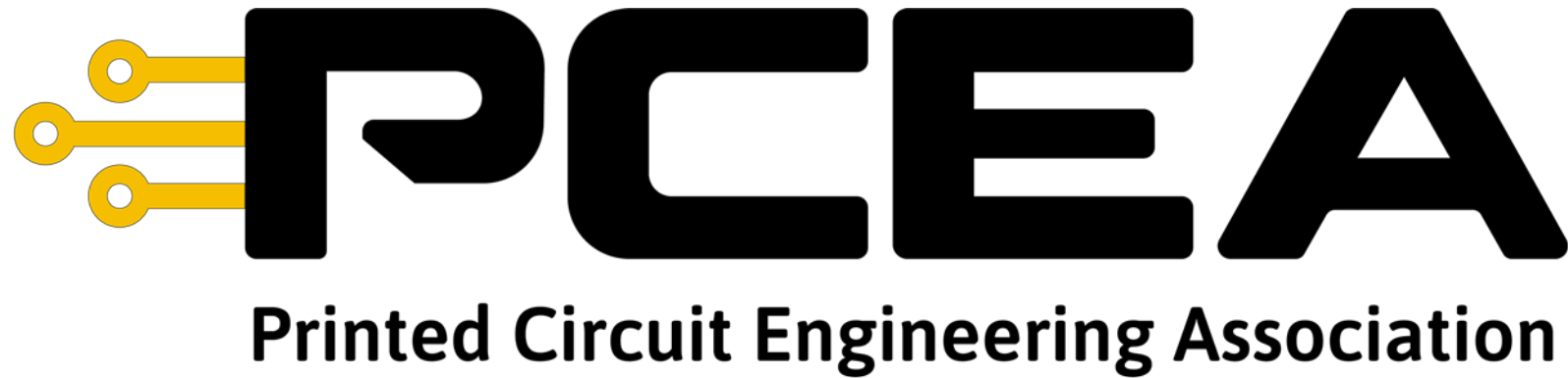


Orange County Chapter Virtual Meeting



May 19, 2021

Introducing Today's Speaker

Orlen Bates Sr. Field Applications Engineer



Orlen has been in the field of PCB design since 1973 and has been with EMA since 2002. He attended Vocational Tech School, earning a degree in Drafting & Design Technologies. With years of printed circuit (PCB), printed wire (PWB) design, DFM knowledge and high-volume manufacturing, Orlen is a highly skilled professional in the field of engineering physical design.

RF DESIGNS THE WAVE OF THE FUTURE

- WHAT ARE RF WAVES?
- BEST PRACTICES TO USE WHEN DESIGNING WITH RF SIGNALS.
- COMMON RF DESIGN PITFALLS AND HOW TO AVOID THEM.
- QUESTIONS YOU SHOULD ASK WHEN WORKING ON RF CIRCUITS.
- STRATEGIES TO MITIGATE RF CIRCUITS FROM RF SIMULATOR TO CAD TOOL.
- A LOOK AT



WHAT ARE RF WAVES?



- Radio frequency (RF):

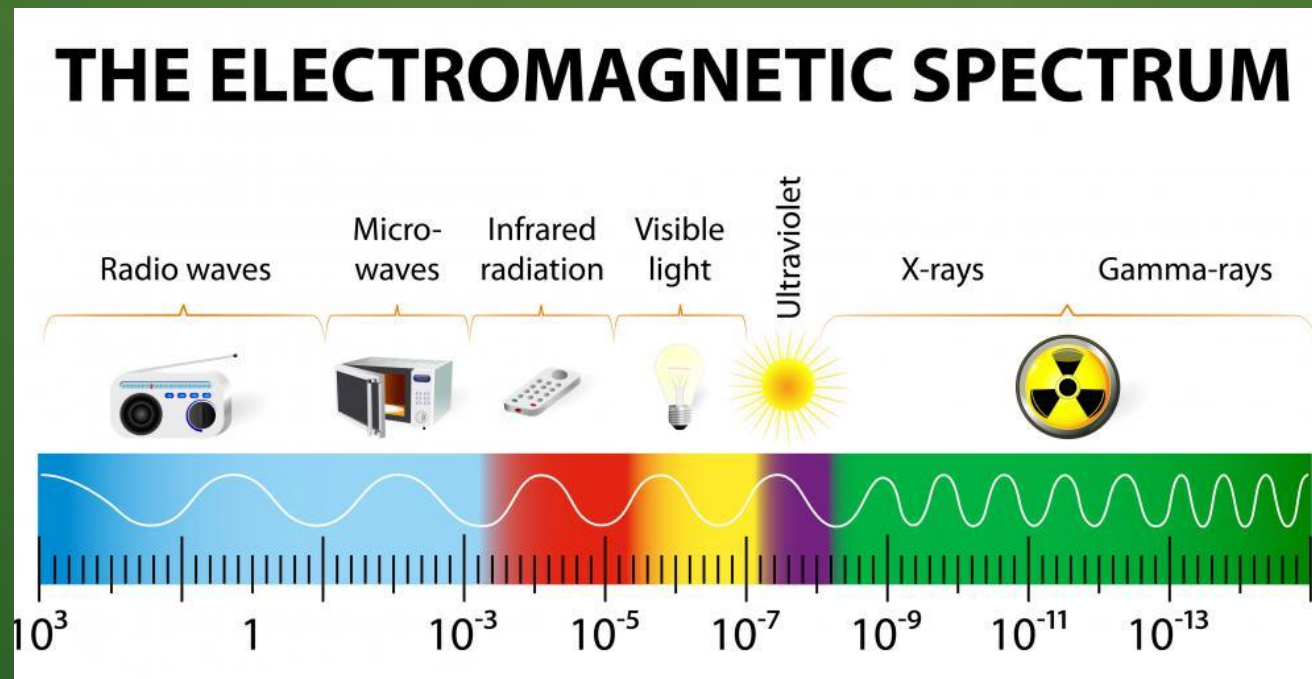
- Rate of oscillation of electromagnetic radio waves in the range of 3 kHz to 300 GHz, as well as the alternating currents carrying the radio signals.
- This is the frequency band that is used for communications transmission and broadcasting.

- Wi-Fi:

- An energy field that is transmitted as waves.
- The distance between Wi-Fi waves is shorter than that of radio waves but longer than microwaves, giving Wi-Fi a unique transmission band that can't be interrupted by other signals.

