THE ART AND SCIENCE of PCB DESIGN

IAP 2025

Lecture 07 - HIGH-SPEED

Advanced Layout: High-Speed

Outline

- Introduction
- High-Frequency routing
- PCBs for Planar Microwave Devices
- Examples

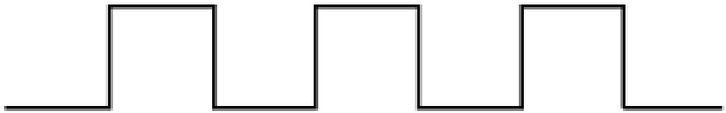
High-Frequency Signals

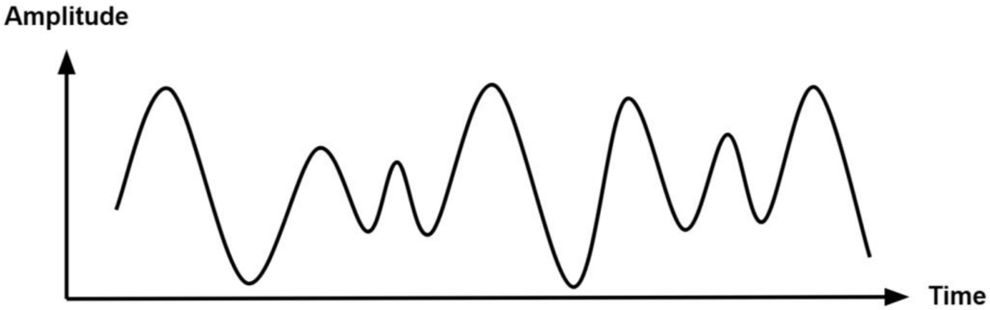
Definition

Signals are merely useful, semi-periodic waveforms that we represent as changing voltage potentials or currents.

Examples include:

- **Analog sensor outputs**
- **Digital communications**
- **Radio signals**
- **Output of an on/off switch**





https://www.monolithicpower.com/en/analog-vs-digital-signal

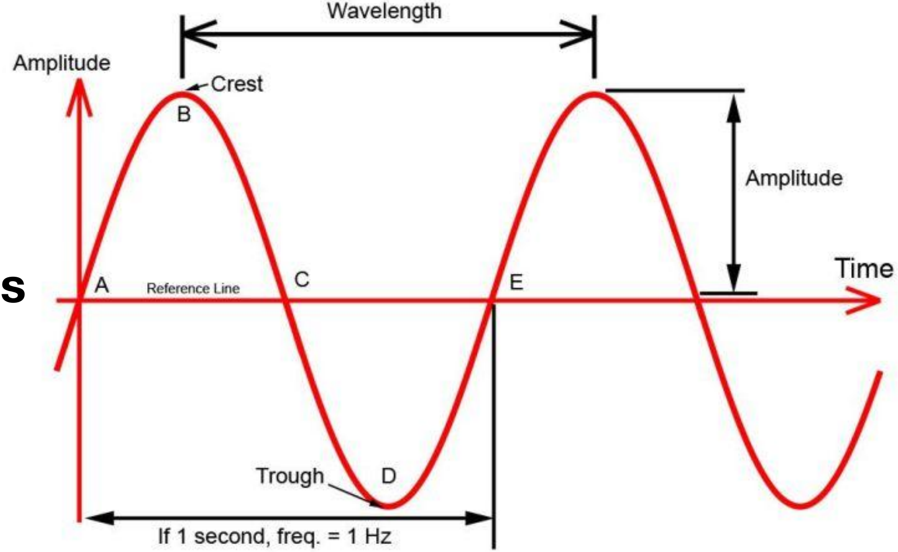
High-Frequency Signals

Definition

THE ART AND SCIENCE of PCB DESIGN
IAP 2025

For simplicity, we can represent all signals as simple sinusoidal waves of current that produce electric and magnetic fields on our PCB

(Fourier decomposition allows us to do this)



@ MIT

High-Frequency Routing

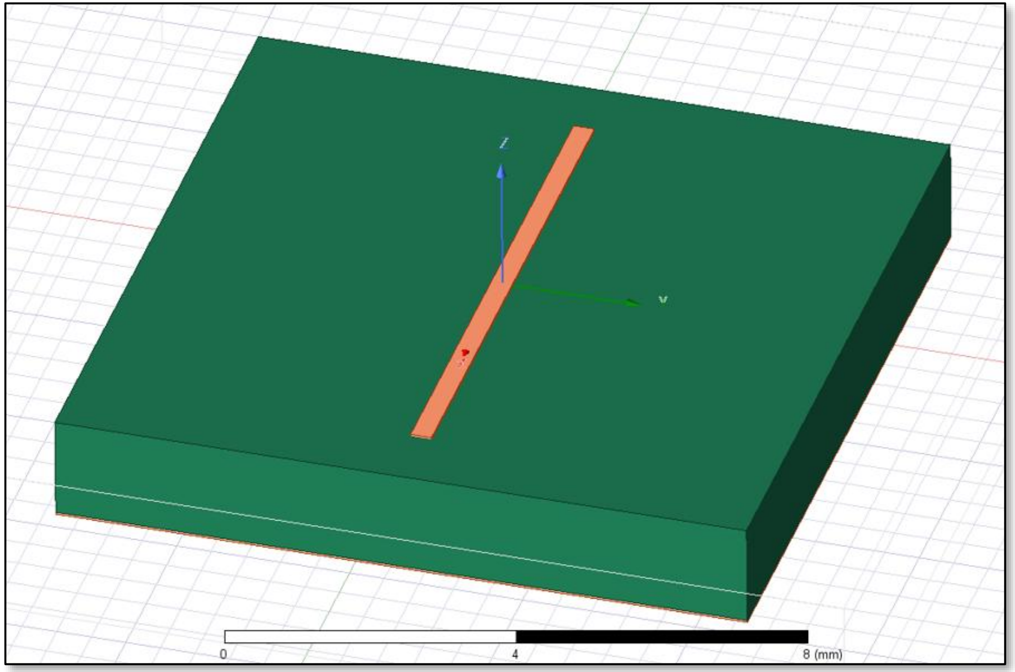
Microstrip

Using Ansys HFSS to analyze how high-frequency signals are affected by PCB design elements

Taking a look at a microstrip model, consists of:

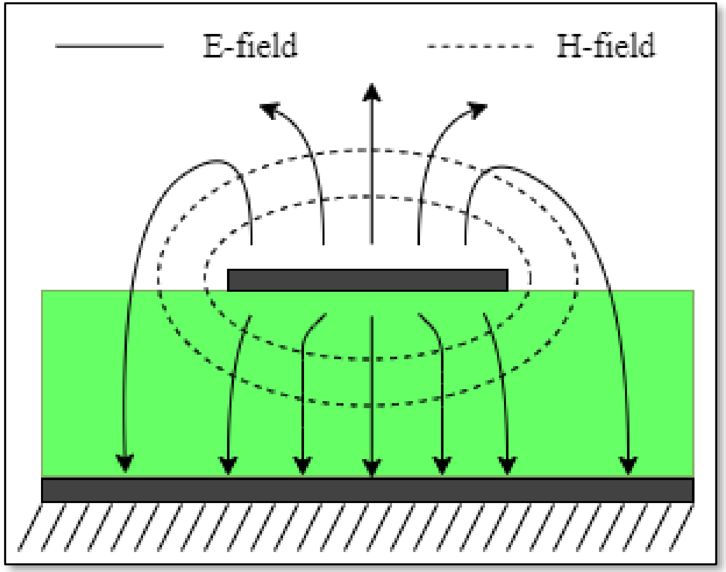
- **Copper trace**
- **Dielectric (FR408HR used)**
- **Copper ground plane**

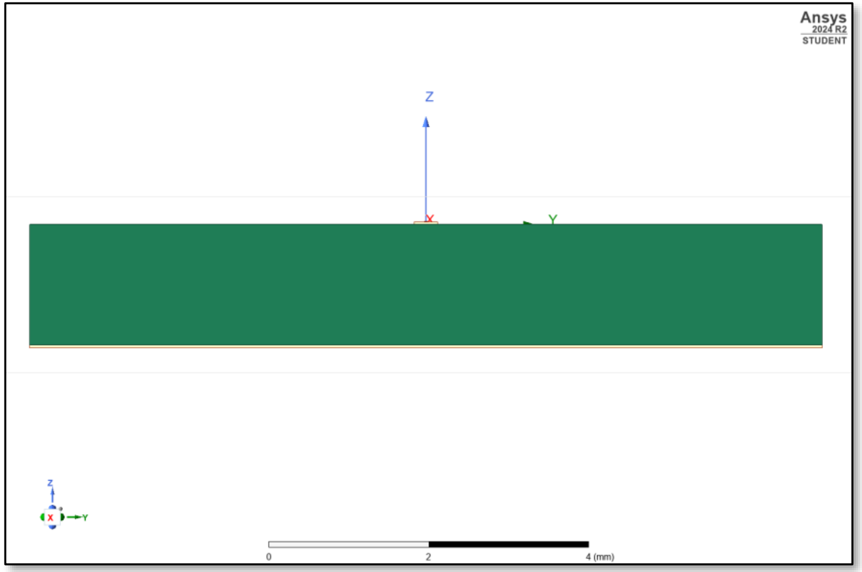
Lumped element ports are attached to either end of the trace



High-Frequency Routing

Microstrip





https://www.signalintegrityjournal.com/articles/2378-measuring-the-bulk-dielectric-constant-dk-on-a-microstrip-with-a-tdr

High-Frequency Routing

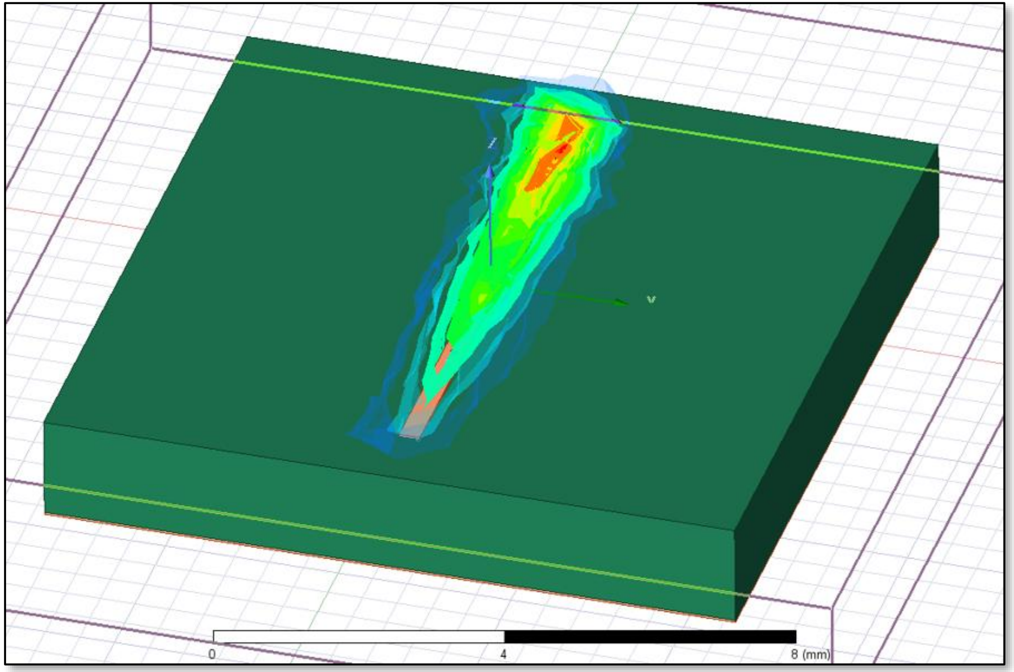
Microstrip

We can inject high-frequency signals on either end/port on the trace and measure how they are affected:

- **Signal degradation**
- **Coupling**
- **Radiation**
- **Reflections**

Able to visualize electric fields throughout materials (analogous to the current flow)

