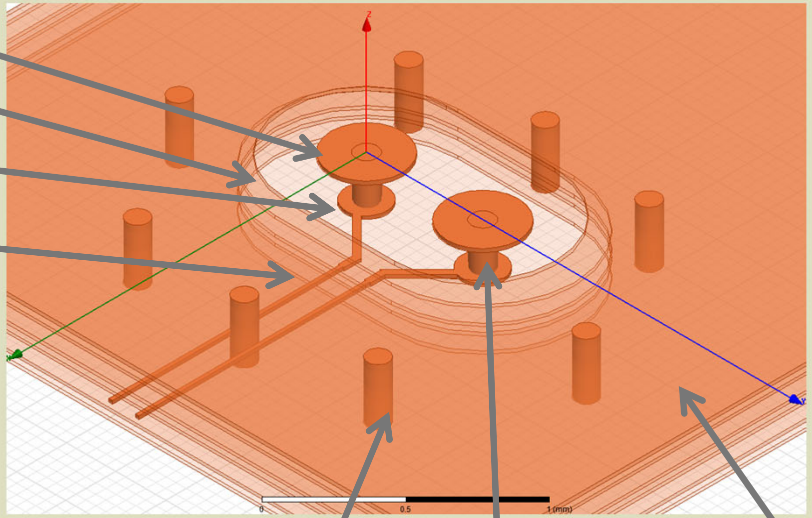
Single signal line feed



Differential/single signal line feed



Fully differential signal line feed



Relevant Parameters

Under control of designer (with limits):

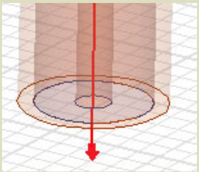
- Contact pad diameter
- Capture pad diameter
- Signal/ground via diameters
- Antipad diameters
- Ground via bolt circle diameter

Potentially limited/no control:

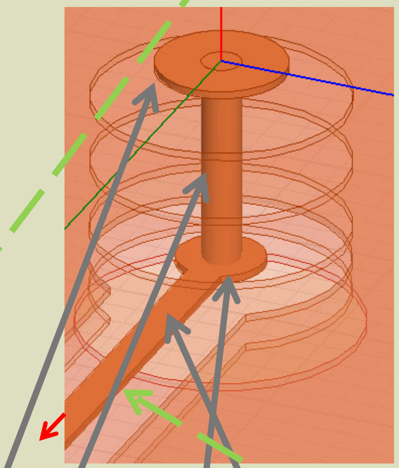
- PCB stackup thicknesses
- PCB materials

Visualization

SPICE simulations will be used instead of FEA to curtail the amount of computational effort for this presentation. For actual optimizations full 3D FEA must be used.

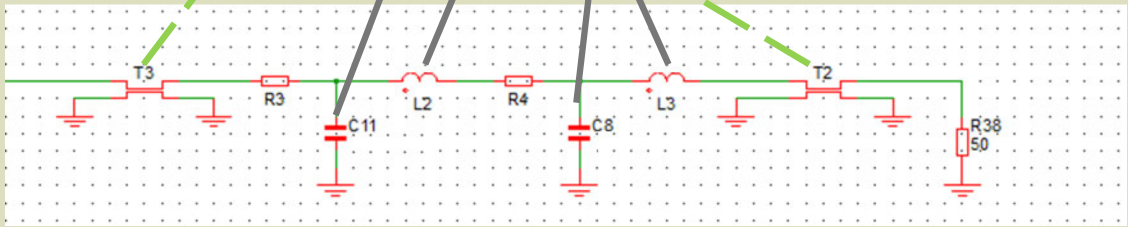


Physical circuit:



Since a good coaxial connector will add few contributions, it is omitted for the PCB examination. To optimize a design for a particular ***connector****, it* ***must be included in the model****.*

Equivalent circuit: (representative of actual conditions)



Simulated cases

Vary dimensions:

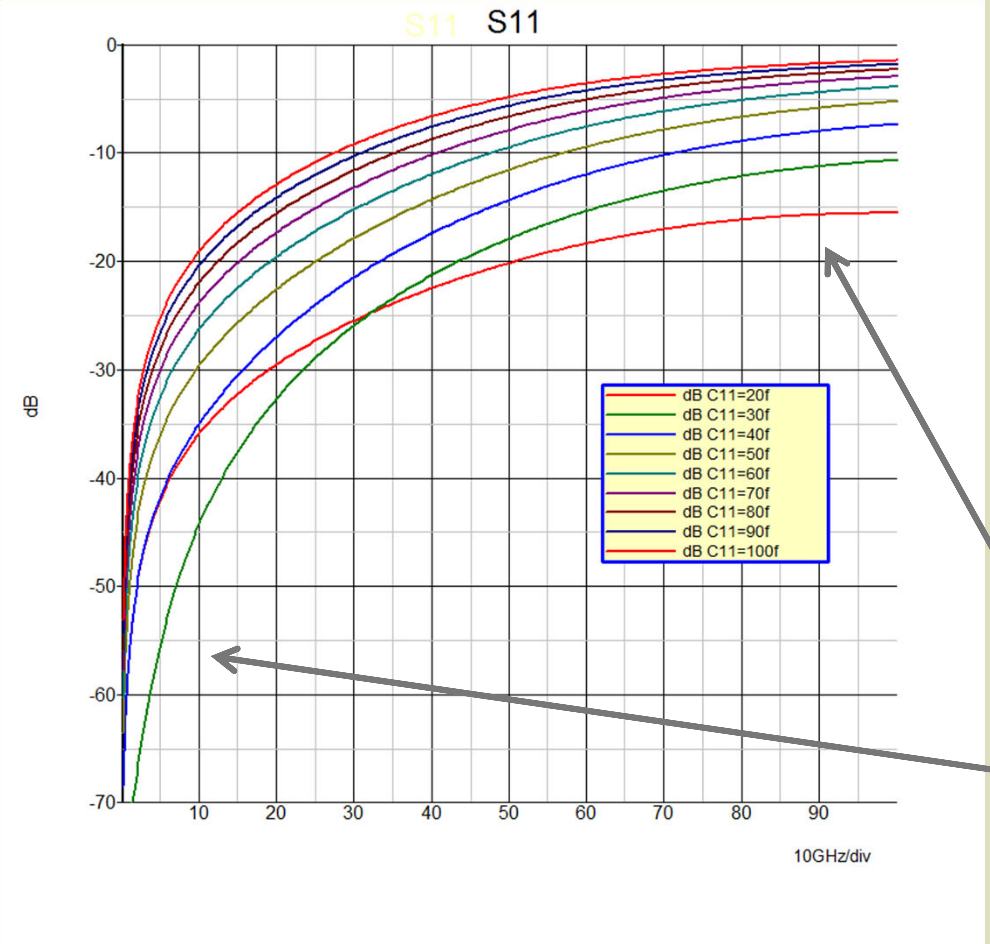
- Contact pad diameter
- Capture pad diameter
- Signal via diameter
- Antipad diameters
- Ground via bolt circle diameter

Potentially limited / no control:

- PCB stackup thicknesses
- PCB materials
- Counterdrill depth

Note:
All presented solutions are case specific. While trends can be established, it is not possible to generalize answers.