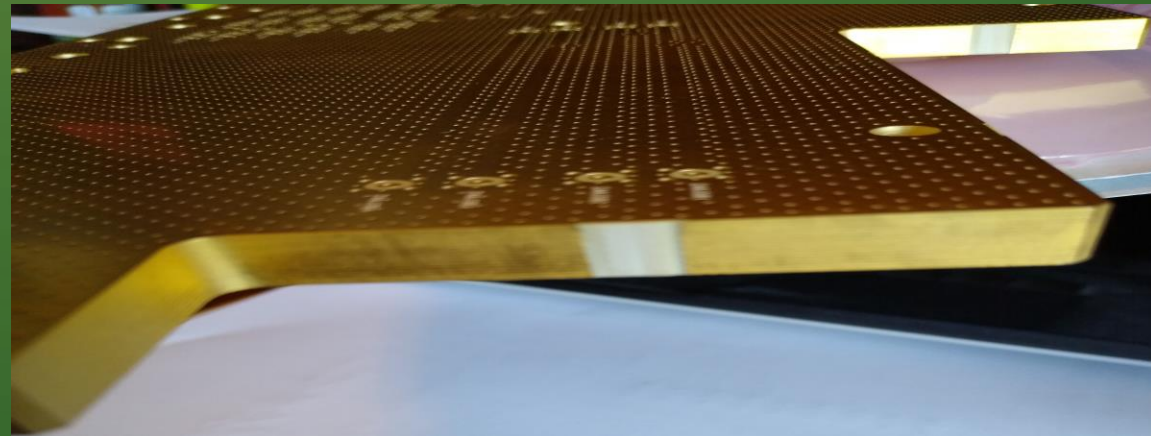
WHAT ARE RF WAVES?

- Microwave:
 - A reference to a class of waves, referenced as a microwave range or spectrum of frequencies that the Wi-Fi signal operates.
 - For example, the Wi-Fi signal from your laptop is often transmitted at 2.4 GHz, a frequency found within the microwave range, or spectrum of 300 MHz to 300 GHz.



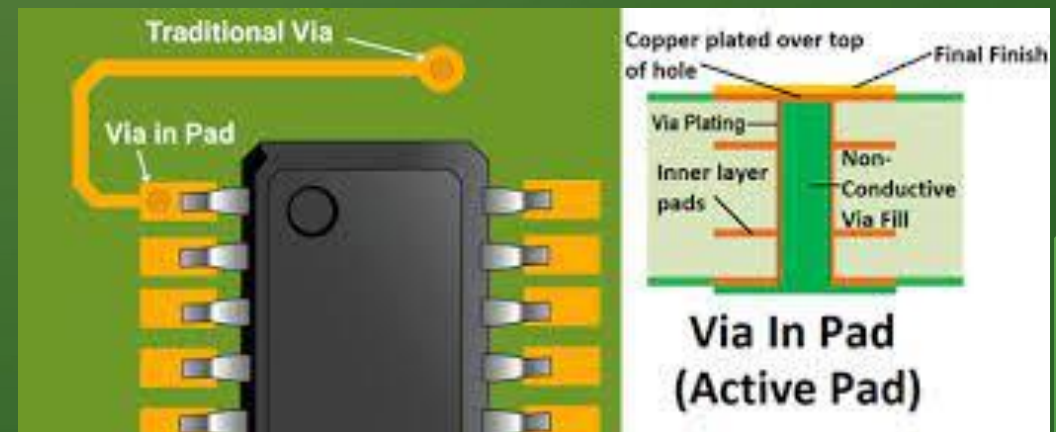
BEST PRACTICES TO USE WHEN DESIGNING WITH RF SIGNALS.

- **RF Ground, RF Ground and say it again RF Ground.**
- Component selection for RF should be SM and not Through Hole.
- RF components created from etch “copper” should be gold plated.
- The use of a multilayer PCB is best.
 - If your PCB design includes only two layer, the top layer should include the power shapes, RF signal lines and RF components. The bottom layer **must** be the RF ground plane.



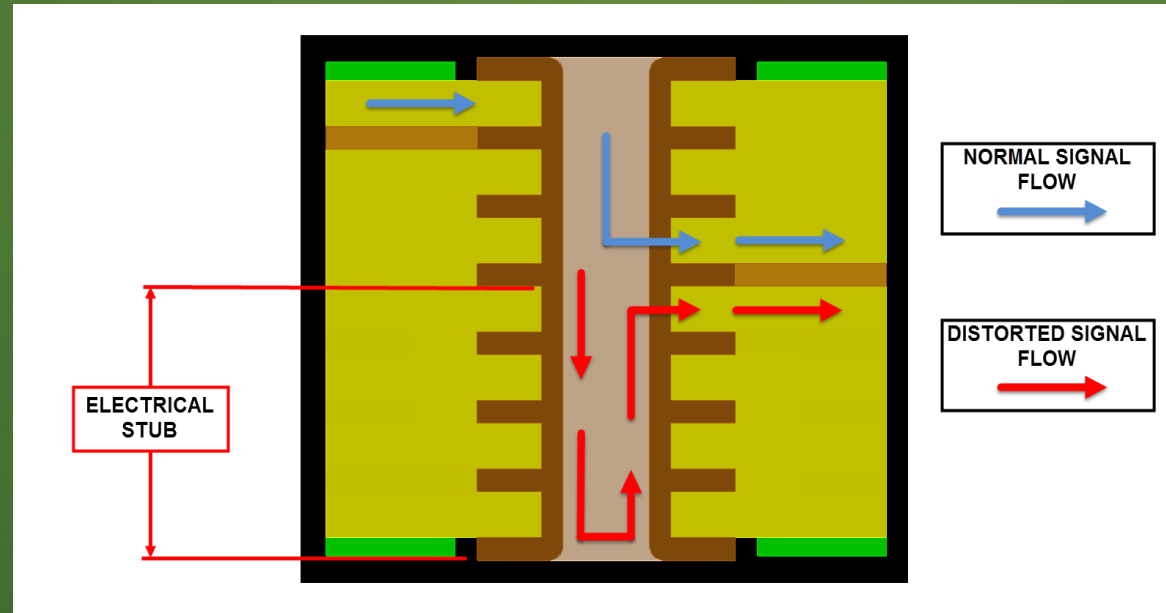
BEST PRACTICES TO USE WHEN DESIGNING WITH RF SIGNALS.

- Use least number of vias in RF routing, max number of vias in RF grounding.
- If you use multilayer PCB, route the short RF lines on the top layer.
 - To reduce the noise, create the power shapes between two ground layers. There must be **absolutely** the RF ground layer under the layer of RF signal lines.
- Connect the ground pins of the RF components to the RF ground plane as short as possible, vias in pad is best.
 - You should use multiple vias to reduce the unwanted effects of the RF ground.