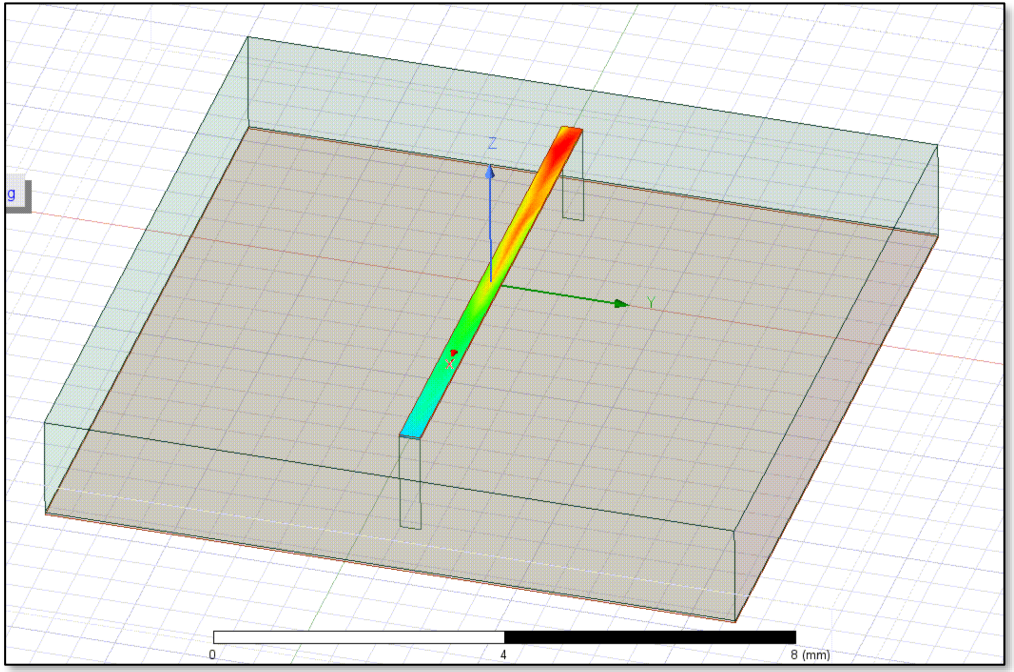
Electric fields radiate around trace



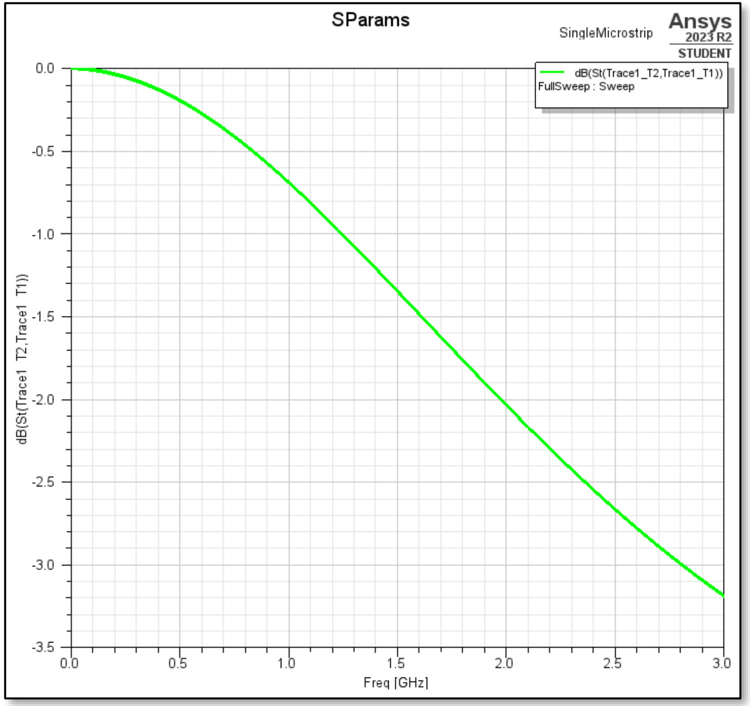
High-Frequency Routing

Microstrip

Electric field plotted on trace (2.4GHz source)



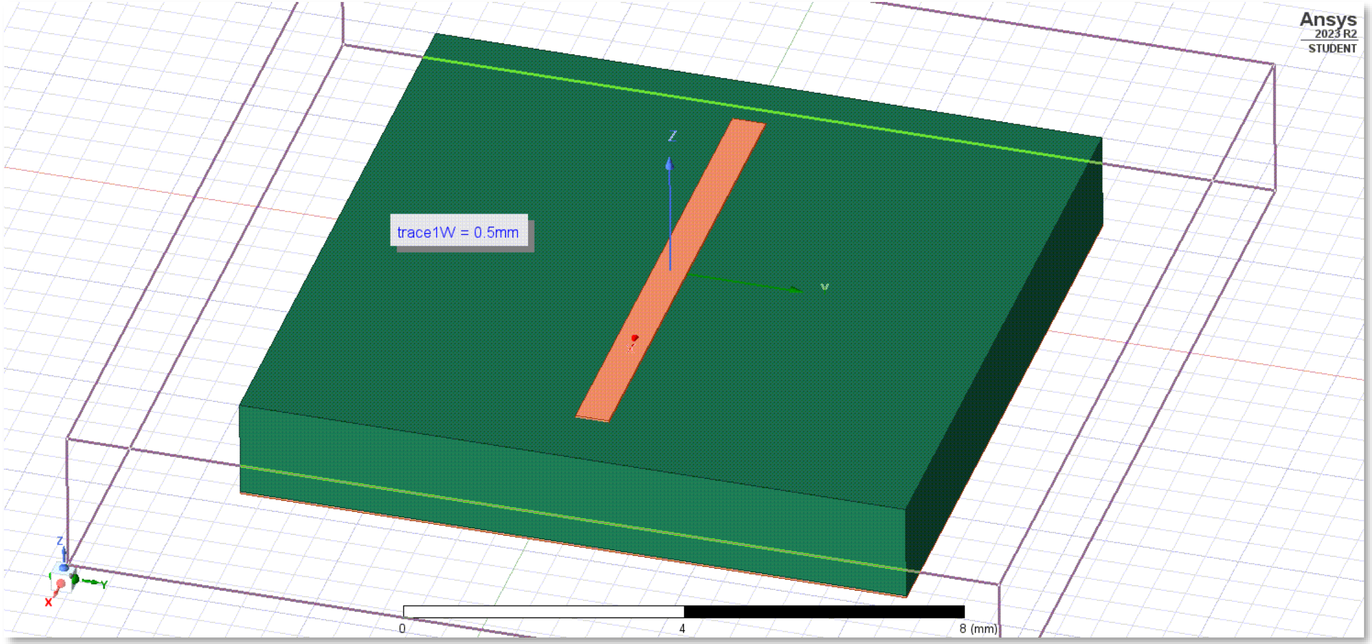
Signal Loss (dB) vs. Frequency



High-Frequency Routing

Impedance Matching

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025



For a microstrip, impedance can be adjusted by changing the trace width.

The ports are 50Ω terminated. Matching the trace impedance close to 50Ω will result in lower loss.

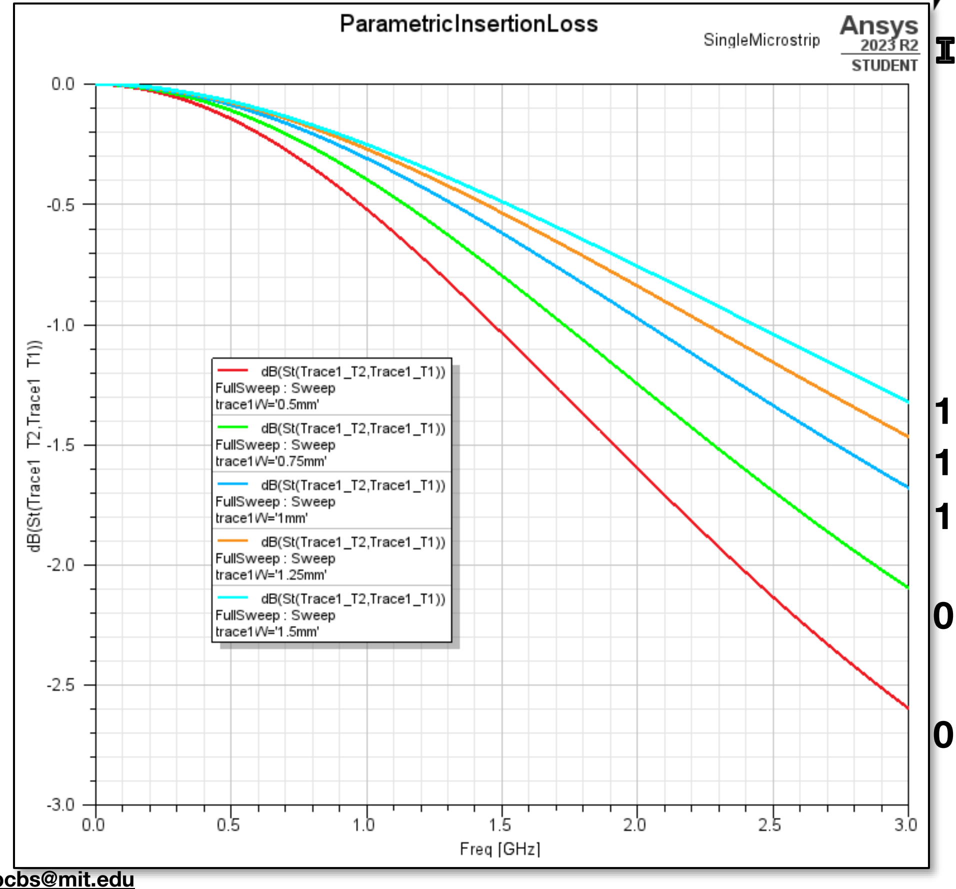
High-Frequency Routing

Impedance Matching

The currently modelled trace has a very high impedance (>50Ω). Therefore, widening it provides a better impedance match.

Impedance characteristics for a microstrip are affected by:

- **Trace width**
- **Trace thickness**
- **Dielectric height**
- **Dielectric Constant (Dk, Er)**



1.5 mm

1.25 mm

1 mm

0.75 mm

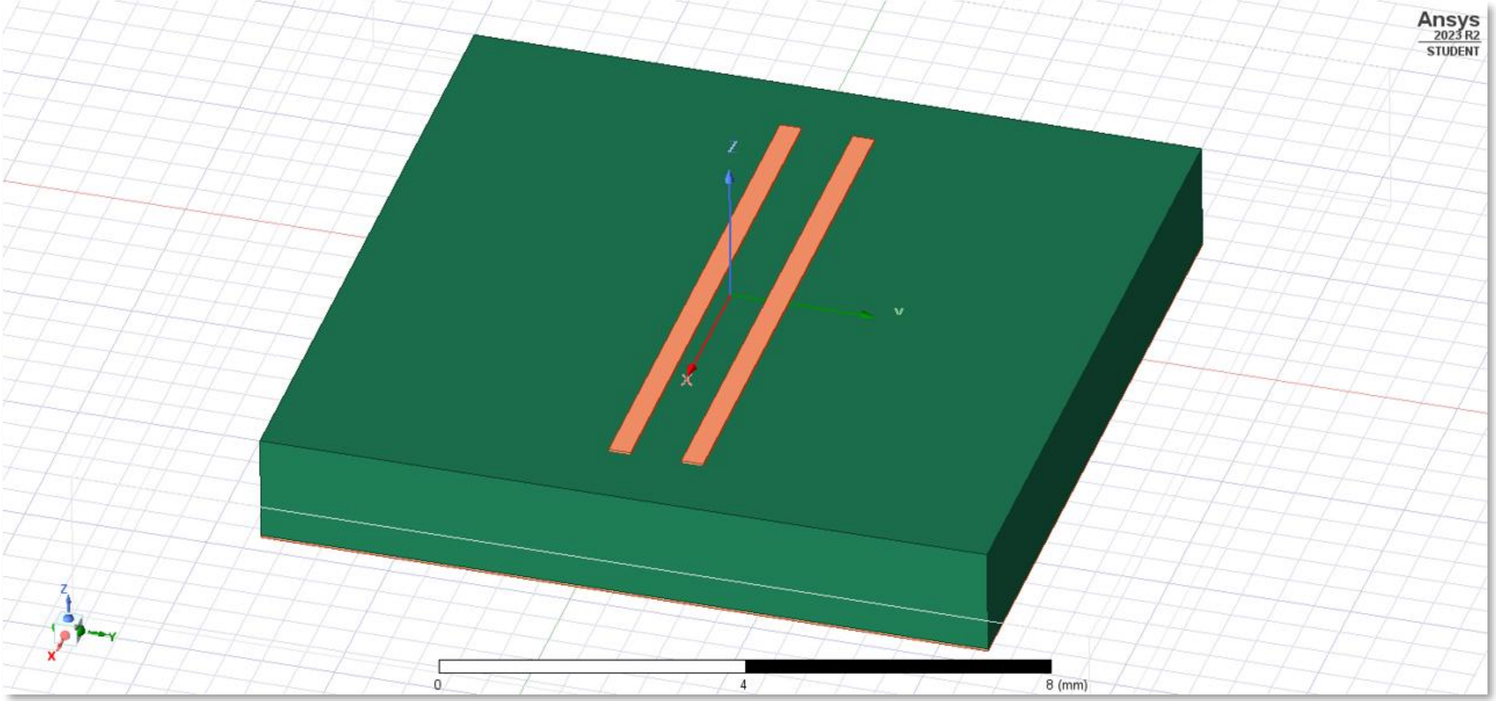
0.5 mm

High-Frequency Routing

Coupling

IAP 2025

What happens if we put another trace in parallel?