TestConX 2023



Return loss

Adjusted:

Contact pad diameter (each curve corresponds to a specific diameter)

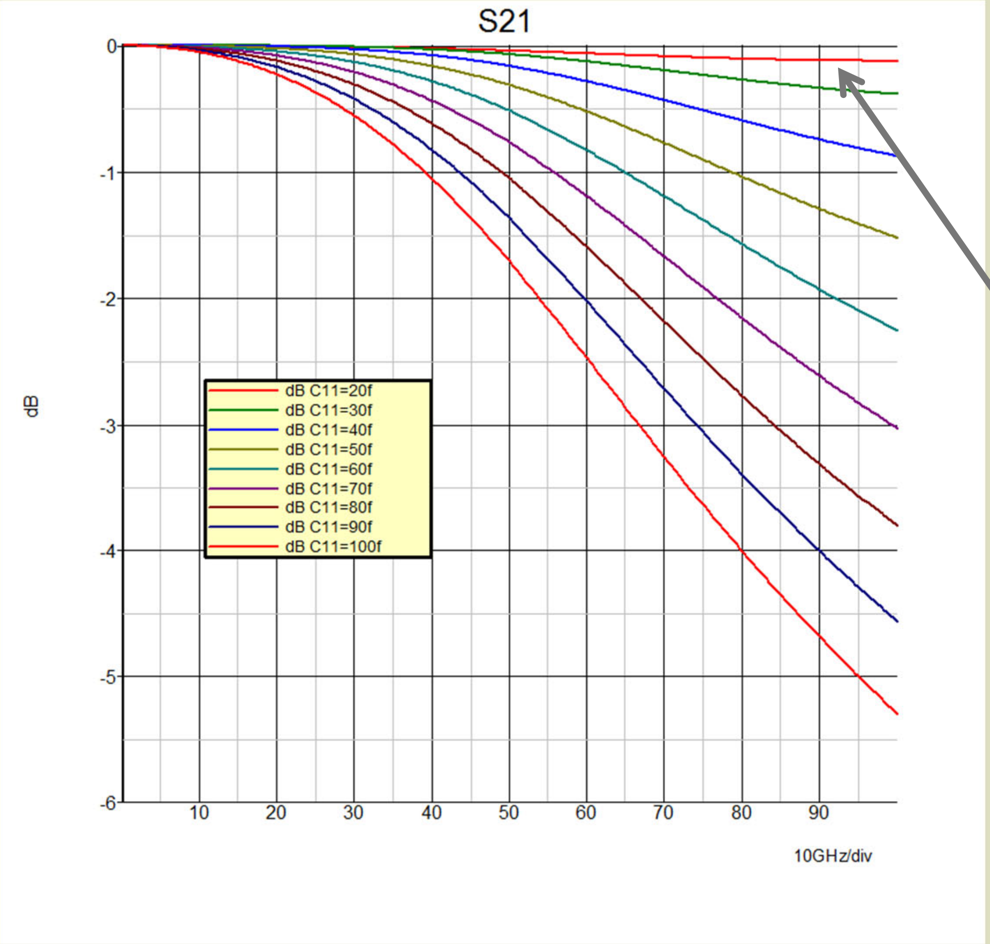
- Small pad gives best performance at high frequencies

- At low frequencies a slightly larger pad is optimal

Insertion loss

Adjusted:

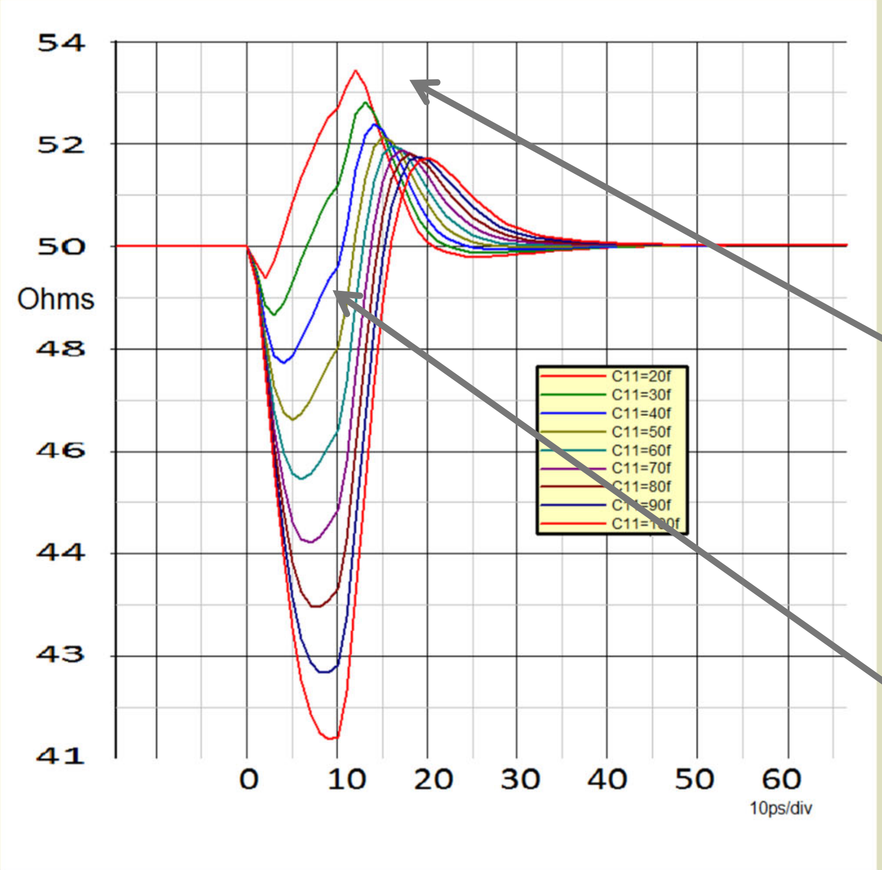
Contact pad diameter



Small pad gives best performance at high frequencies

TestConX 2023

TDR



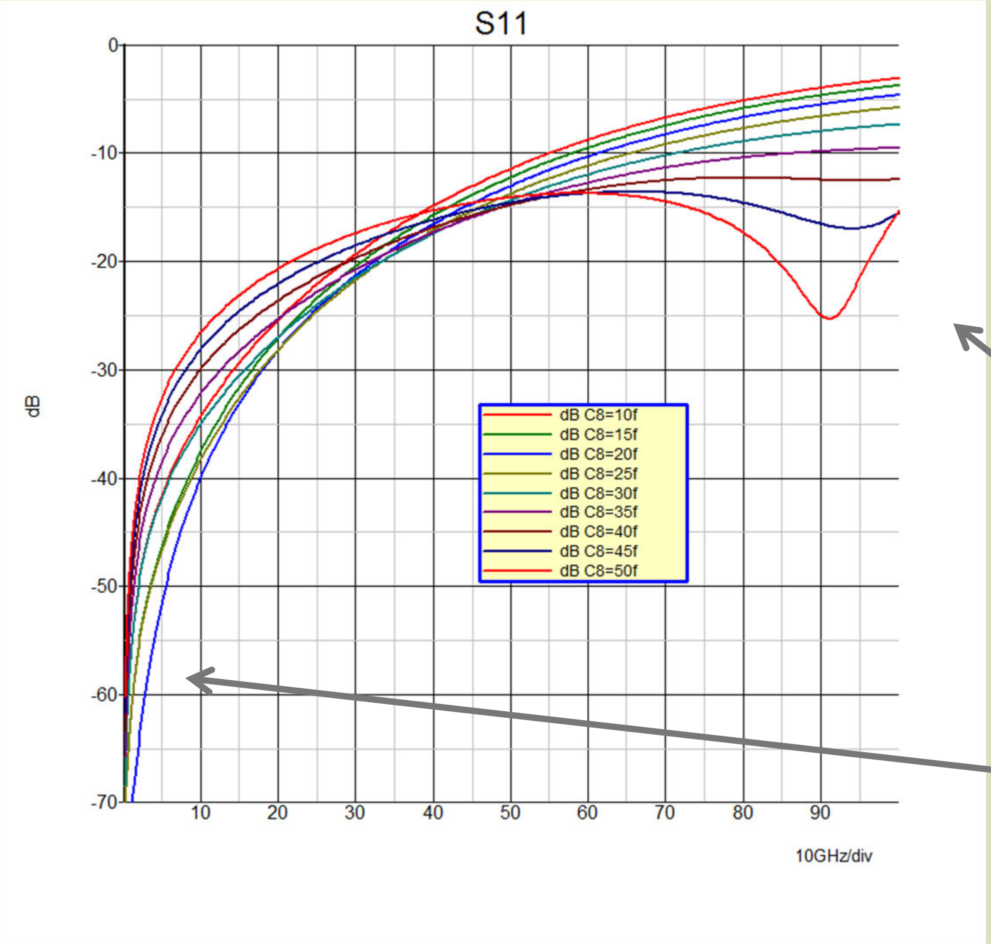
Adjusted:

Contact pad diameter

- Small pad gives higher impedance peak

- At low frequencies a slightly larger pad provides better balance

Return loss



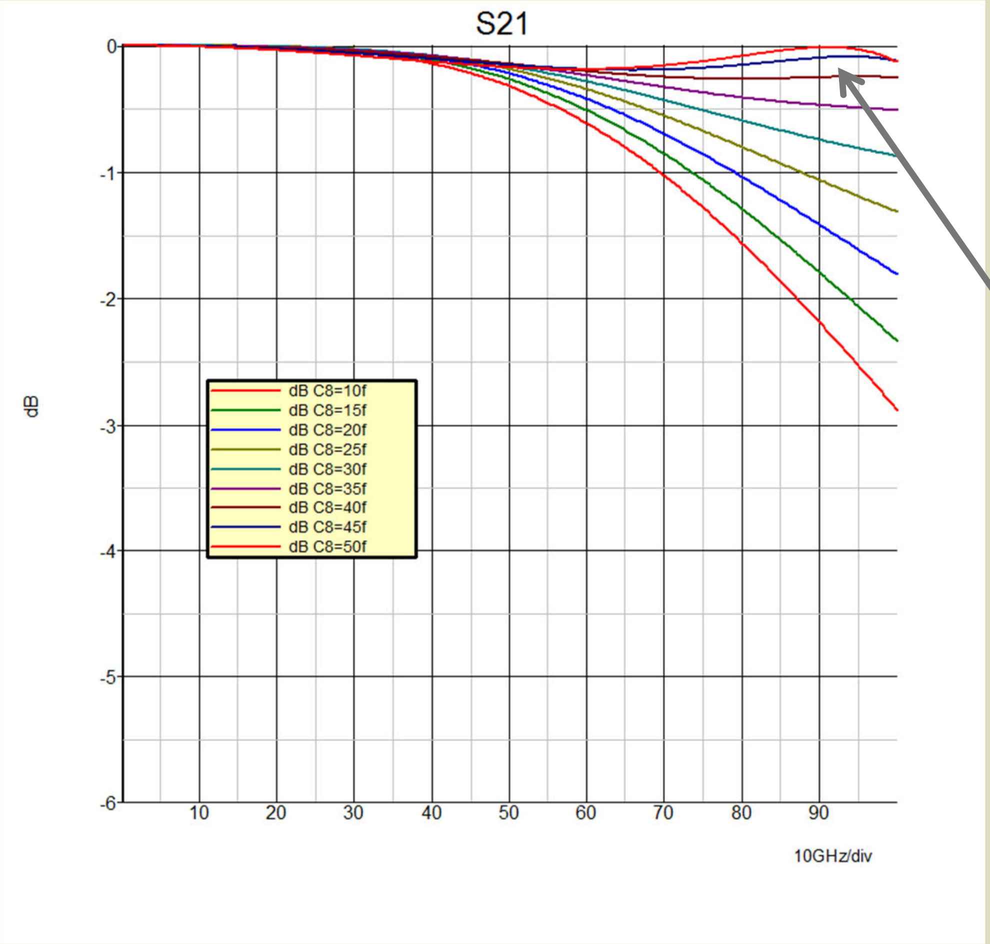
Adjusted:

Capture pad diameter

- Small pad gives best performance at high frequencies

- At low frequencies a slightly larger pad is optimal

Insertion loss

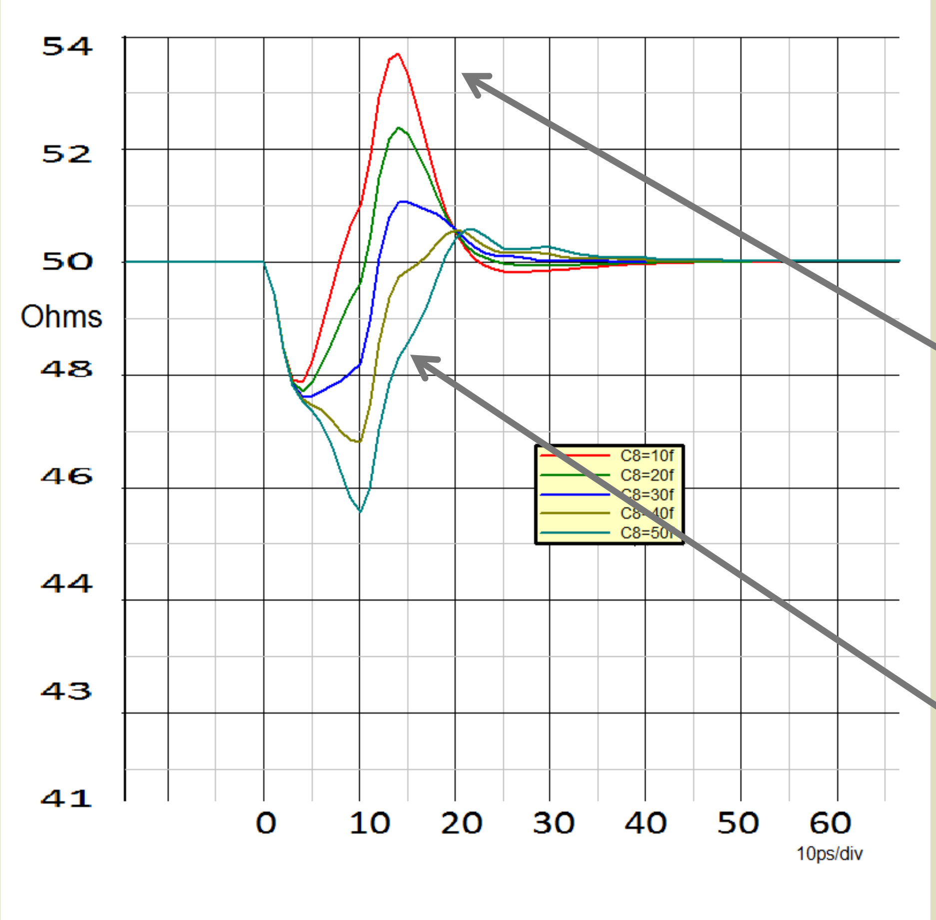


Adjusted:

Capture pad diameter

- Small pad gives best performance at high frequencies

TDR



Adjusted:

Capture pad diameter

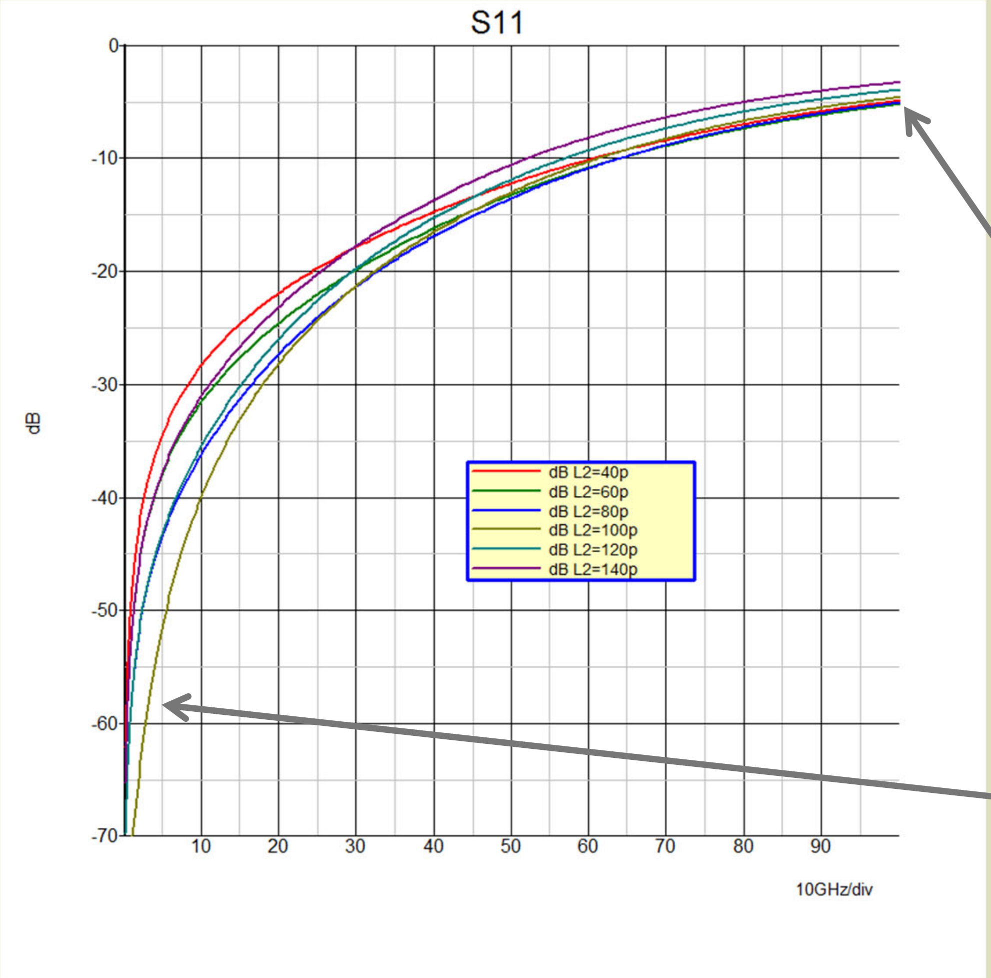
- Small pad gives higher impedance peak

- At low frequencies a slightly larger pad provides better balance

Return loss

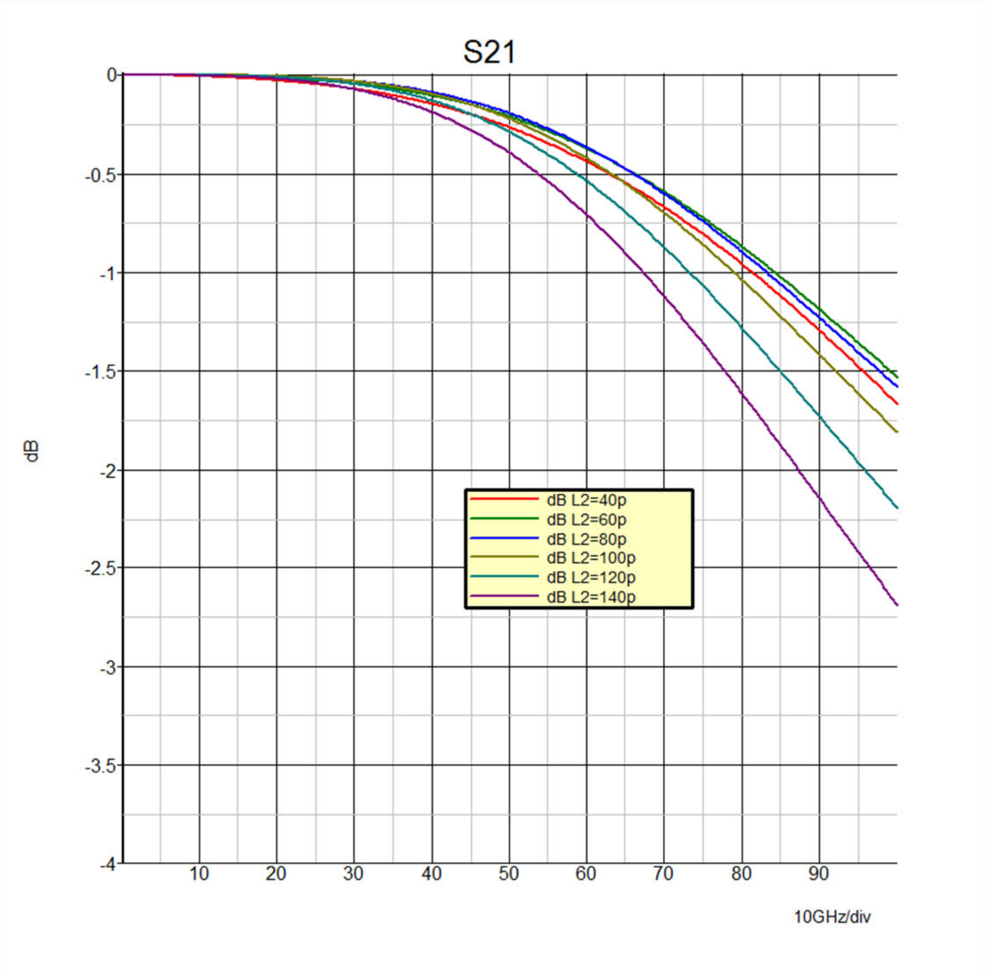
Adjusted:

Signal via diameter



- Smaller via gives best performance at high frequencies

- At low frequencies a moderate via diameter is optimal



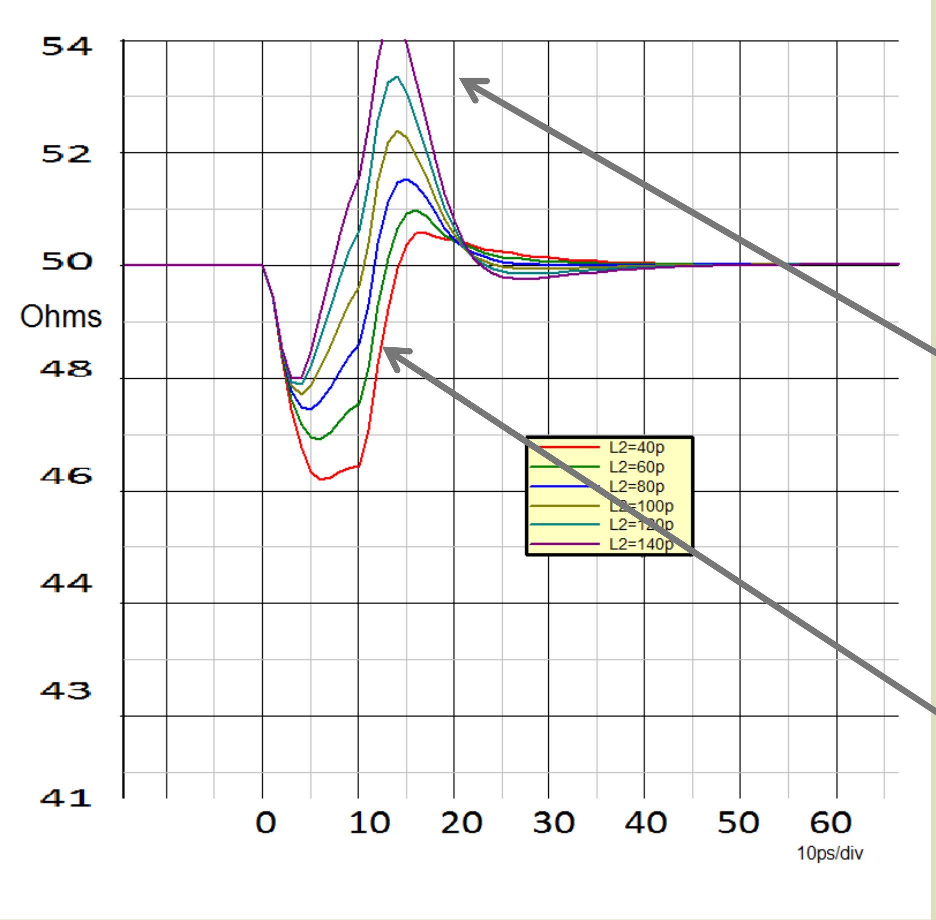
Insertion loss

Adjusted:

Signal via diameter

- Small but not smallest via gives best performance at high frequencies

TDR



Adjusted:

Signal via diameter

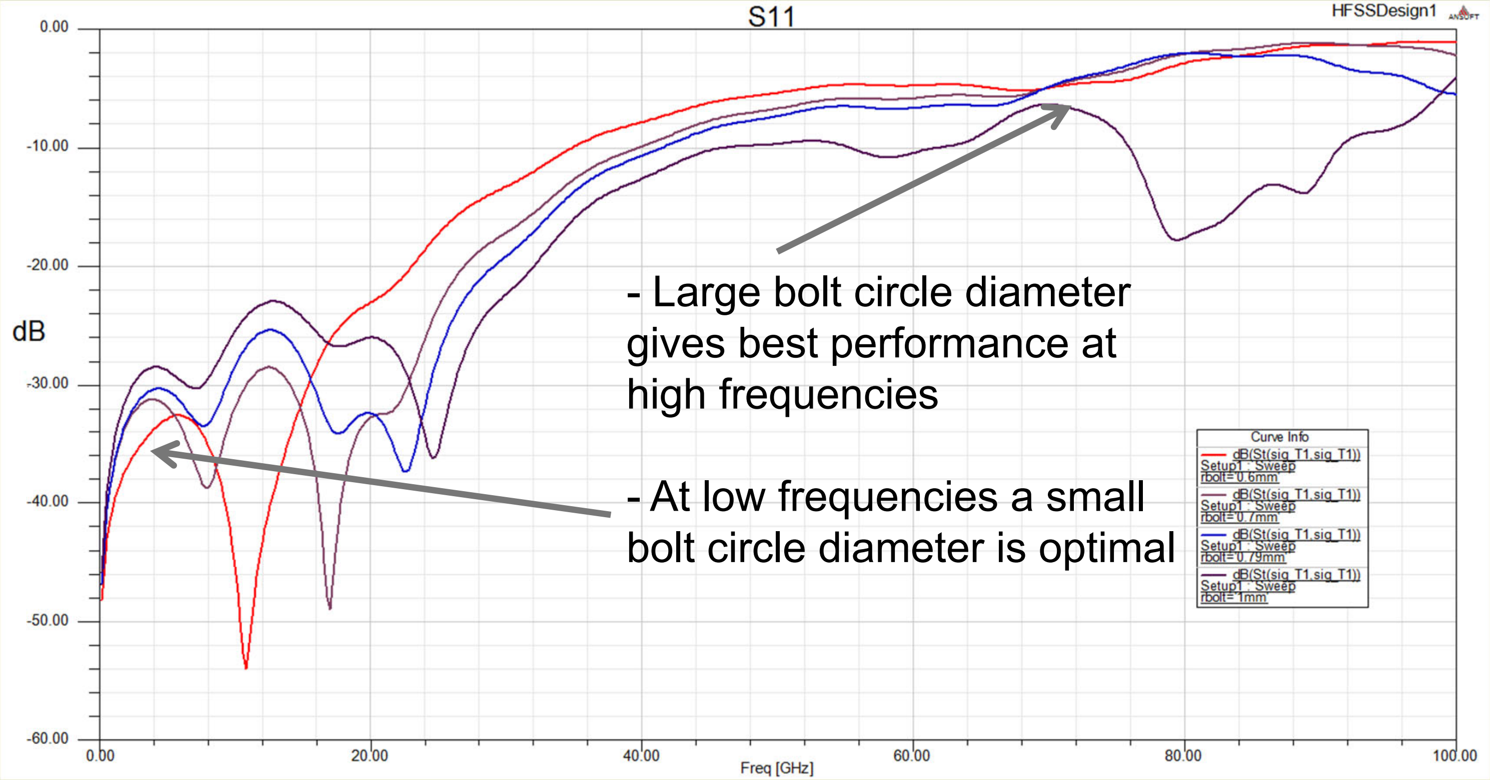
- Smaller via gives higher impedance peak