BEST PRACTICES CONTINUES

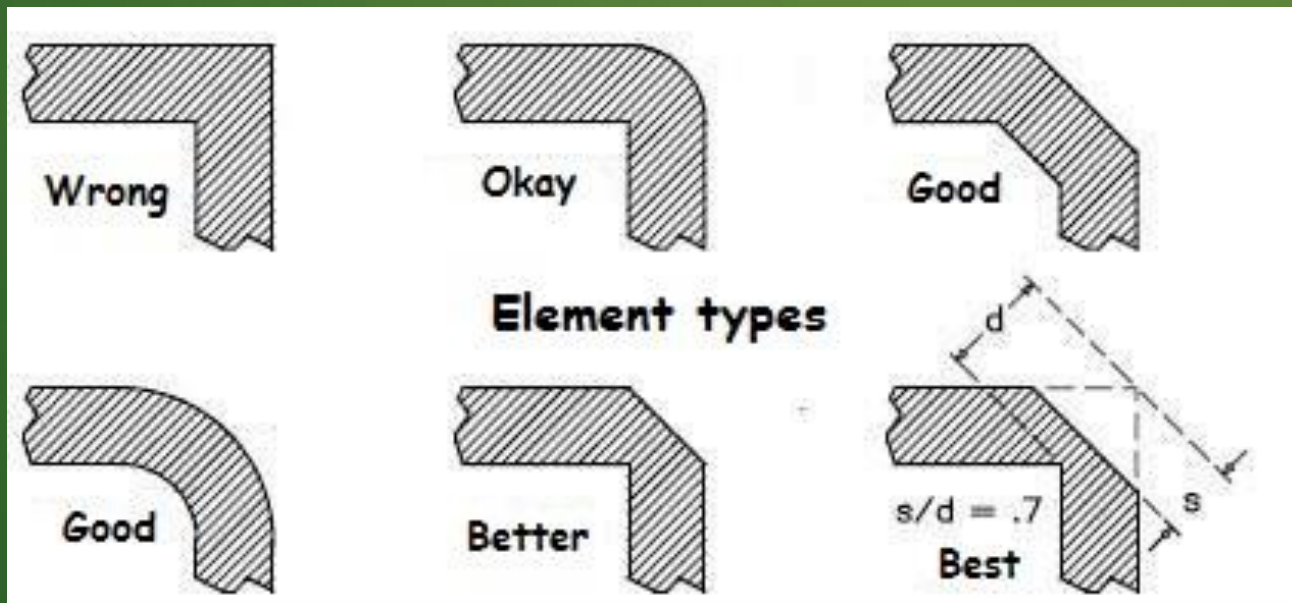
- Backdrilling should be used with through-hole component and vias. This will remove that unwanted antenna.



- Coils generate strong magnetic fields, place them isolated from each other to prevent the interaction. This includes purchased components and etch created coil or planar components.

BEST PRACTICES CONTINUES

- The proper corner routing are shown below.

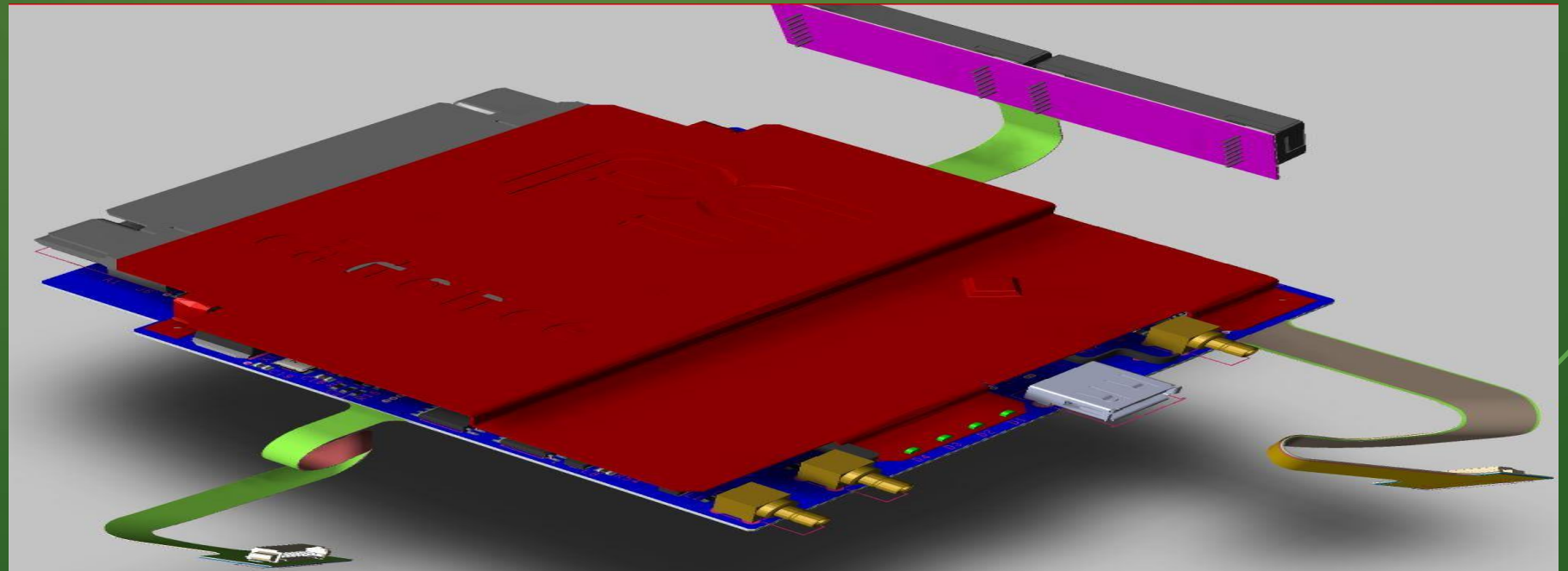
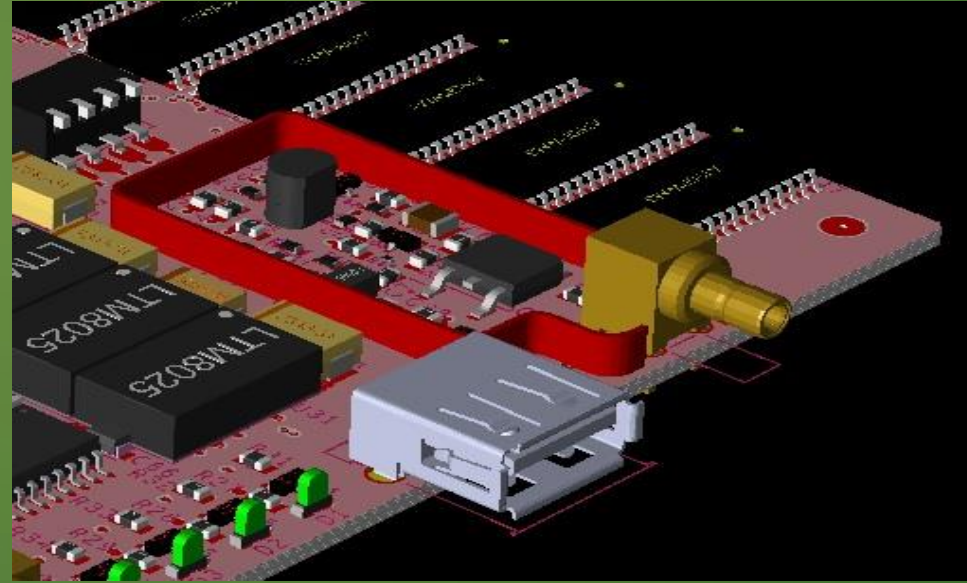


The screenshot shows the routing options menu in a PCB design software. The options are:

- Top (Act)
- Lyr2_Gnd (Alt)
- BACKDRILL-VIA-1-18_ (Via)
- Ground above: (dropdown)
- Ground below: LYR2_GND (dropdown)
- Net: DUMMY NET (dropdown)
- Trace Meander
- Element type: Curved (dropdown menu open showing: 90D-Unmitered, 90D-Mitered, 90D-Opt-Mitered, Curved)
- Line width: (dropdown)
- Radius: (dropdown)
- Line lock: 90 (dropdown) Relative
- Insert RF Component (button)
- Snap to connect point
 - Snap to pad edge
 - Variable line width
 - Taper width difference
- Physical length: 0.000000 MILS
- Working frequency: 1.00 GHz (dropdown)
- Electrical length: 0.000000 lambda
- Initialize clearance

RF GROUND, RF GROUND SAY IT AGAIN RF GROUND.