@ MIT

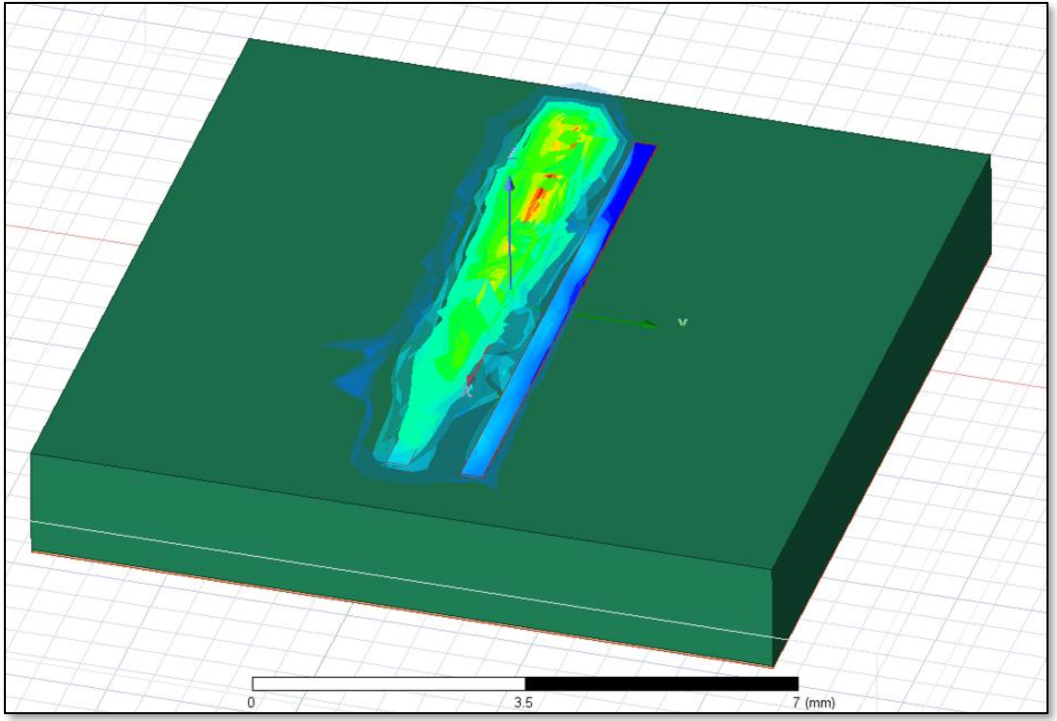
High-Frequency Routing

Coupling

Notice how the electric fields propagating through the air around the excited (left) trace reaches the parallel (right) trace and induces internal fields

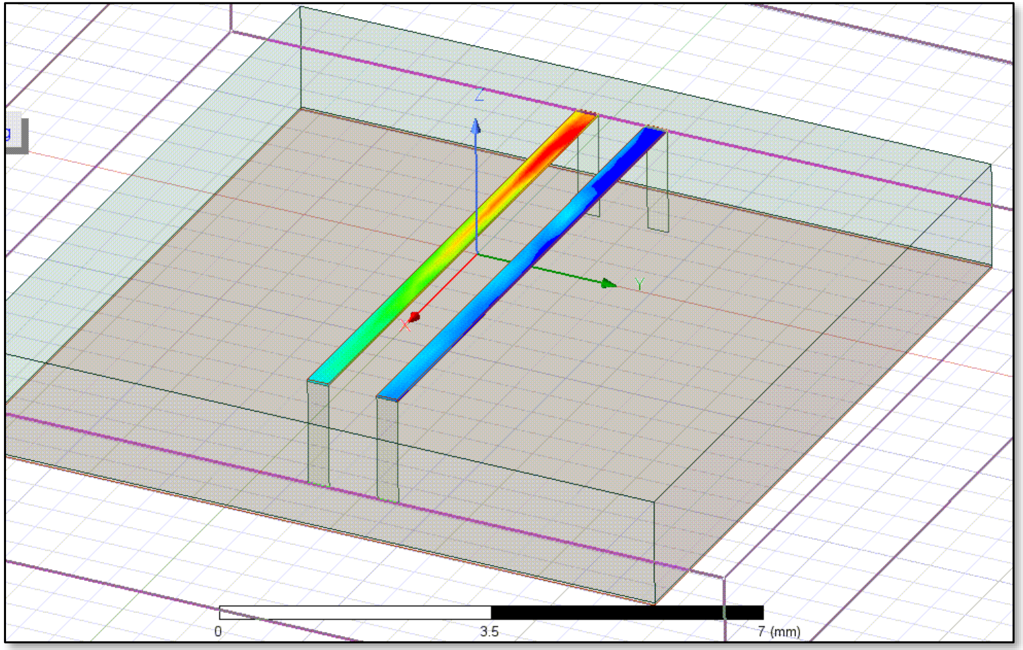
A smaller, measurable copy of the signal is being coupled to the other trace

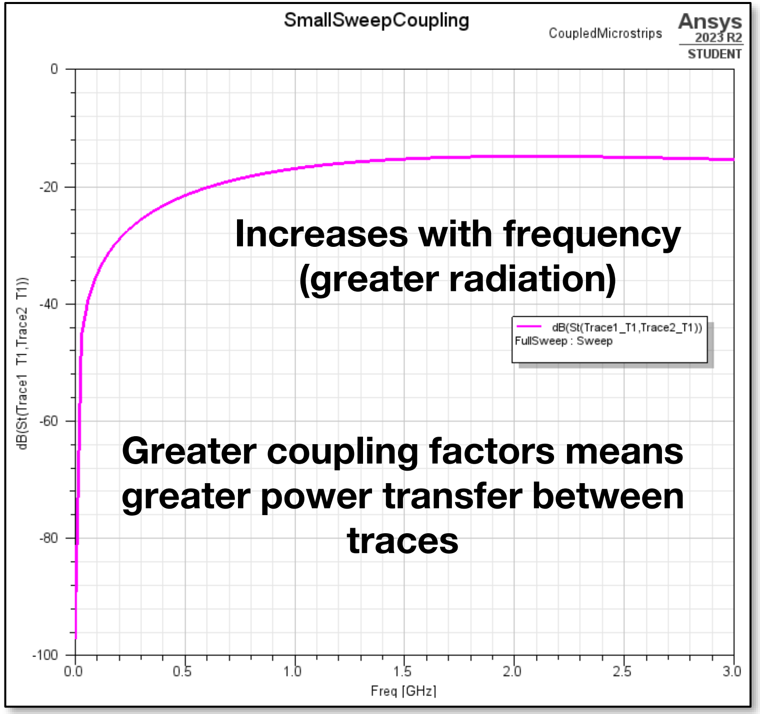
This can become a big issue for sensitive systems



High-Frequency Routing

Coupling





Increases with frequency (greater radiation)

Greater coupling factors means greater power transfer between traces

High-Frequency Routing

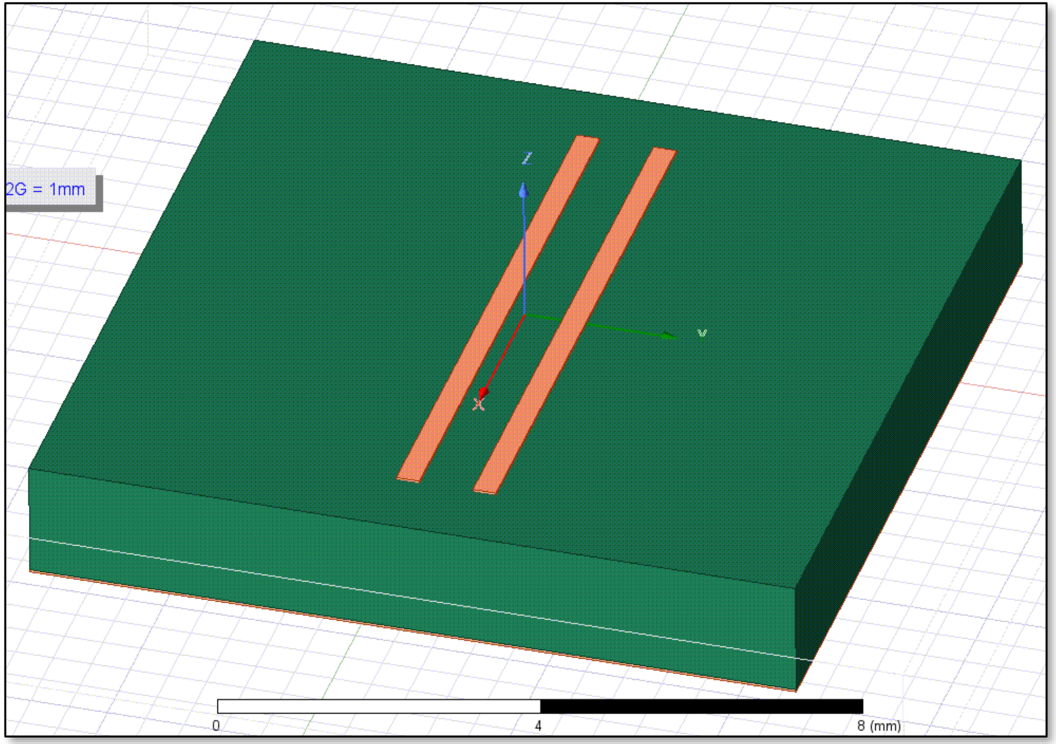
Mitigating Coupling

Coupling can adversely affect our signals.

At best, the coupled signals increase noise in our signal. At worse, they appear as valid signals at a receiving device (crosstalk)

We can reduce coupling by avoiding parallel signal traces (i.e., have them cross perpendicularly on different layers)

Increasing the separation between traces can also help



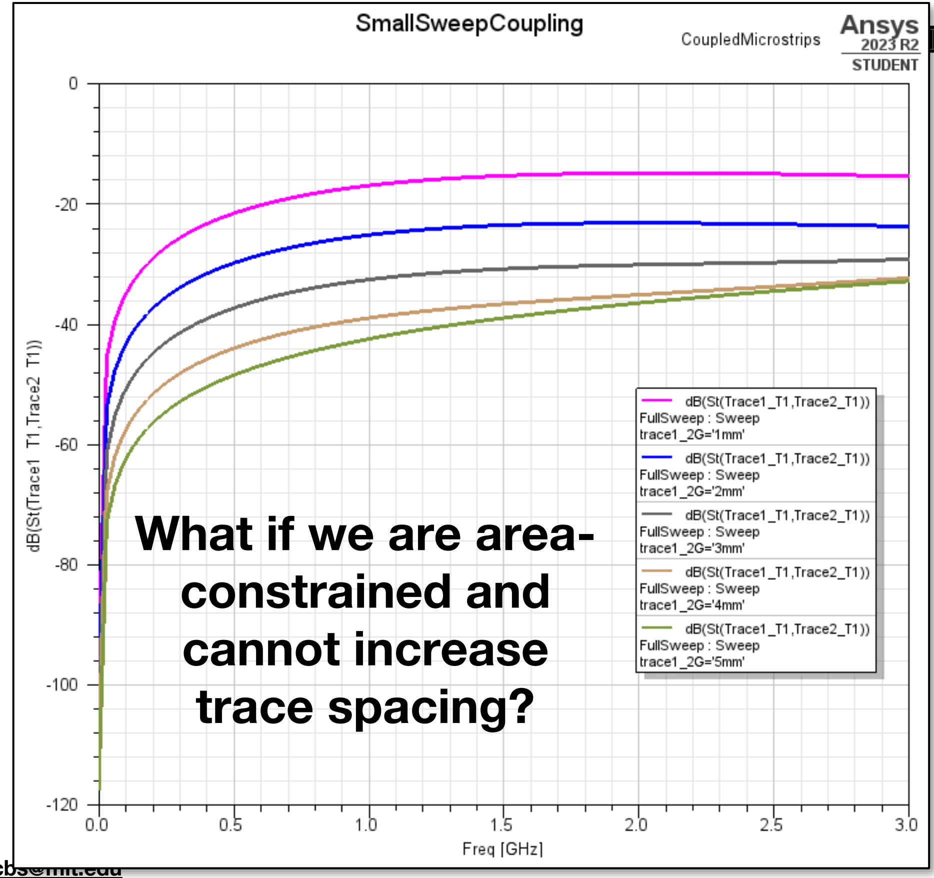
High-Frequency Routing

Mitigating Coupling

Increasing the distance between parallel traces will weaken coupling fields (inversely proportional to distance)

For differential traces, this coupling is utilized to maintain a particular differential impedance

- **Ensure to follow impedance requirements for differential signal trace pairs**



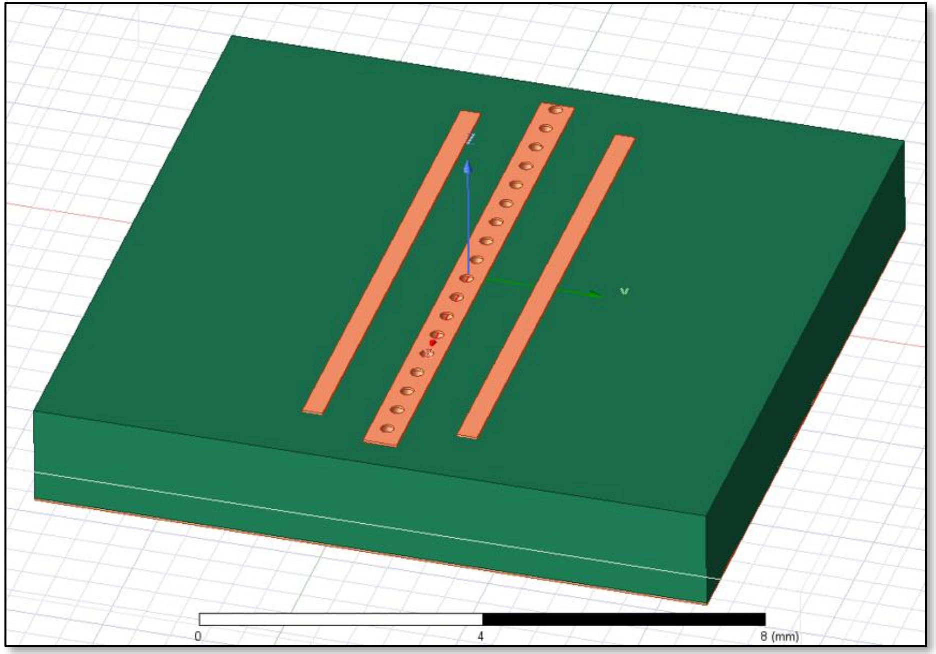
What if we are area-constrained and cannot increase trace spacing?