- At low frequencies a slightly larger via provides better balance

TestConX 2023

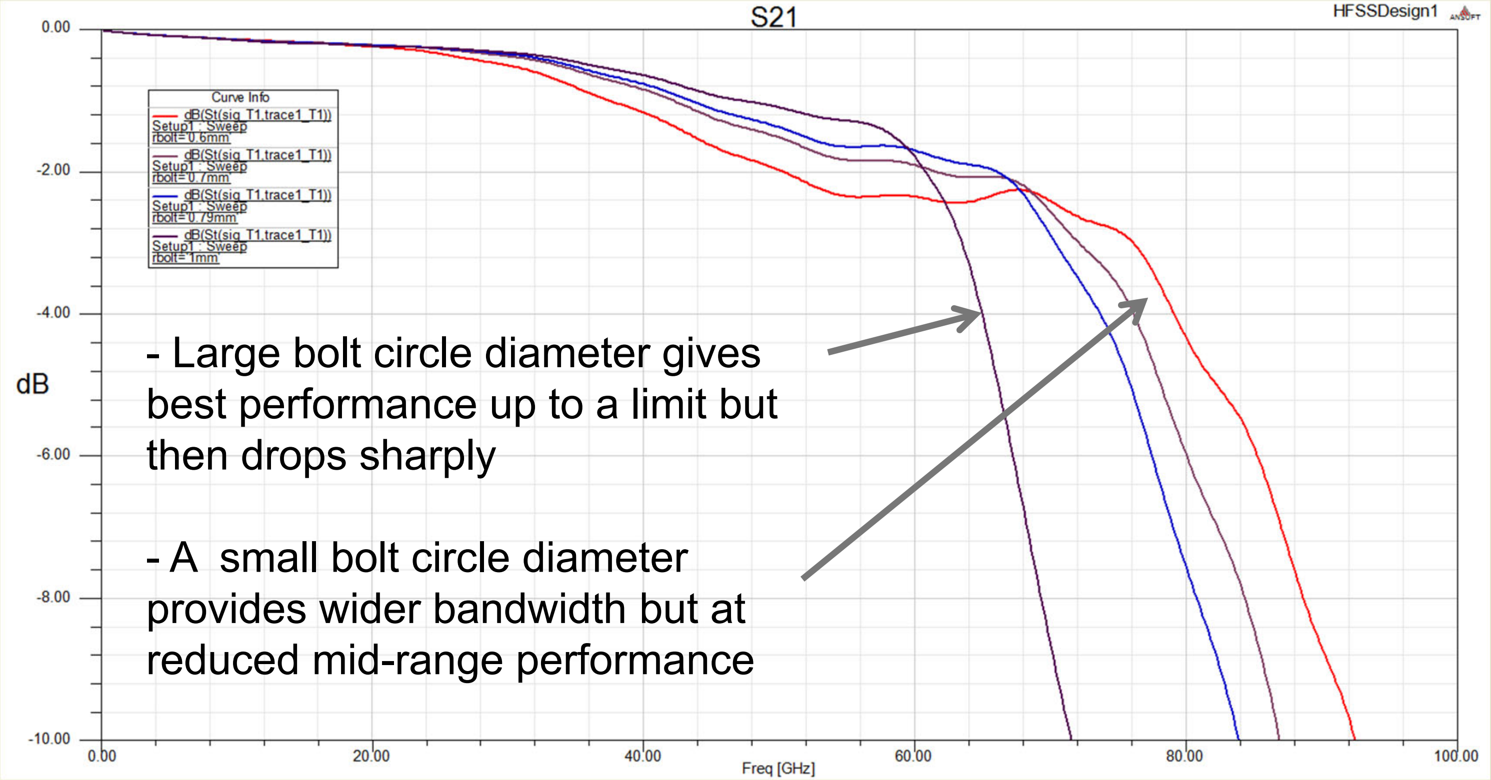
Return loss

Adjusted: Ground via bolt circle diameter



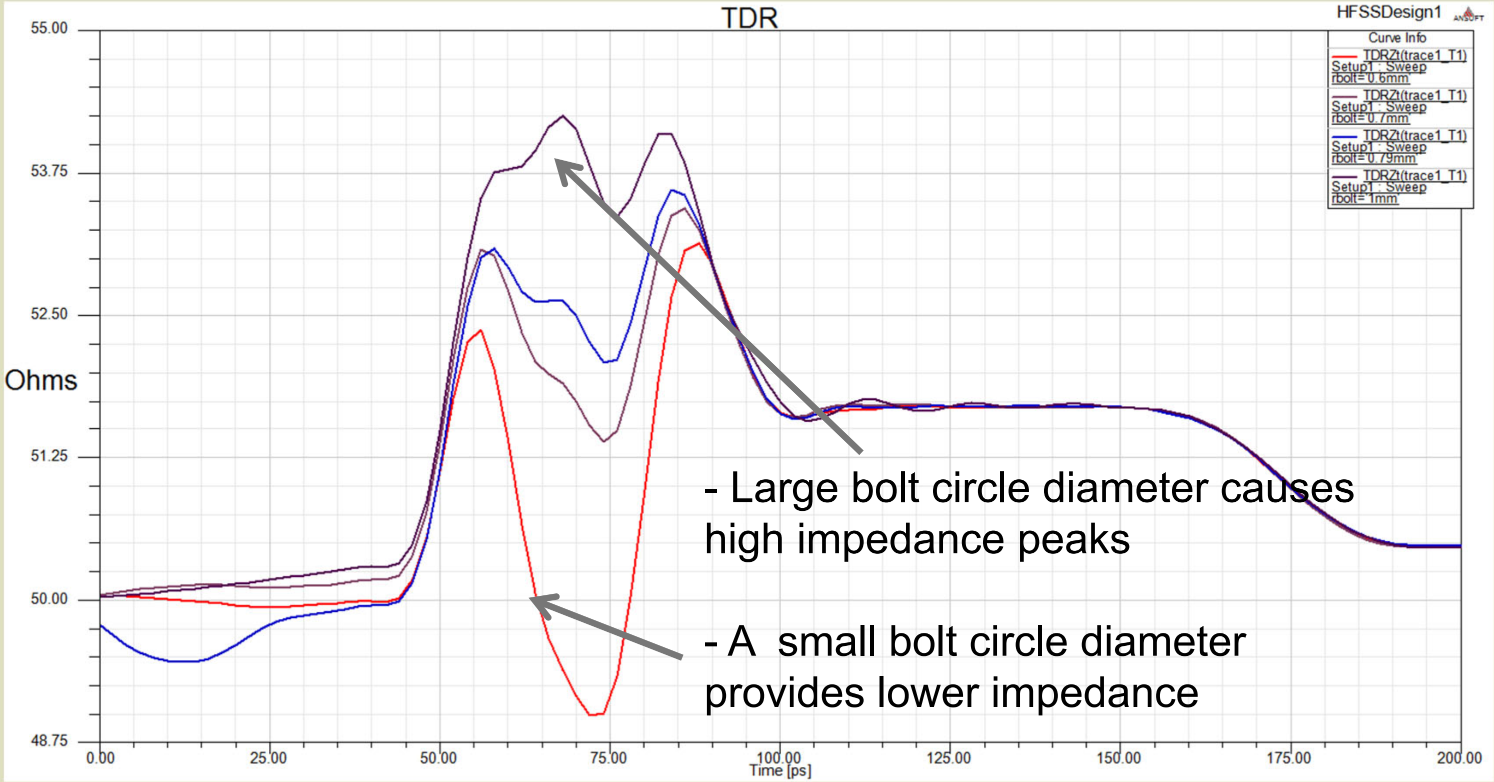
Insertion loss

Adjusted: Ground via bolt circle diameter



TDR

Adjusted: Ground via bolt circle diameter



Comments

- Due to the relatively large number of variables a simple “optimal” design is likely not available and judgment calls regarding parameter selection may be necessary
- Design for best performance at the highest frequencies can compromise low end performance
- Especially for the bolt circle diameter design compromises must be examined to achieve the desired overall performance