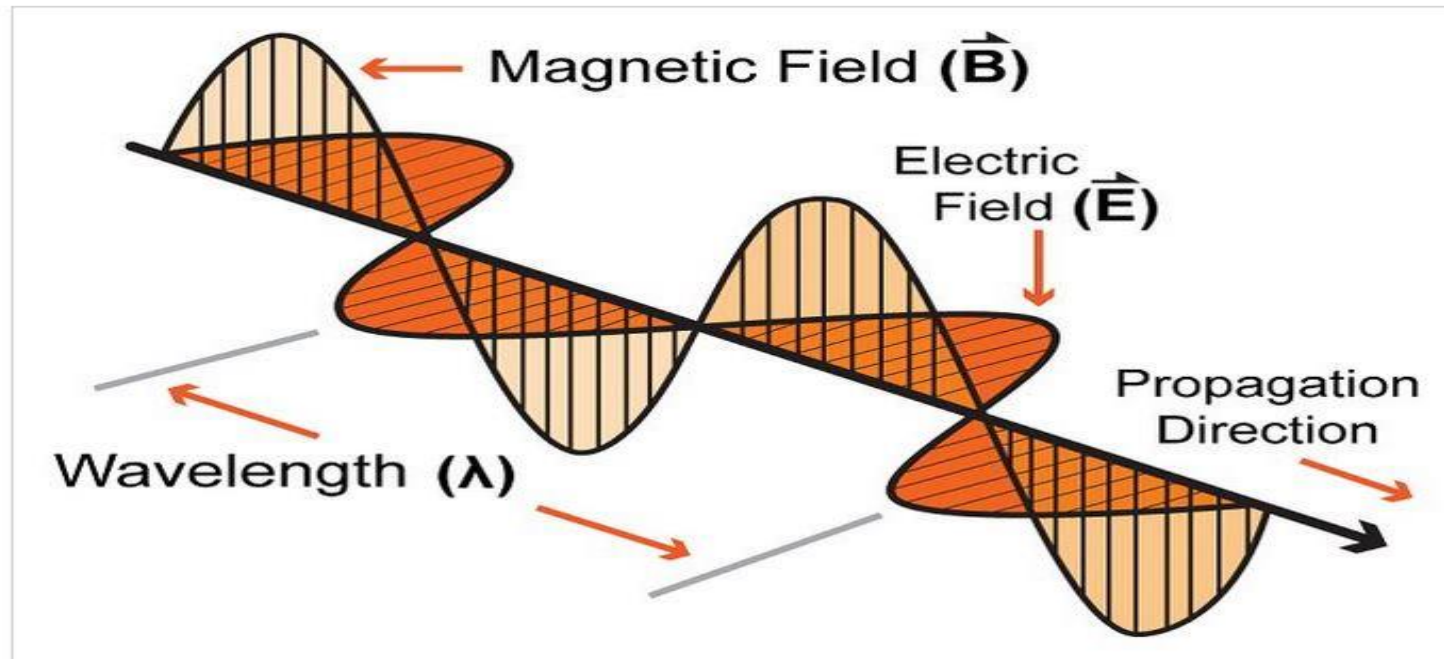
- There are times when a RF Fence is required.
- Also times a complete cover over the design is needed.



COMMON RF DESIGN PITFALLS AND HOW TO AVOID THEM.

- Respect the wave.

Provide a continuous, low-impedance ground plane whenever possible.

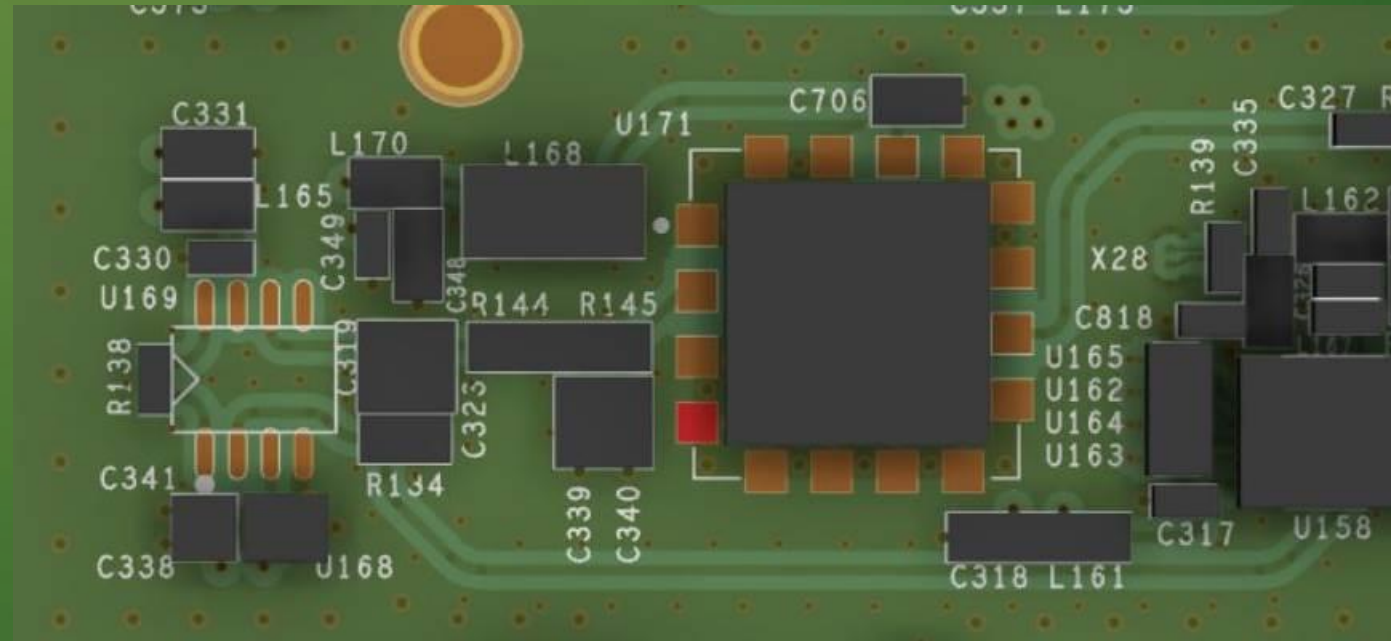
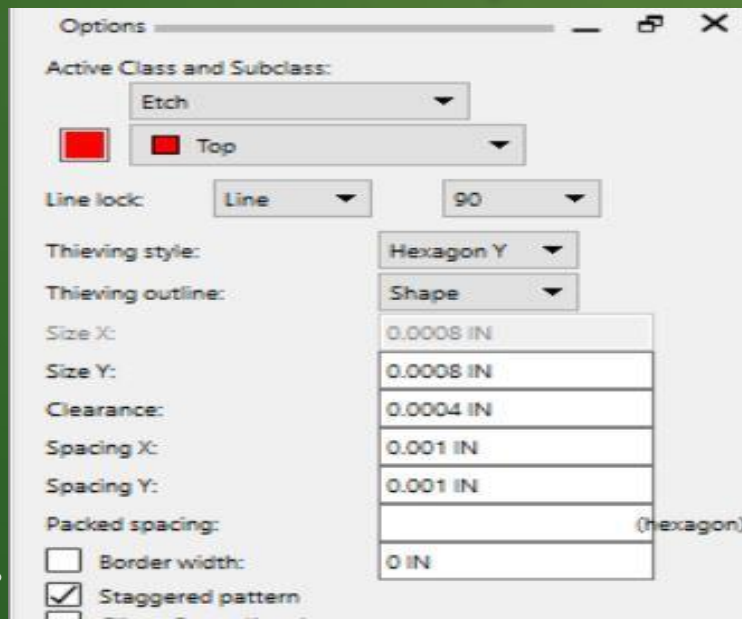