IAP 2025

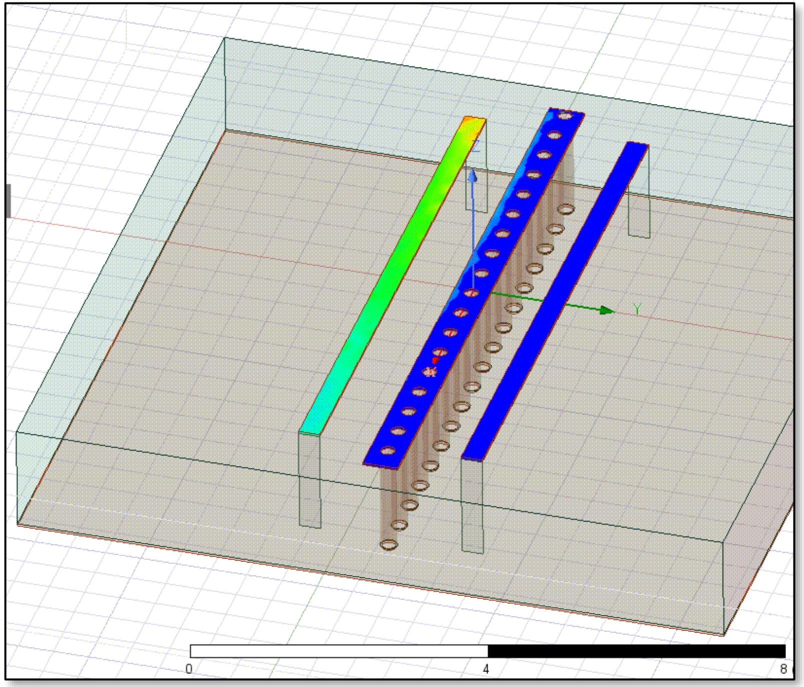
High-Frequency Routing

Mitigating Coupling

Add a ground trace/pour/via-fence between



Radiated fields couple to the return path (GND)

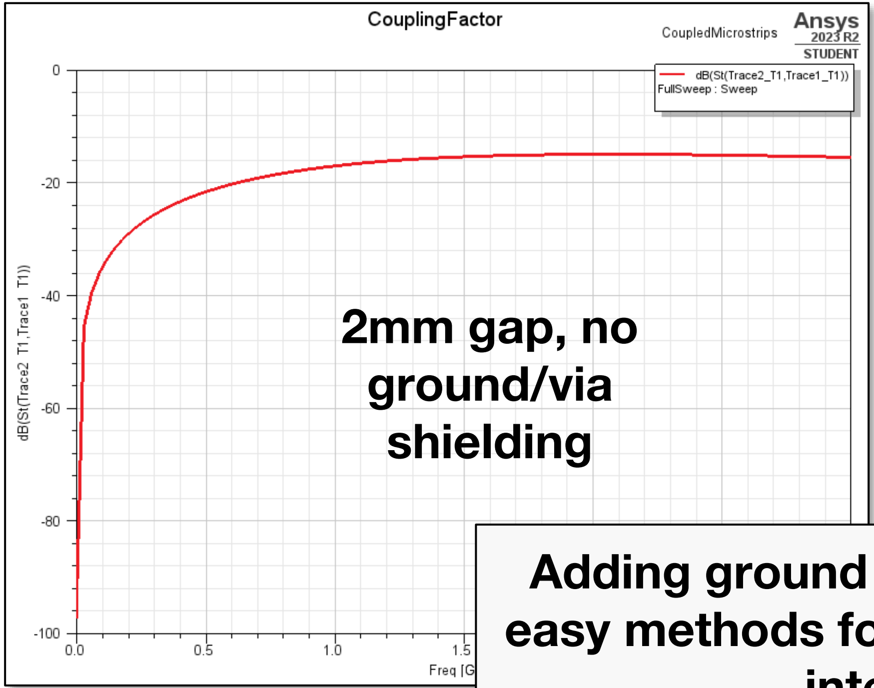


THE ART AND SCIENCE of PCB DESIGN
IAP 2025

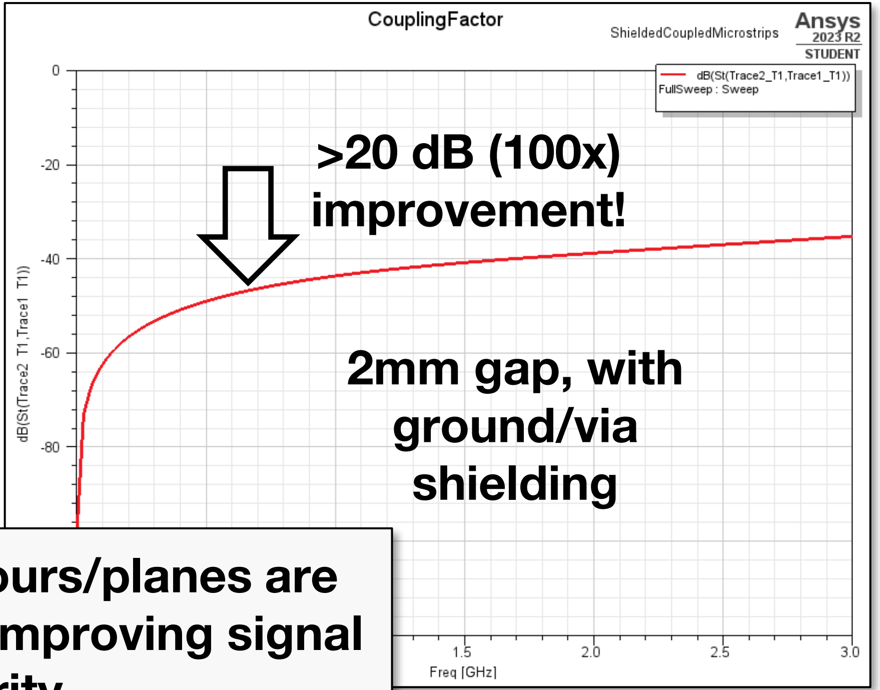
@ MIT

High-Frequency Routing

Mitigating Coupling



2mm gap, no ground/via shielding



>20 dB (100x) improvement!

2mm gap, with ground/via shielding

Adding ground pours/planes are easy methods for improving signal integrity

High-Frequency Routing

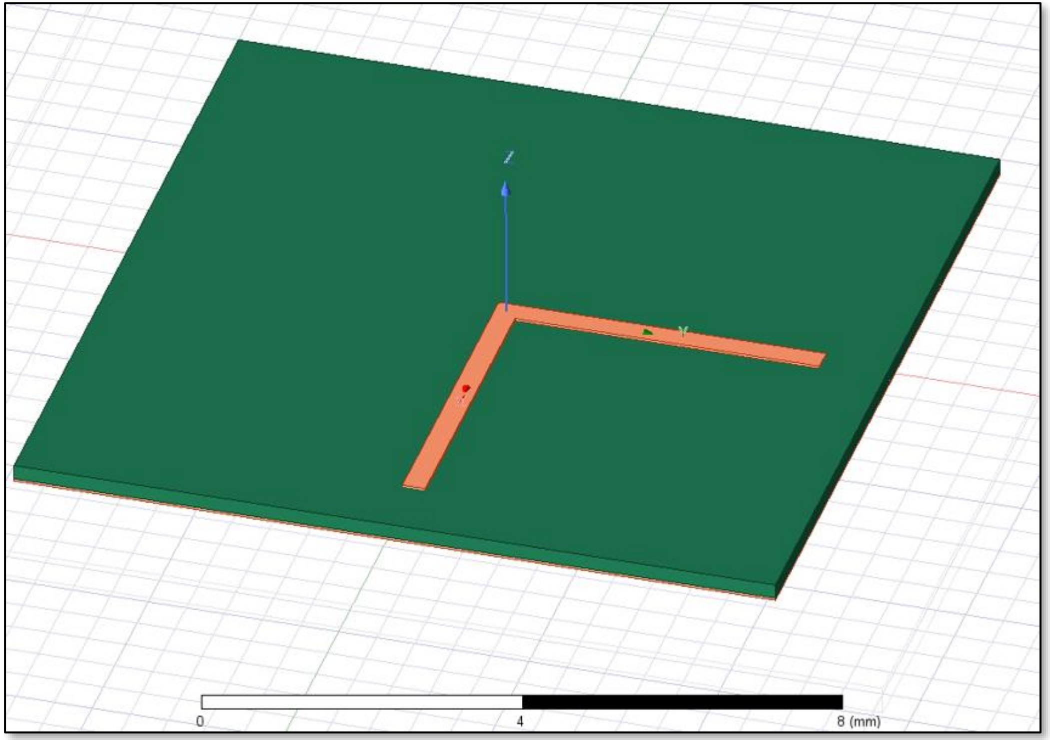
Microstrip Bends

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

Impedance mismatches can occur at abrupt discontinuities in a trace, such as sharp (90 degree) bends

The bend itself can resonate and have its own (different) impedance from the rest of the microstrip

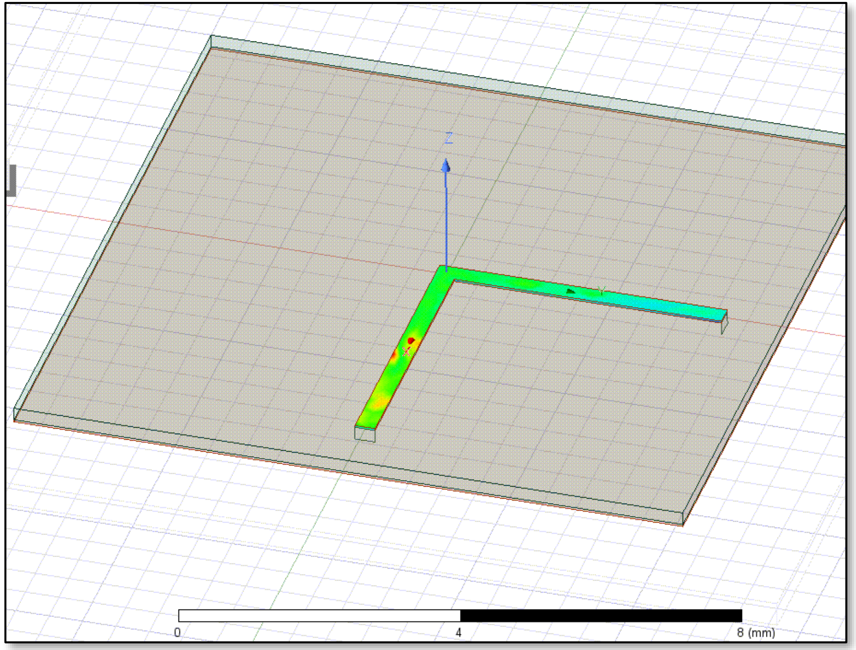
The result is signal reflections, radiation, and degradations!