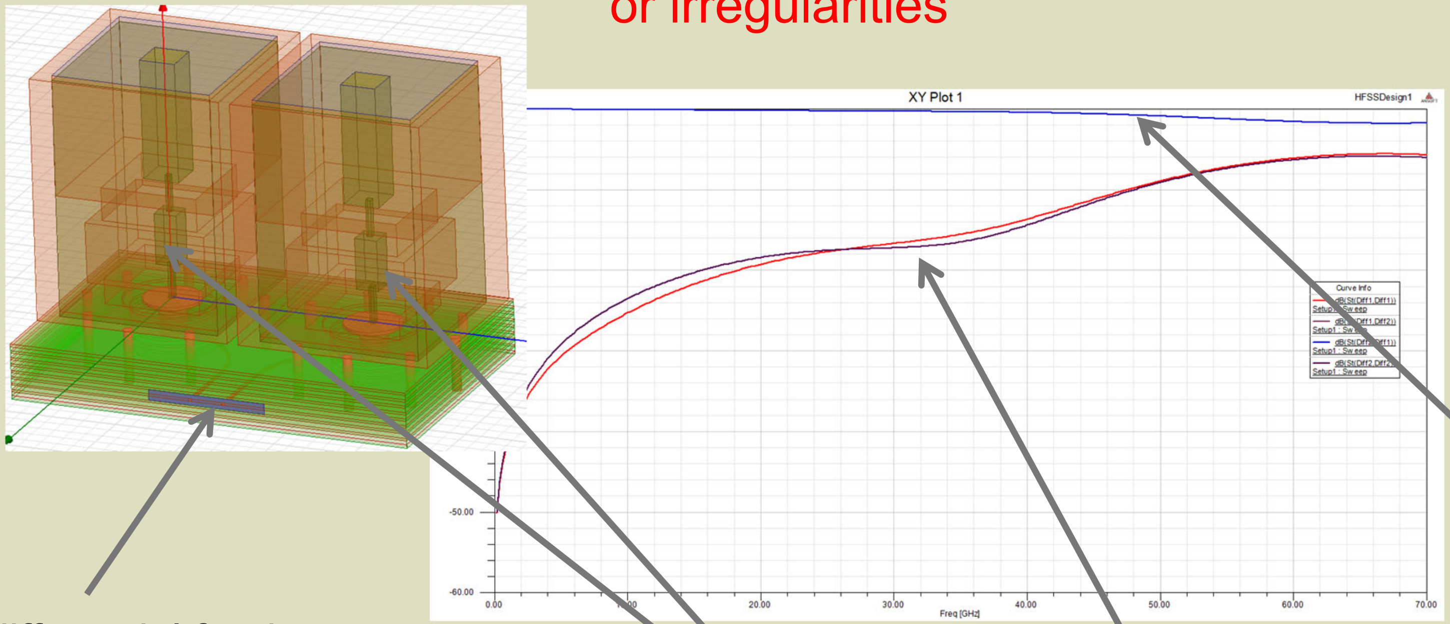
Examples of Pitfalls

- Problems with design architecture
  - Coupled-to spaces
  - Ground via locations
    - Single-ended (see above, must be impedance controlled)
    - Differential (distance to signal, symmetry)
- Problems with simulation environment itself
  - Space selection
  - Radiation boundaries

TestConX 2023

Resonance-free design

Reference: While not at best performance (unoptimized) there are no resonances or irregularities



PCB with differential feed

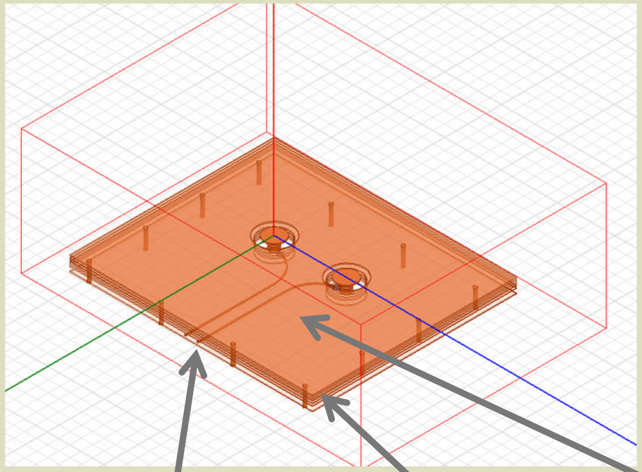
Generic coaxial feed structures

Return loss

Insertion loss

Coupled-to 'cavity'

Problem: Resonances and irregularities



PCB with differential feed

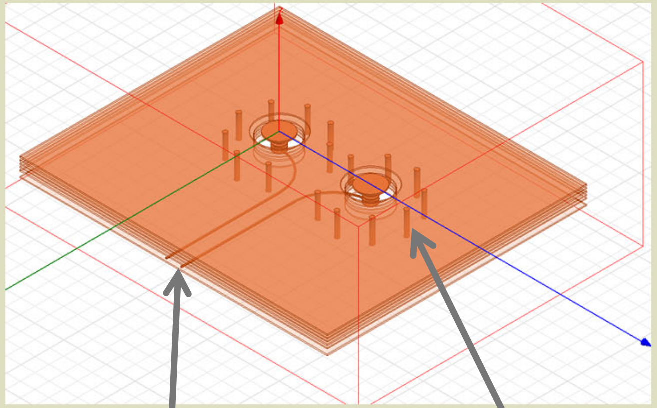
Ground vias form a large enclosed space far from signal vias

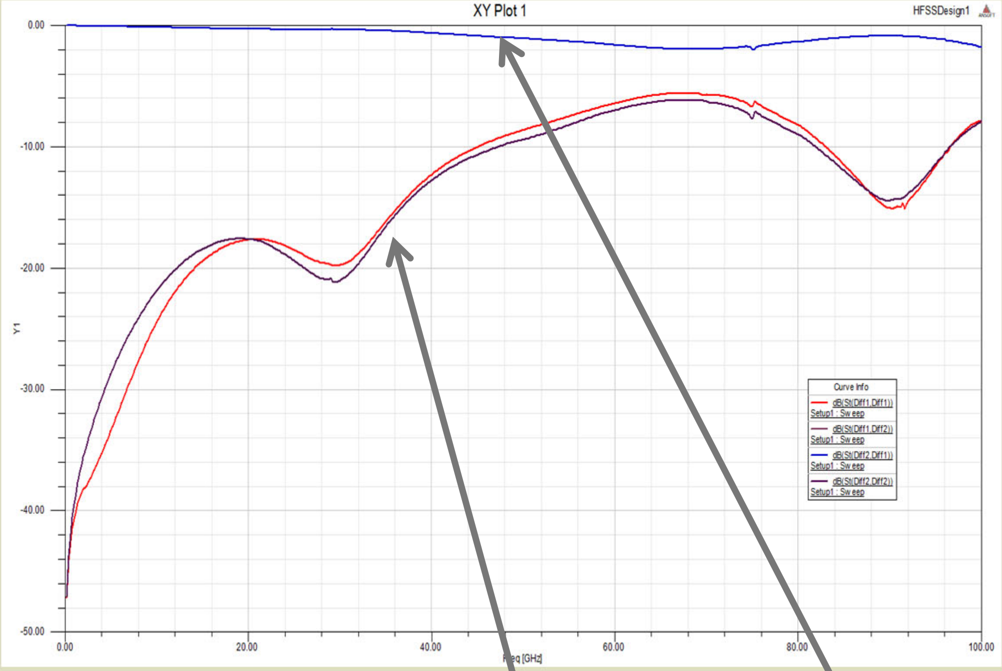
Return loss

Insertion loss

Ground via ‘shield’