The electric and magnetic components of an electromagnetic wave are represented as perpendicular sinusoids.

The key to this propagation phenomenon is the self-sustaining relationship between the electric and magnetic components of electromagnetic radiation. A changing electric field generates a magnetic field, and a changing magnetic field generates an electric field. This mutual regeneration is manifested as a distinct entity, namely, an electromagnetic wave. Once generated, this wave will travel outward from its source, careening day after day, at the speed of light, toward the depths of the unknown.

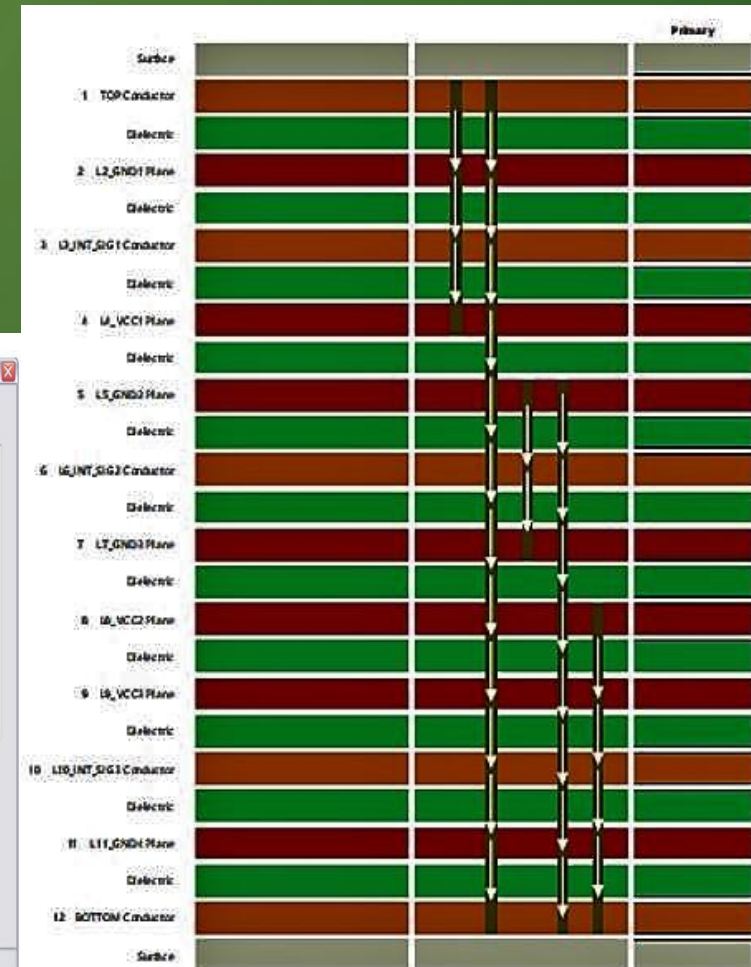
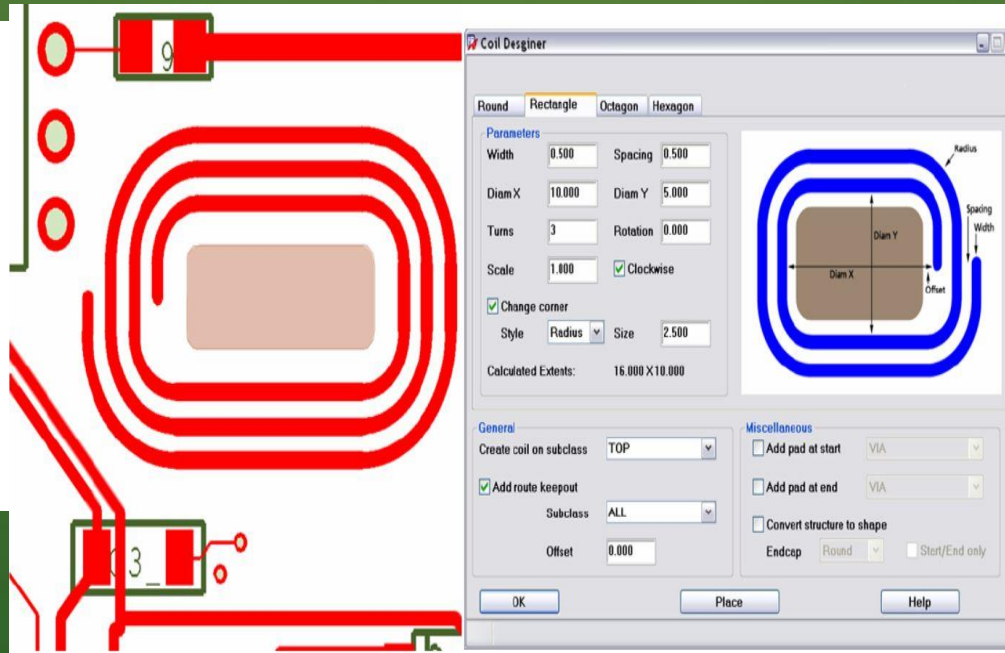
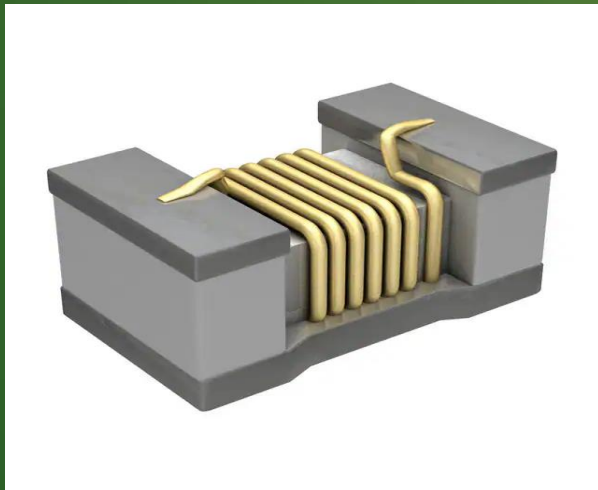
COMMON RF DESIGN PITFALLS AND HOW TO AVOID THEM.