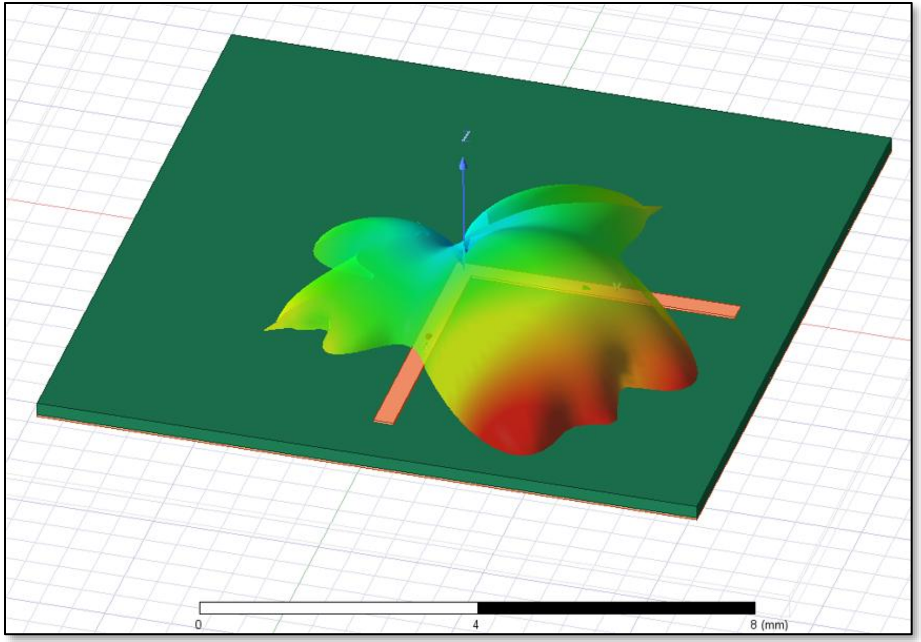


@ MIT

High-Frequency Routing

Microstrip bends

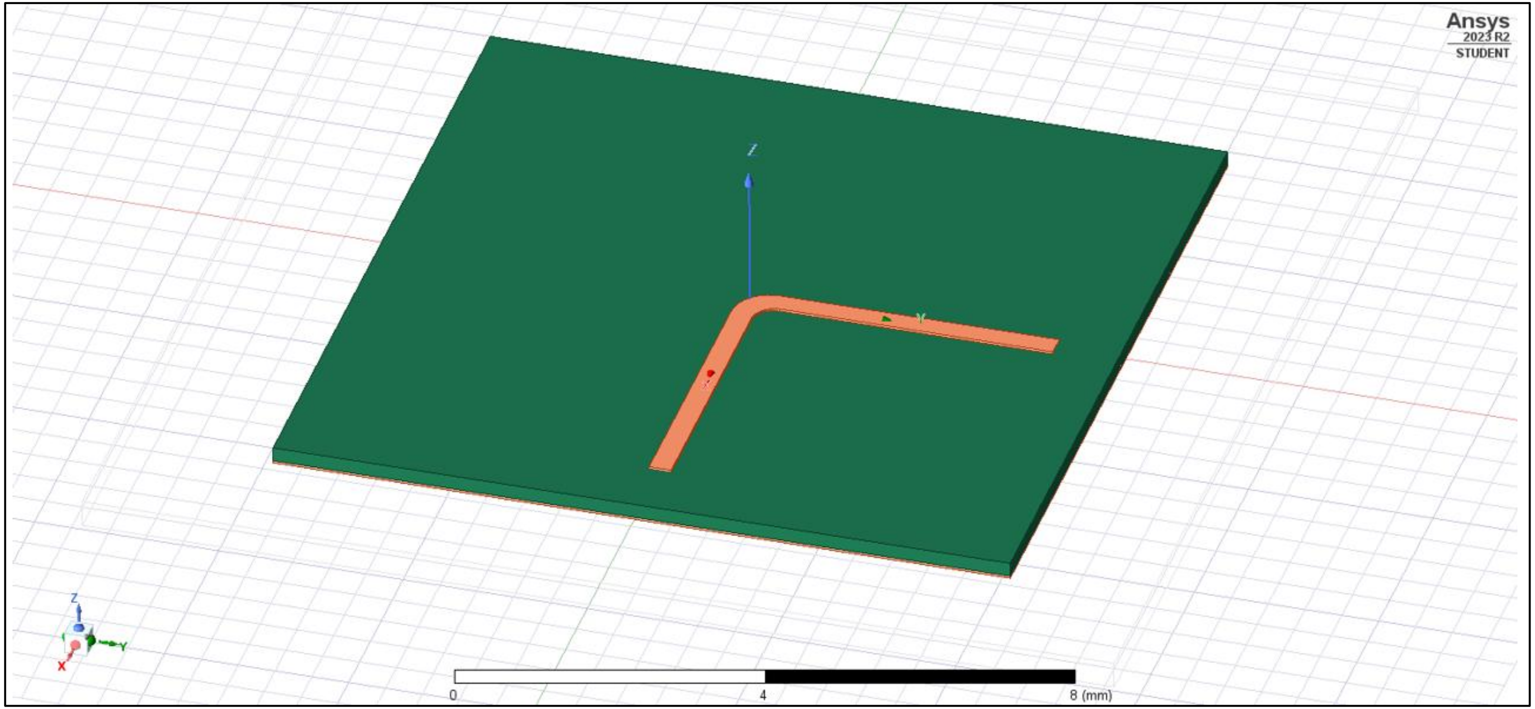




High-Frequency Routing

Microstrip Bends

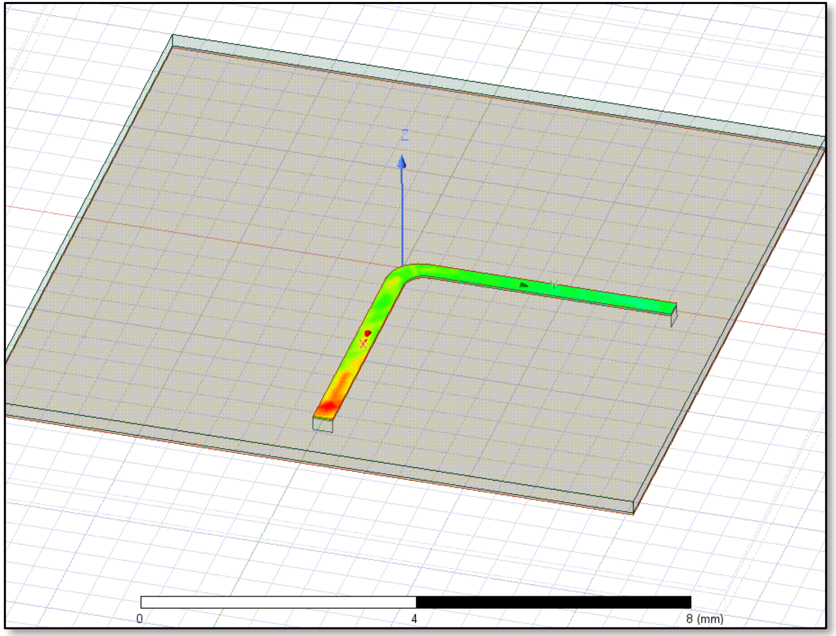
Solution: smooth, wide bends



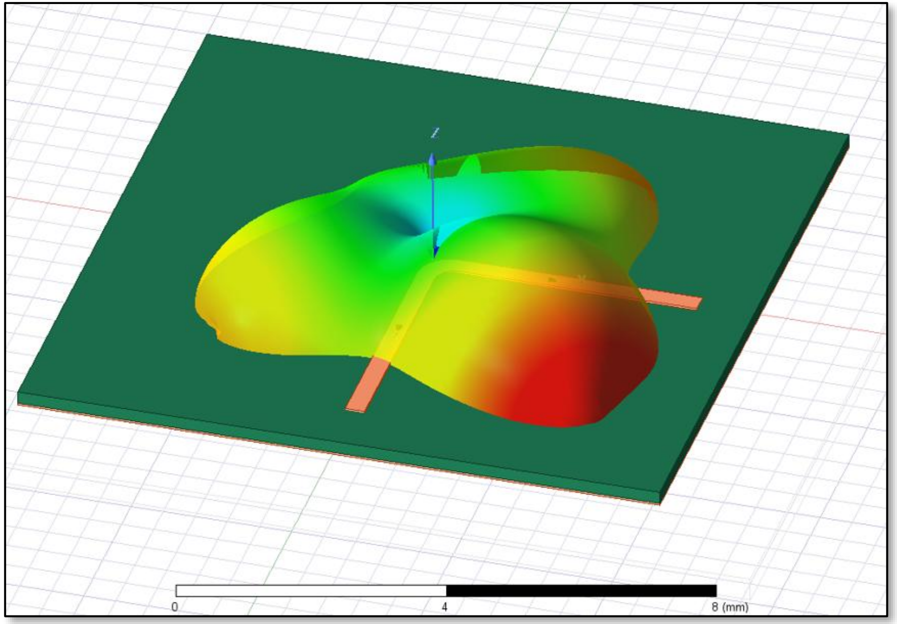
High-Frequency Routing

Microstrip Bends

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025



Even field distribution throughout bend



Smooth, weaker near-field radiation patterns

High-Frequency Routing

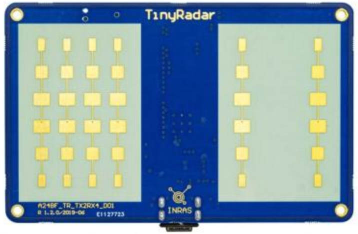
Key Takeaways

- ☐ As signals reach greater frequencies, their wavelengths decrease, which results causes greater susceptibility to wave phenomenon and parasitic circuit elements
- ☐ Impedance mismatches between traces, discontinuities, and devices can lead to signal degradation, reflection, and radiation, which we can simulate
- ☐ Use appropriate high-frequency PCB structures (e.g., microstrip, coplanar waveguide) when impedance matching is needed
- ☐ Close, parallel traces can couple to one another, use appropriate spacing, ground pours, and via fences to reduce coupling
- ☐ Sharp trace turns can cause impedance mismatches, use smooth bends instead

Examples

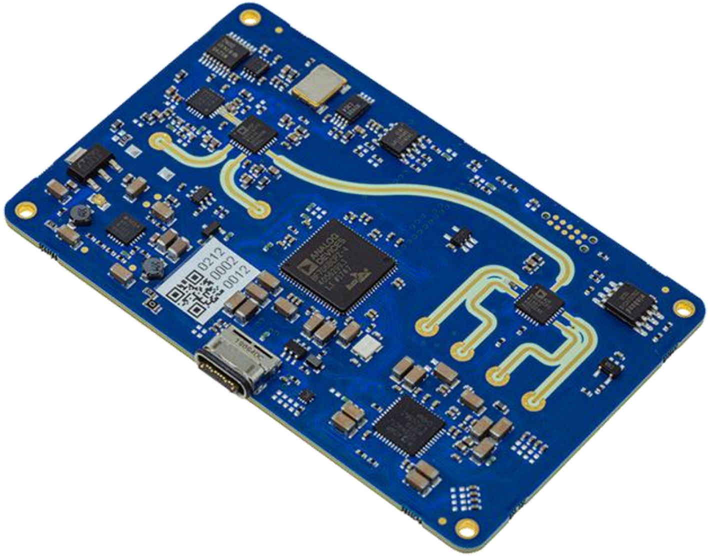
Examples

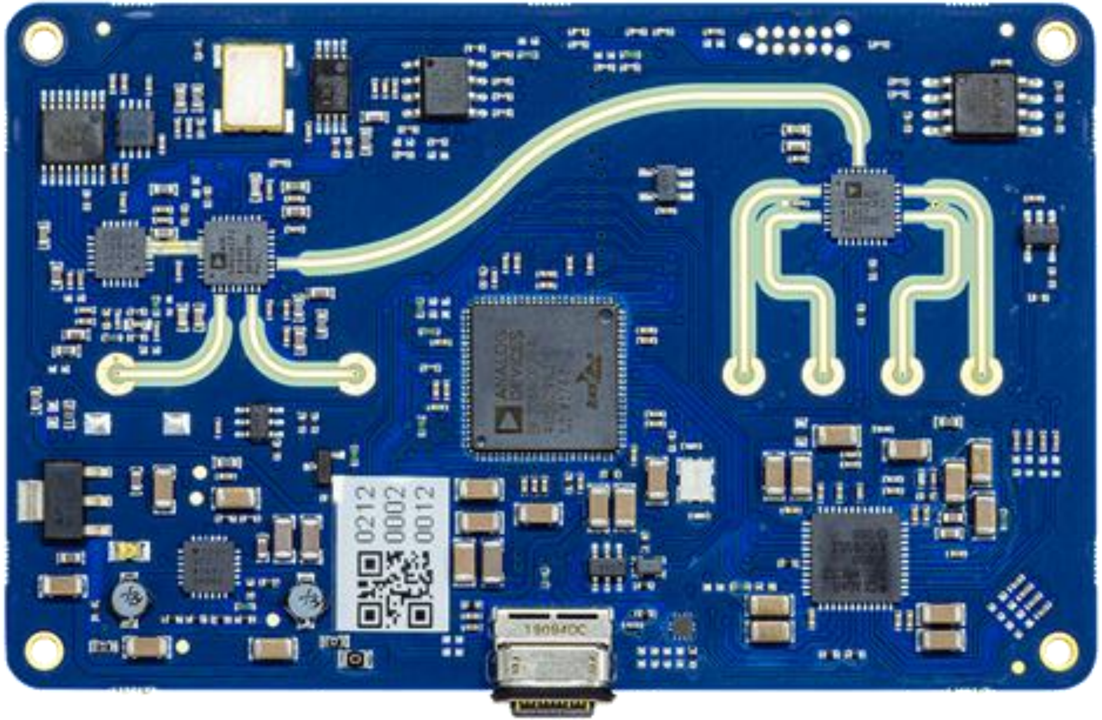
24 GHz Radar



Phased Array!