Benefit: Significantly reduced resonances and irregularities





Ground vias form a small enclosed space near signal vias

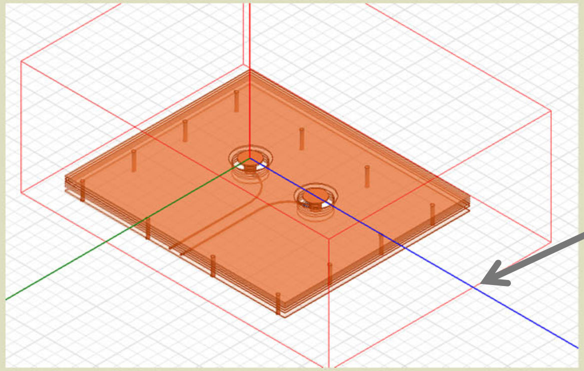
PCB with differential feed, radiation boundaries spaced away from edges

Return loss

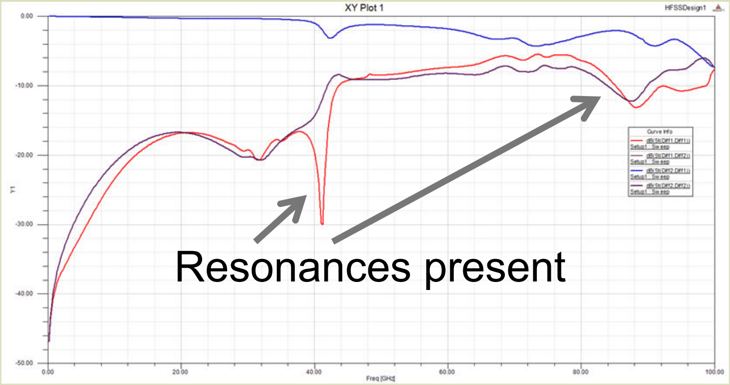
Insertion loss

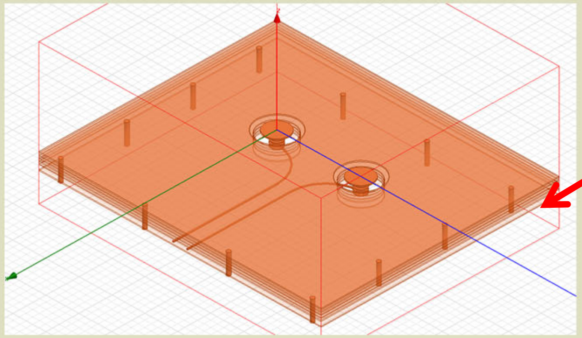
Radiation boundary selection

Risk: Resonances may become suppressed if boundary borders against PCB extents is selected

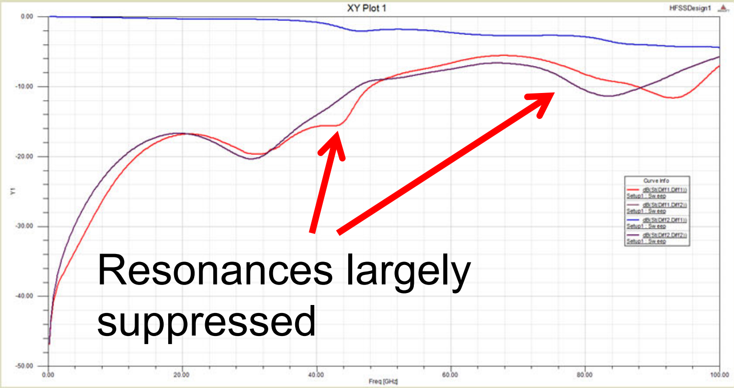


Radiation boundary distant





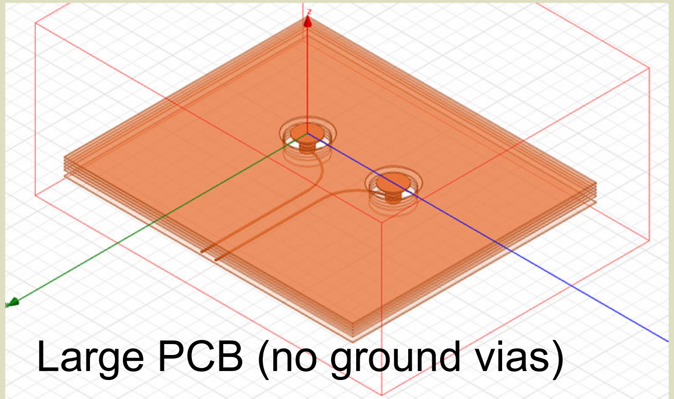
Radiation boundary tight



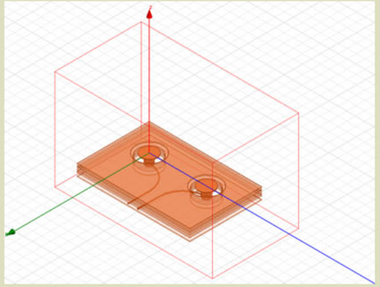
TestConX 2023

Problem size selection

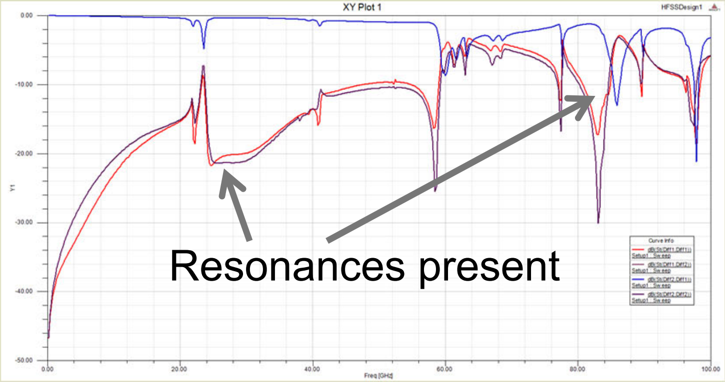
Risk: Resonances and coupling may be overlooked

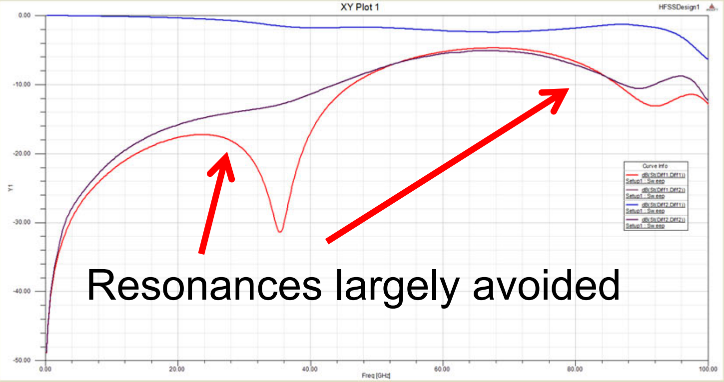


Large PCB (no ground vias)



Small PCB (no ground vias)





Conclusions

- Optimization of PCB parameters is mandatory to achieve any performance at high frequencies
- A comprehensive understanding of the simulation environment is necessary to arrive at ‘close to valid’ solutions in PCB simulations and optimizations
- Ground via location must be carefully controlled even in differential signaling environments
- Multiple “optimal” designs may exist and depend on initial parameter selection