- Ground both ends of copper pours and stitch many vias wherever possible.
- No copper inlands near the RF circuitry, do not use copper thieving near the RF circuitry.
- If the board includes multilayer ground planes, provide many ground vias wherever the signal trace makes a transition from one side of the plane to the other.
- Keep trace lengths to a minimum.
- Start the layout with critical circuits as close to the device as possible.



COMMON RF DESIGN PITFALLS AND HOW TO AVOID THEM.

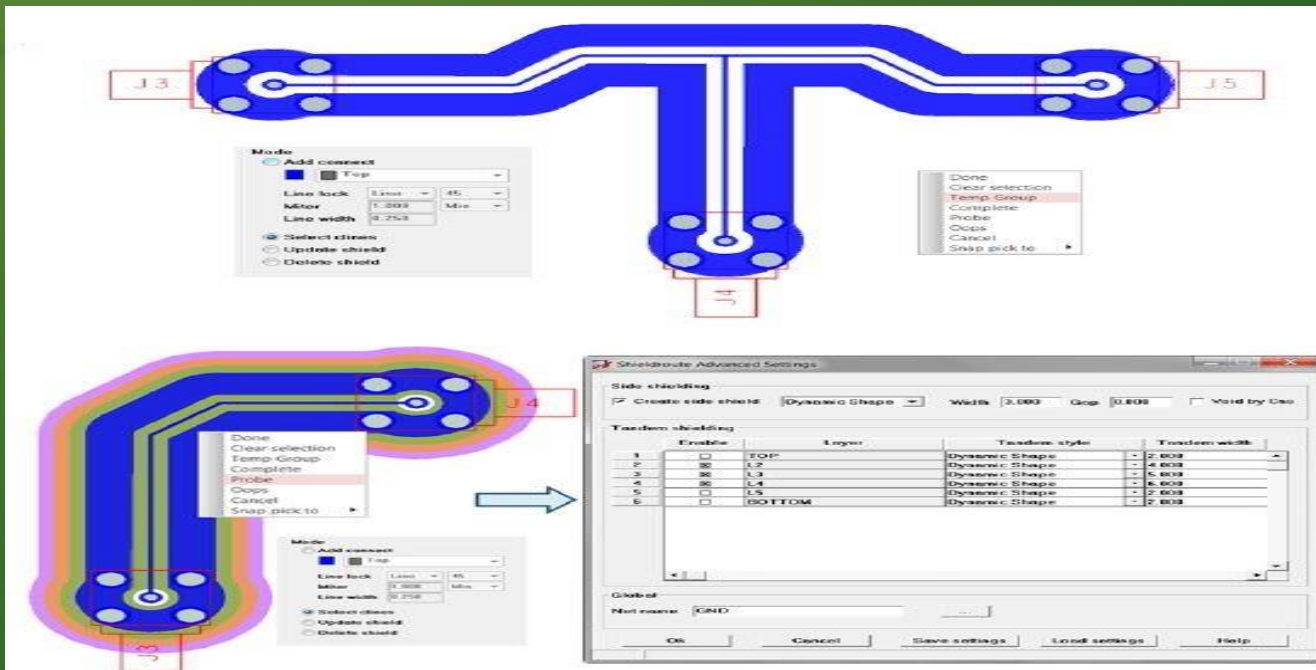
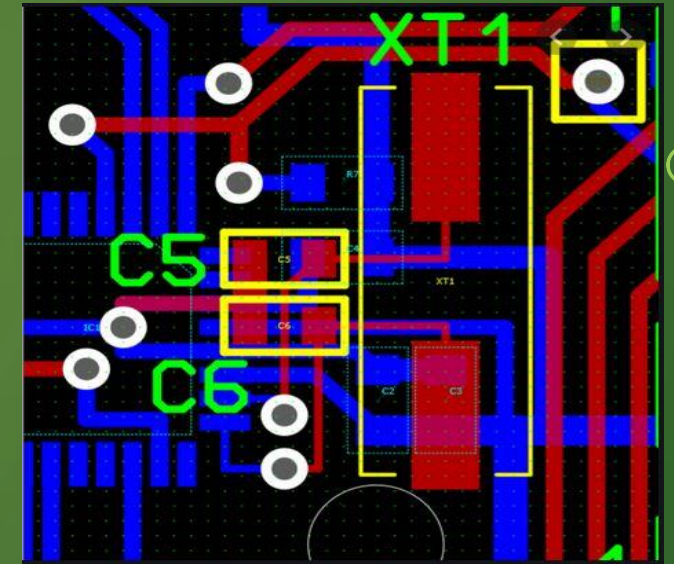
- Be sure to model the inductance of vias in sensitive circuits.
- Use isolated vias for separate portions of a filter or matching network.
- Note that thinner PCBs reduce the influence of parasitic inductance in vias. Backdrilling is needed in thick PCB stackups.
- Avoid the use of planar trace inductors.
- Use wire-wound inductors whenever possible.



COMMON RF DESIGN PITFALLS AND HOW TO AVOID THEM.

- Use short traces between the crystal and RF device.
- Keep interconnect traces separated as much as possible.
- If you suspect excessive shunt capacitance, vacate the ground plane under the crystal.
- Always provide continuous grounds under traces.
- Orient sensitive traces orthogonally.

If traces must be run in parallel, ensure adequate separation or use shield traces.



QUESTIONS YOU SHOULD ASK WHEN WORKING ON RF CIRCUITS.

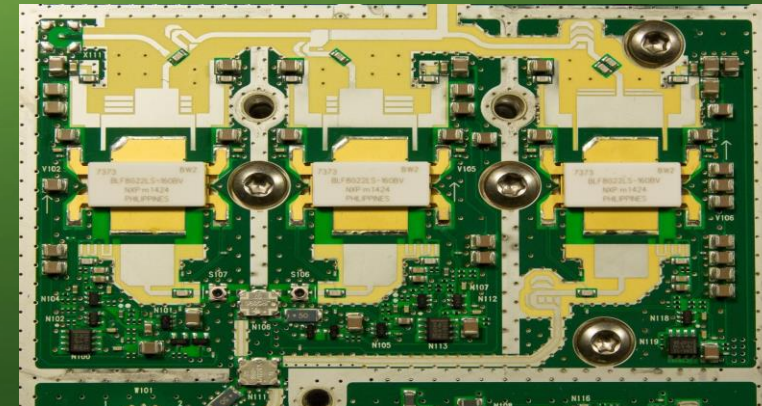
- Product usage for Health and Safety requirements.
- Product usage for environment conditions.