THE ART AND SCIENCE OF PCB DESIGN
IAP 2025

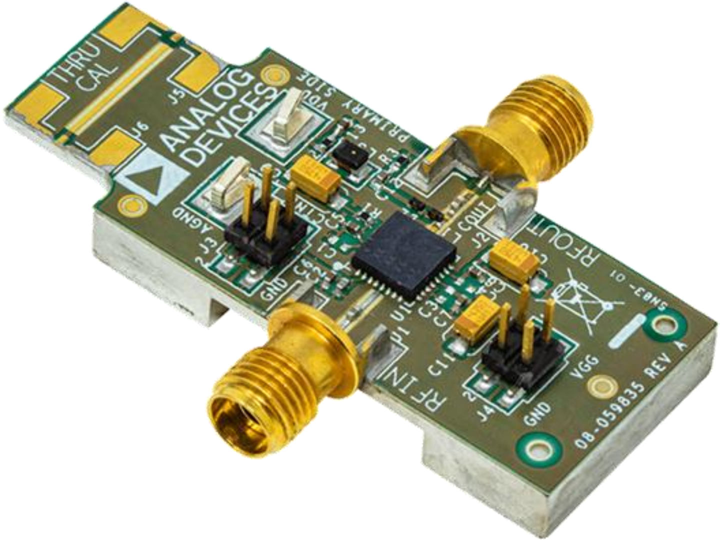


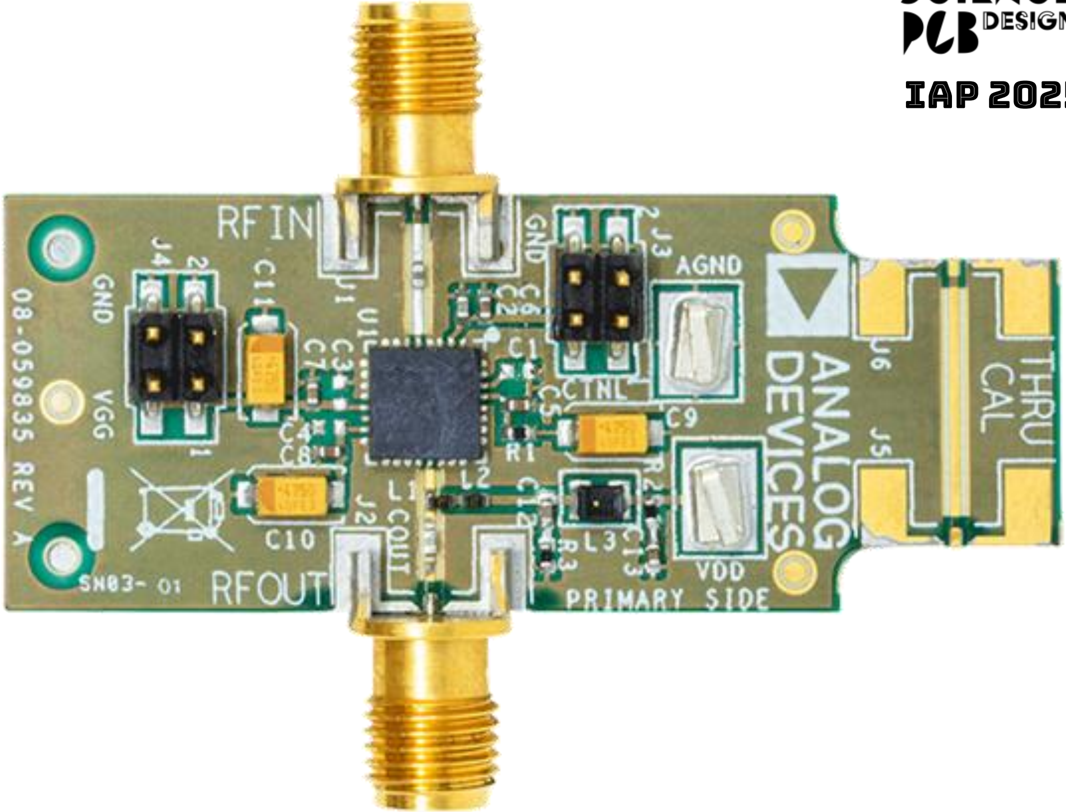


https://www.analog.com/en/design-center/evaluation-hardware-and-software/evaluation-boards-kits/eval-tinyrad.html#eb-overview

Examples

28 GHz Amplifier



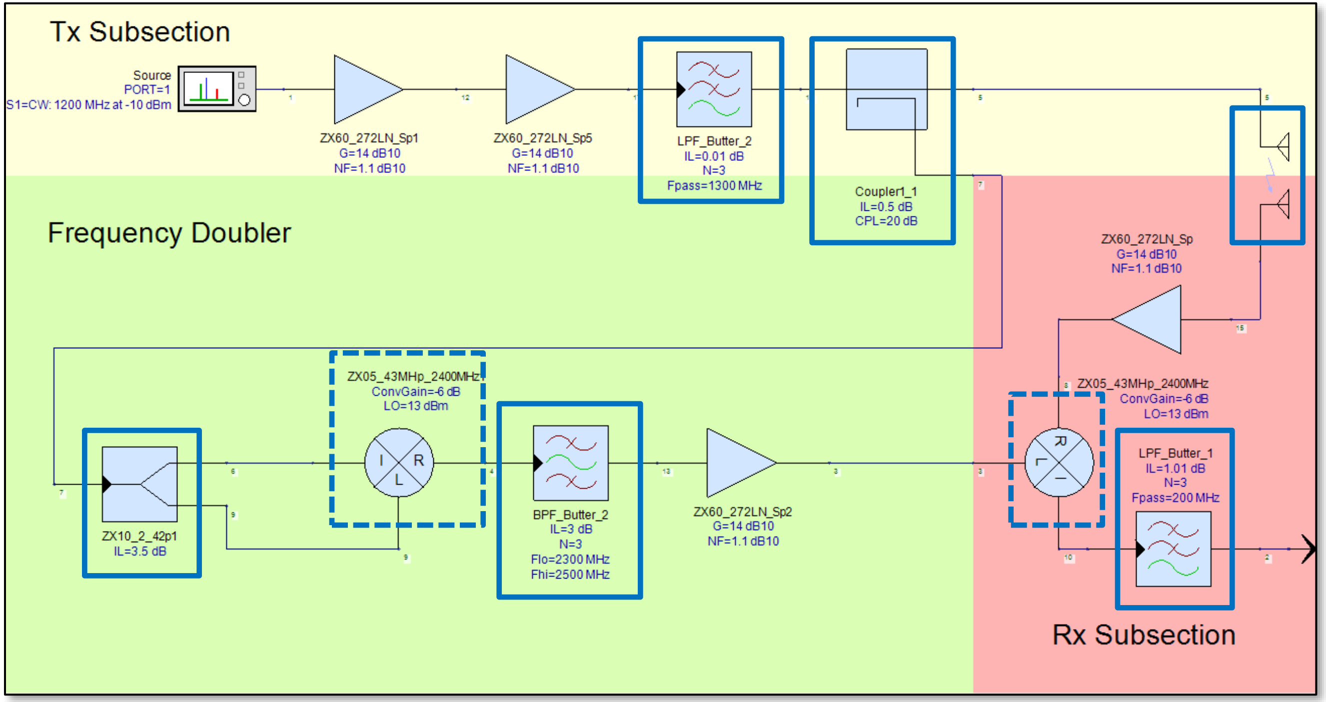


https://www.analog.com/en/design-center/evaluation-hardware-and-software/evaluation-boards-kits/EVAL-HMC994APM5.html#eb-overview

Planar Microwave Devices

Planar Microwave Devices

RF Passive Components



We can use PCBs to create passive microwave components

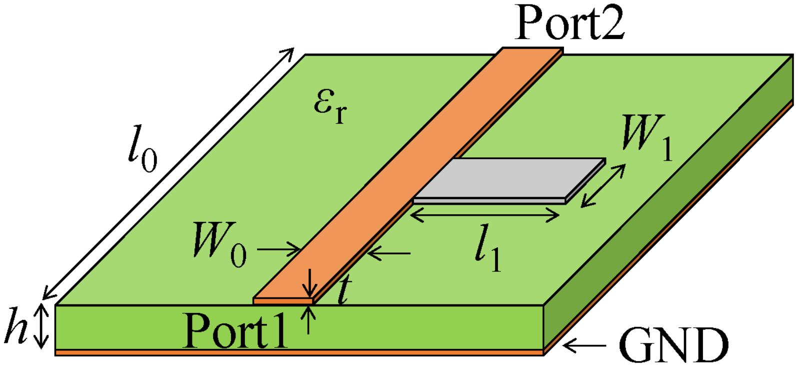
PCBs offer a cheaper medium for microwave device fabrication

Also allows for greater integration of an RF system onto a single board

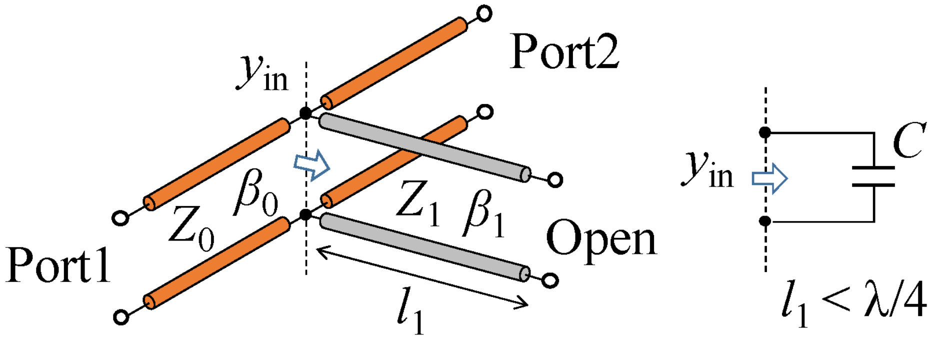
Microstrip Resonators

Open-Circuit Stub

A resonator can be created using a length of microstrip and a termination



(a)



(b)

Kusama, Y.; Isozaki, R. Compact and Broadband Microstrip Band-Stop Filters with Single Rectangular Stubs. Appl. Sci. 2019, 9, 248. https://doi.org/10.3390/app9020248

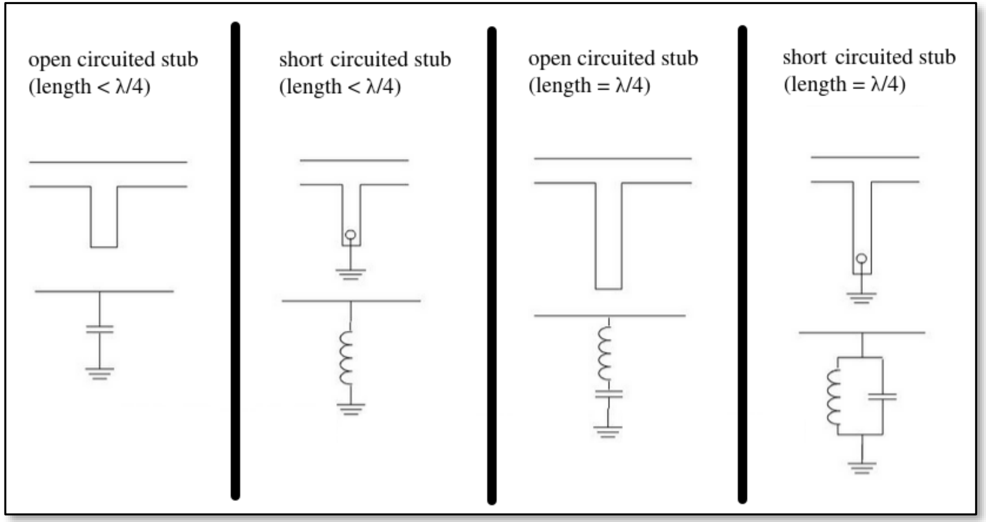
In this case, an open-circuit stub can be added

- **Acts like a shunt capacitor when its length is small**

Microstrip resonators

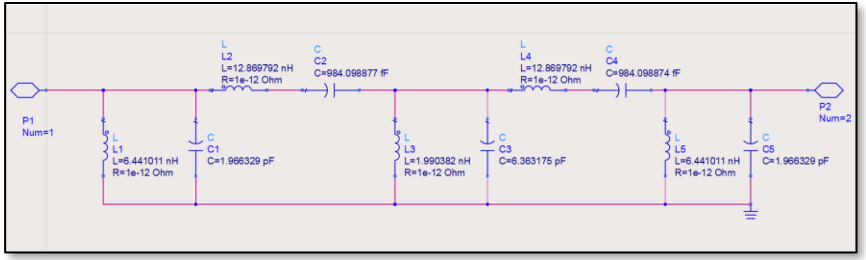
Filters

Different stub lengths and terminations can create different LC combinations



https://electronics.stackexchange.com/questions/460101/deriving-microstrip-stub-equations

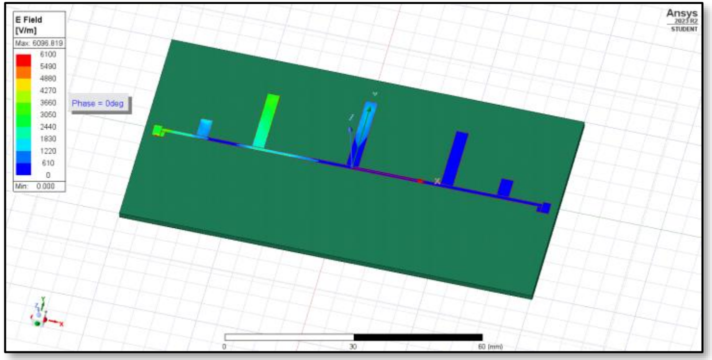
We can utilize this effect to create filters

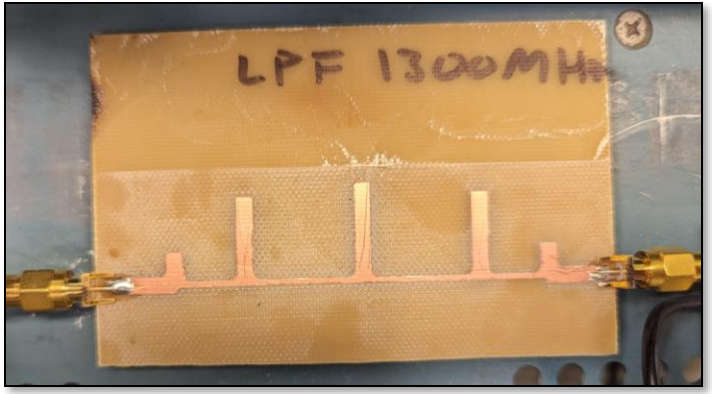


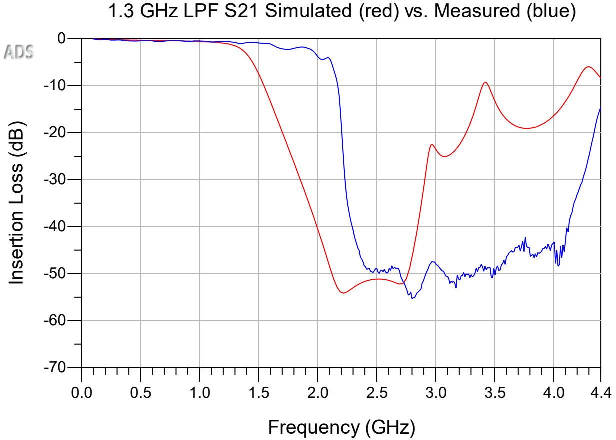
Microstrip Devices

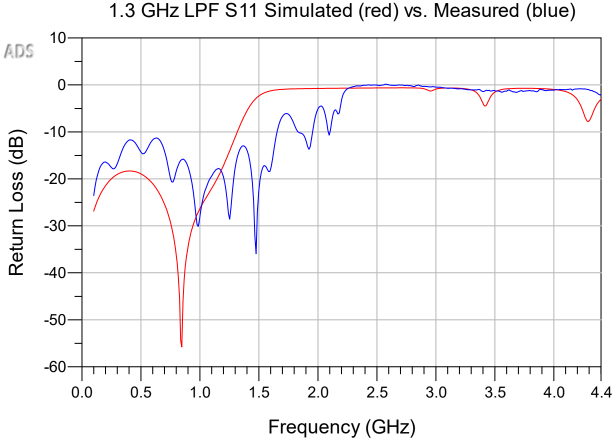
L-Band Stub Low Pass Filter

THE ART AND SCIENCE of PCB DESIGN
IAP 2025





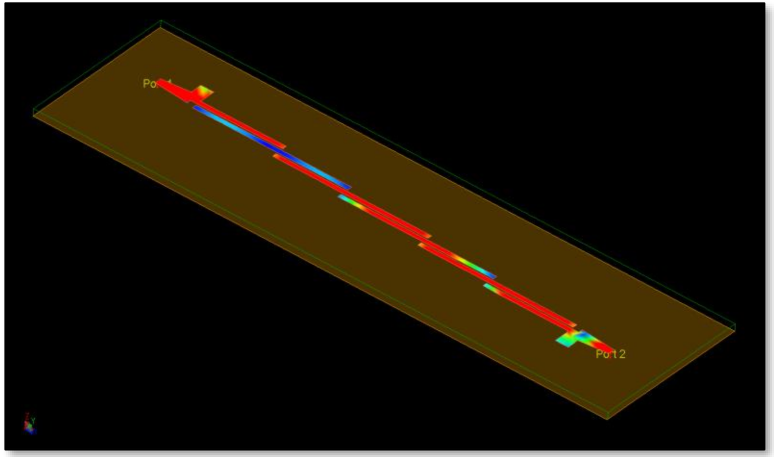

1.3 GHz LPF S21 Simulated (red) vs. Measured (blue)

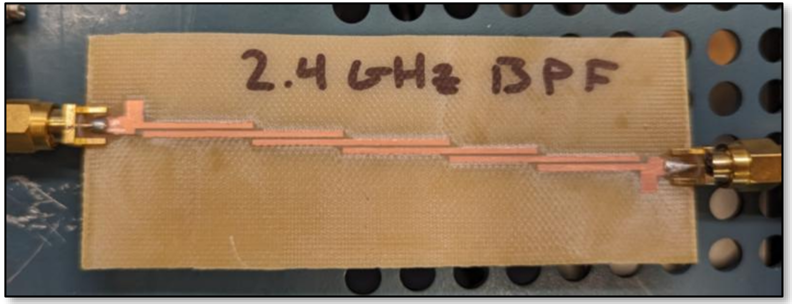

1.3 GHz LPF S11 Simulated (red) vs. Measured (blue)

Microstrip Devices

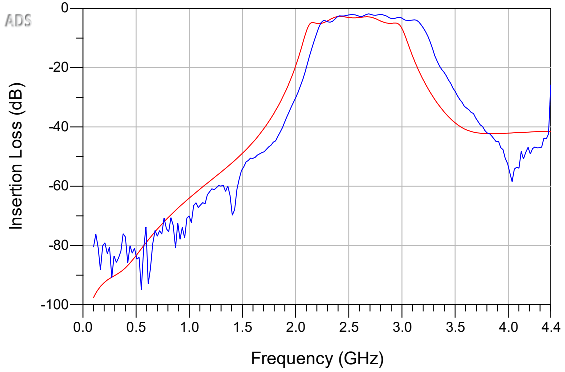
S-Band Coupled Line Band Pass Filter

THE ART AND SCIENCE of PCB DESIGN
IAP 2025

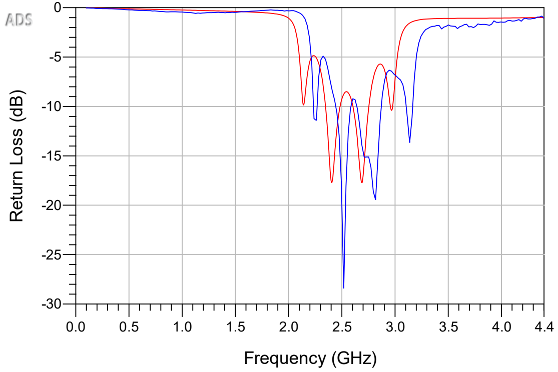




2.4 GHz BPF S21 Simulated (red) vs. Measured (blue)



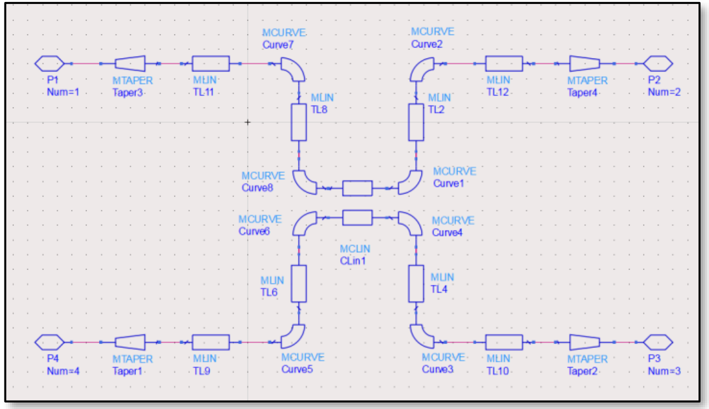
2.4 GHz BPF S11 Simulated (red) vs. Measured (blue)

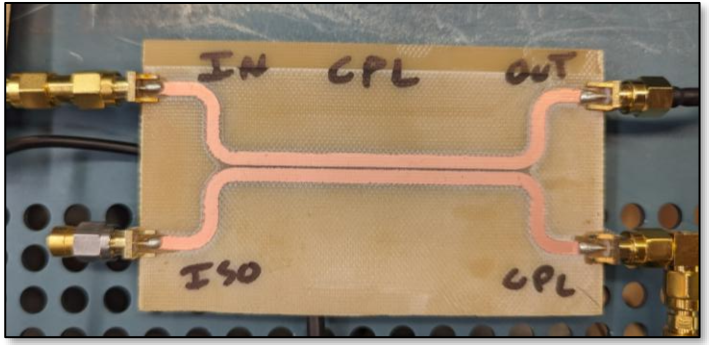


@ MIT

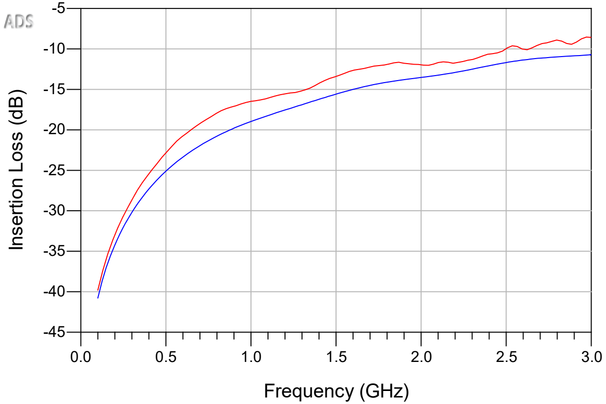
Microstrip Devices

S-Band Directional Coupler

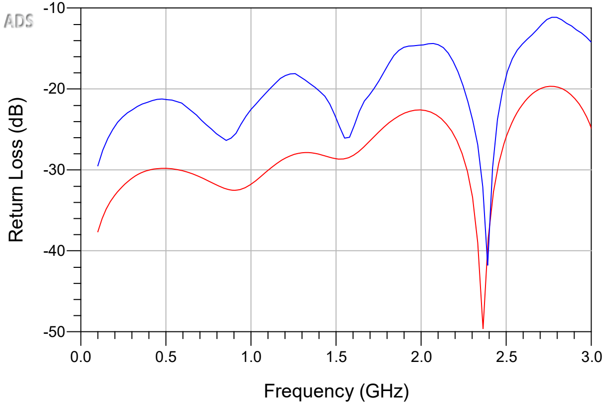




Directional Coupler Coupling Simulated (red) vs. Measured (blue)



Directional Coupler S11 Simulated (red) vs. Measured (blue)

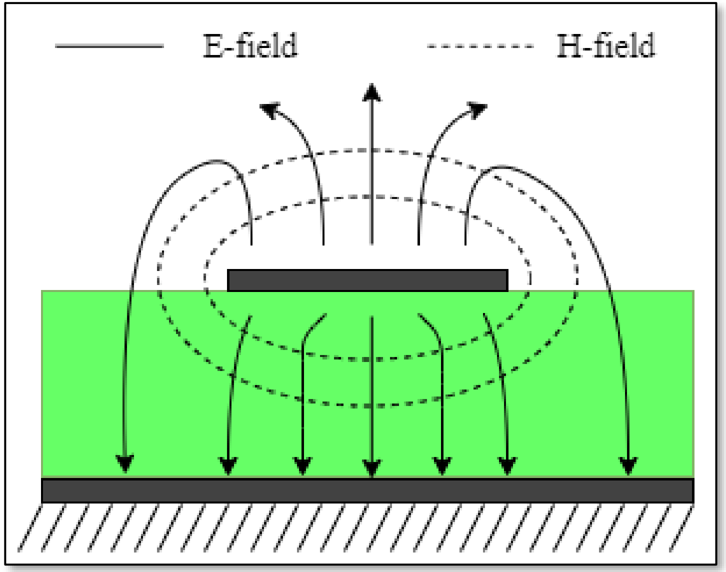


Same principle as the coupling microstrips!