- Product lifecycle expectations.



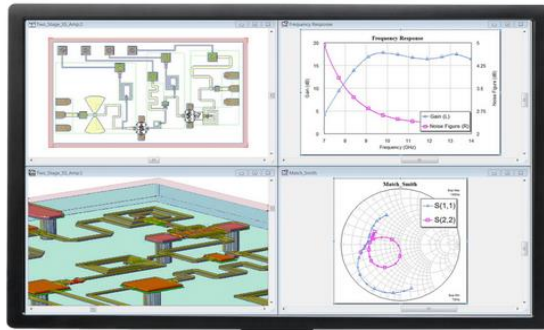
- PCB stack-up & material usage.



STRATEGIES TO MITIGATE RF CIRCUITS FROM RF SIMULATOR TO CAD TOOL.

- OrCAD PCB input with DXF or IFF.
- Allegro PCB RF Option IFF bidirectional.
- RF Simulator software:

1. AWR - Microwave Office



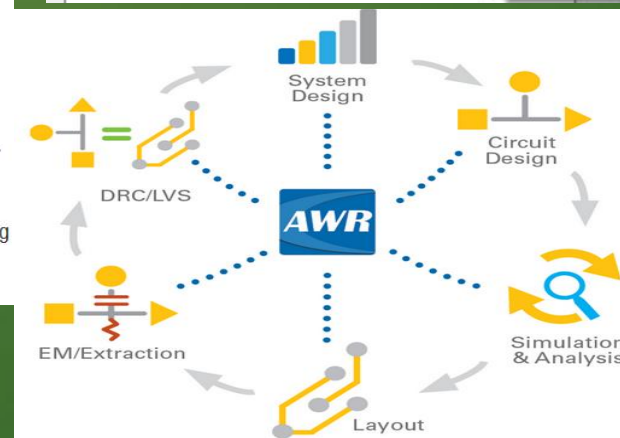
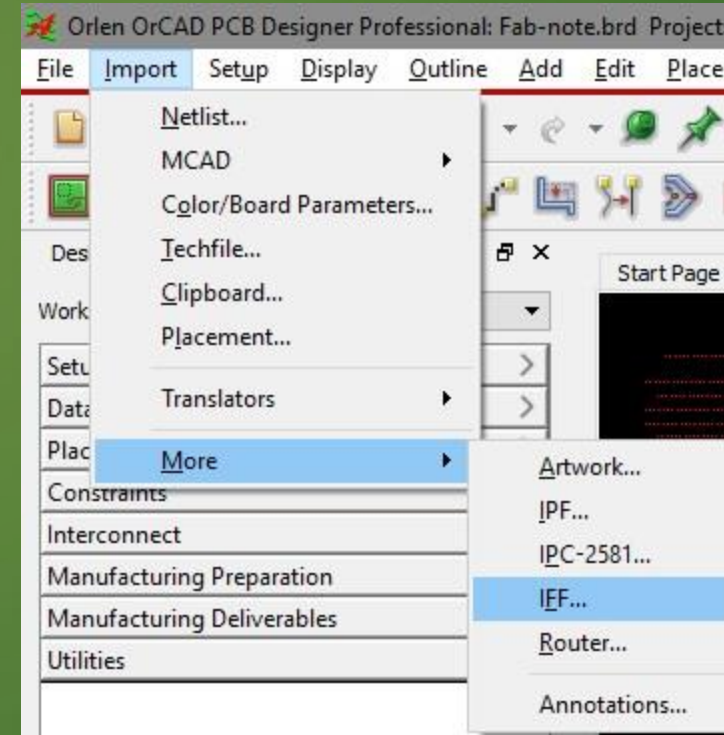
Microwave Office

Circuit design software that delivers accurate simulation for RF/microwave product development

Leading manufacturers use **Microwave Office** circuit design software to accelerate product development of high-frequency electronics. The intuitive interface, innovative design automation, and powerful harmonic-balance circuit simulation ensure greater engineering productivity and accelerated design cycles. RF-aware layout, high-frequency models, design automation, and powerful HB simulator ensure greater engineering productivity and speed up design cycles for today's most challenging applications.

2. Advanced Design System (ADS)

3. ANSYS HFSS



inspectAR

inspectAR, a CAD neutral tool, turns your mobile phone or external webcam into an intelligent view port and overlays an augmented reality containing essential schematic information on top of your board. This makes it easier to locate critical component information during testing and assembly, resulting in time savings of up to 50%



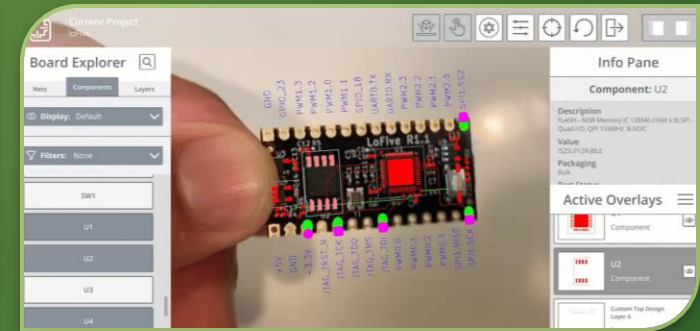
EFFECTIVE COLLABORATION

Provide your engineers with everything they need to have a truly collaborative work environment. Watch testing as it occurs and cut out errors



STREAMLINED DOCUMENTATION/DATA EXCHANGE

Eliminate wasted time finding datasheets, looking up test points in your CAD tool, and glancing back and forth to double check component placement with all your information in one.



FASTER TIME-TO-MARKET

Reach your consumers before your competition. Accelerate your time-to-market with up to a 50% reduction in assembly, test, and validation time.

THANK YOU FOR YOUR TIME !!

EMA Sales for your area:



Jen Gallanosa

Outside Sales Account Manager
(858) 663-6036

-  jeng@ema-eda.com
-  JenG@ema-eda.com
-  (858) 663-6036



Ron Canestro

Account Manager
716-341-5254

-  ronc@ema-eda.com
-  ronc@ema-eda.com
-  716-341-5254



Orlen Bates

Field Applications Engineer/Physical
Design Consultant
(585) 507-0332

-  orlenb@ema-eda.com
-  orlenb@ema-eda.com
-  (585) 507-0332

EMA AE

EMA Design Automation, Inc.

Electronics CAD Experts

30+ Years of EDA Experience

Sole NA Cadence Partner
Since 2003

Dassault VS Hi-Tech Partner
Since 2020

PCB is in our DNA

Complete PCB Design
Solutions

Staffed with Experienced
Design Engineers

Experienced Systems Integrator

Custom Deployments

Enterprise / PLM Integration

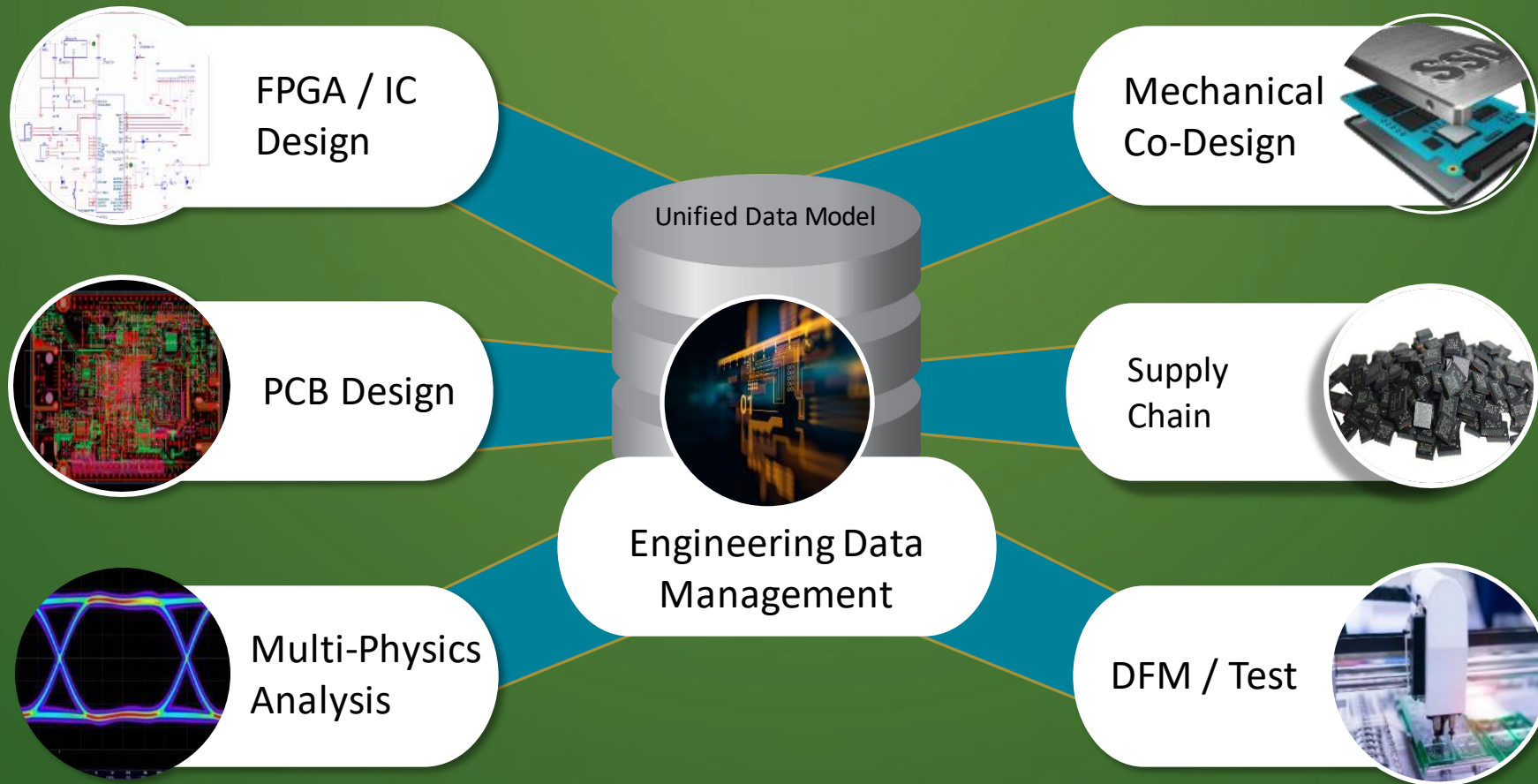
Process Assessments

100+
Employees

4,000+
Customers

“ Our customers
success is our
success ”

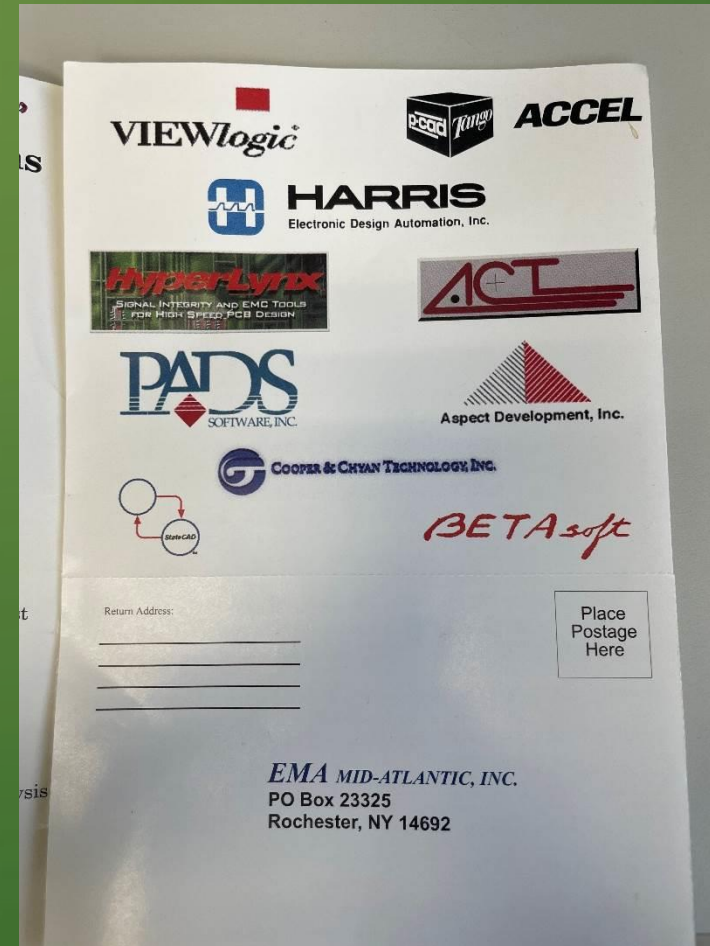
Complete Scalable Solutions



Training | Custom Development | Libraries

Our Mission

- Provide Complete Solutions to Solve Customer Problems
- Improve Productivity through Integration
- Enable Engineers with Data in Context
- Support the Design Community and Share Knowledge



Wrap Up

Thank you all for attending today's webinar

Special thanks to



for Hosting and Sponsoring today's event

Collaborate, Educate, and Inspire

Join the PCEA at www.pce-a.org

Please Join Our Chapter

scott.mccurdy.PCEA@gmail.com

Become a member sign-up at www.pce-a.org