Microstrip Devices

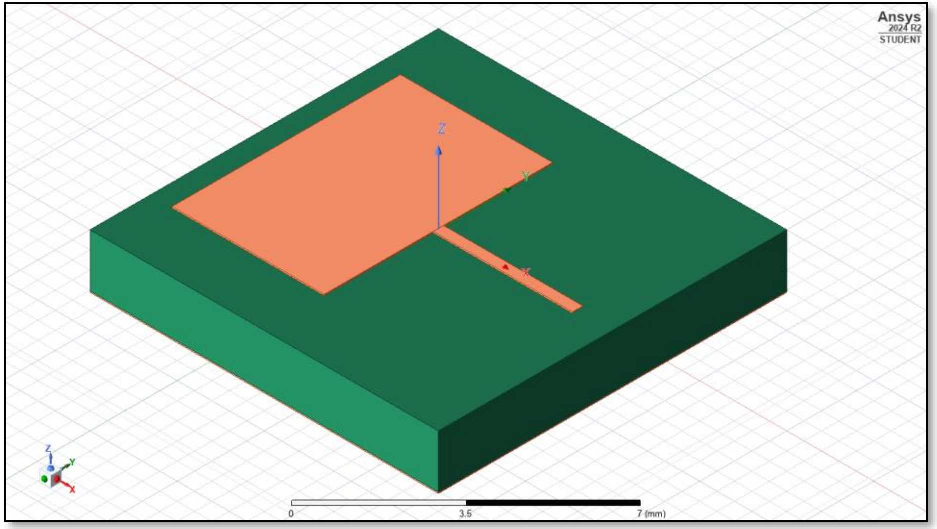
Patch Antennas

Recall how the microstrip has fringing fields?



https://www.signalintegrityjournal.com/articles/2378-measuring-the-bulk-dielectric-constant-dk-on-a-microstrip-with-a-tdr

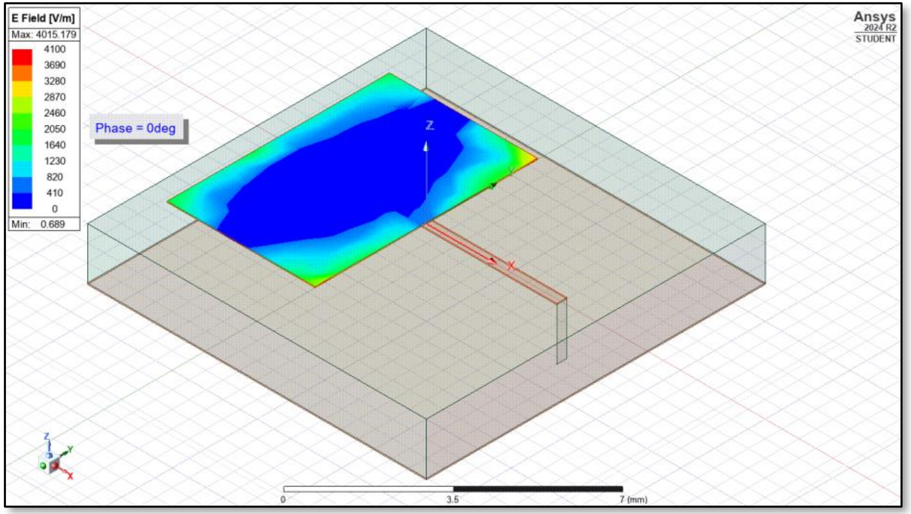
We can use the fringing fields to radiate power → Microstrip Antenna!



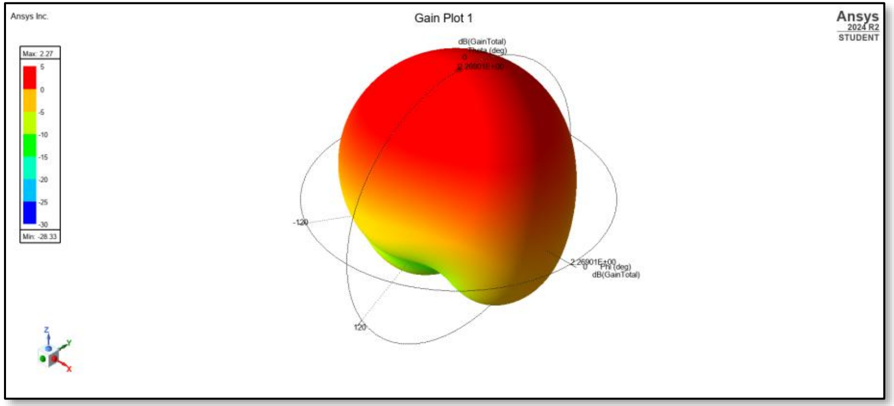
Microstrip Devices

Patch Antennas

Creating a resonating wave in the microstrip patch results in changing fringing fields

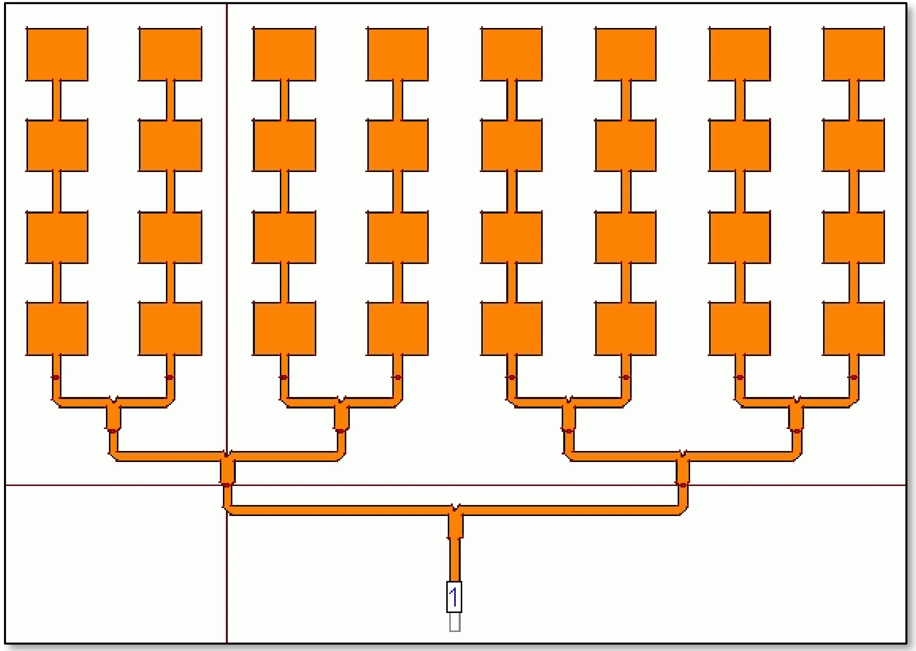


Fringing fields form a far-field radiation pattern

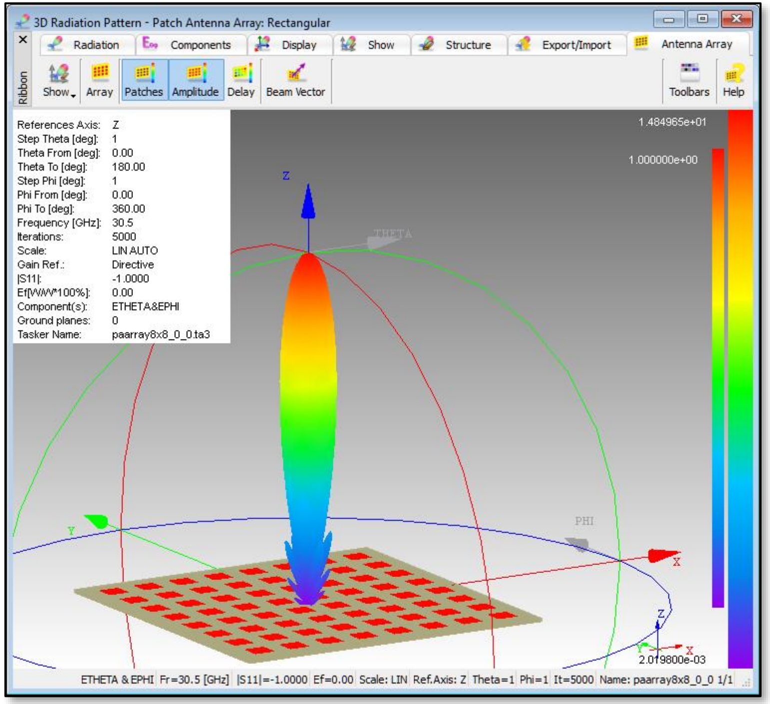


Microstrip Devices

Patch Antenna Arrays



https://www.researchgate.net/figure/Series-fed-patch-antenna-array-for-60-GHz-Both-developed-antenna-arrays-have-minimum_fig4_267718251

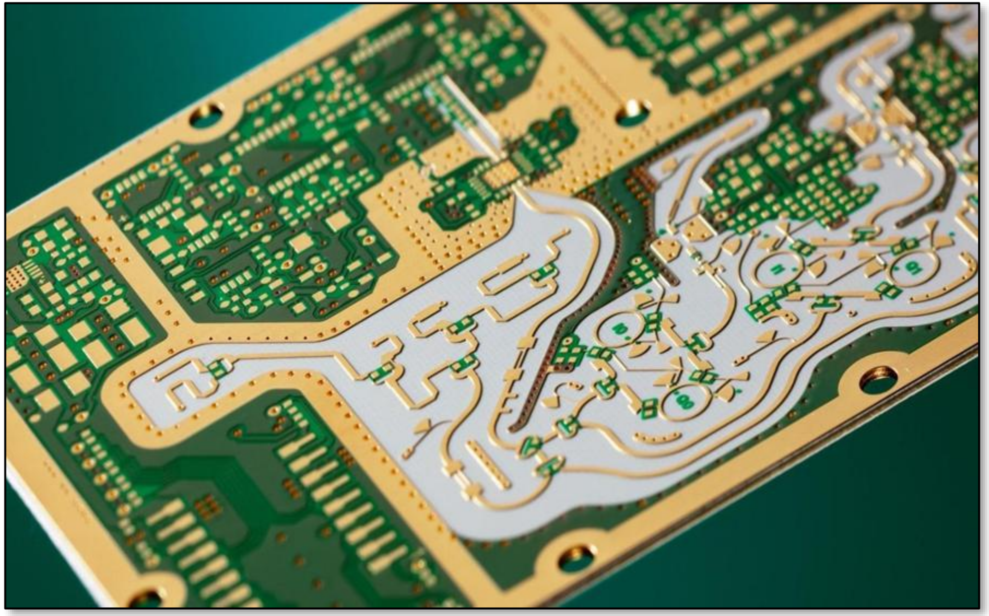


References Axis: Z
Step Theta [deg]: 1
Theta From [deg]: 0.00
Theta To [deg]: 180.00
Step Phi [deg]: 1
Phi From [deg]: 0.00
Phi To [deg]: 360.00
Frequency [GHz]: 30.5
Iterations: 5000
Scale: LIN AUTO
Gain Ref.: Directive
|S11|: -1.0000
Ef[W/W*100%]: 0.00
Component(s): ETHETA&EPHI
Ground planes: 0
Tasker Name: paarray8x8_0_0.ta3

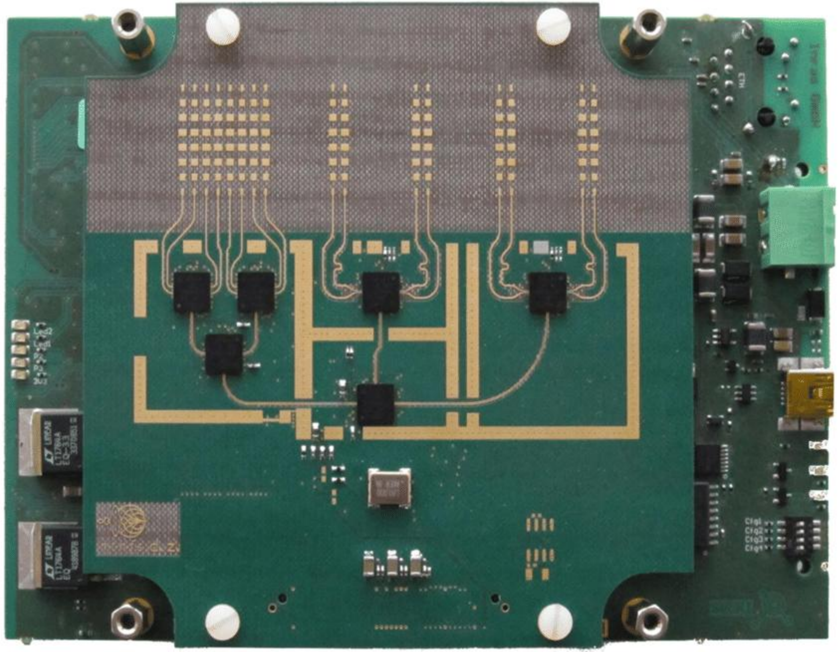
ETHETA & EPHI Fr=30.5 [GHz] |S11|=-1.0000 Ef=0.00 Scale: LIN Ref.Axis: Z Theta=1 Phi=1 It=5000 Name: paarray8x8_0_0 1/1

https://www.qwed.eu/QuickWave/help/qw-modeller_examples_guide/15_rectangular_patch_antenna_8x8_array.htm

Integrated Microwave Assemblies



https://www.viasion.com/radar-pcb/



https://www.researchgate.net/figure/mage-of-the-radar-system-The-PCB-on-top-is-the-RF-frontend-which-holds-the-RF_fig2_313686839