Association Connecting Electronics Industries

IPC®

Impedance in a Flex Application: What Happens to the Signal?

Happy Holden
Chief Technical Officer
Foxconn

Photo not available

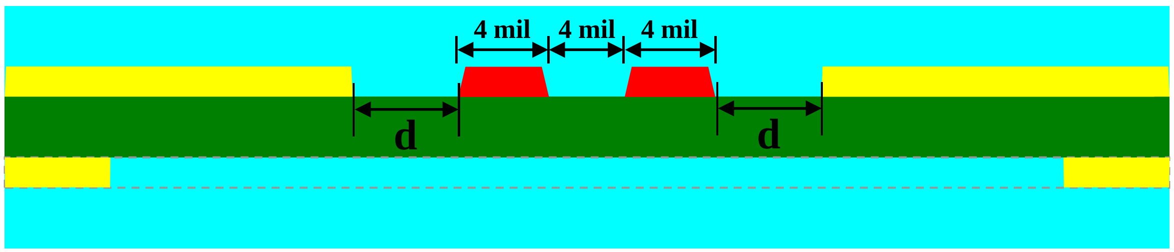
Happy Holden
Chief Technical Officer
Foxconn

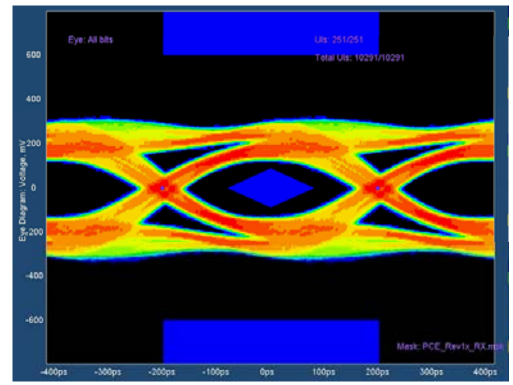
About the Author

Happy Holden is the Chief Technical Officer and VP for Foxconn-Hon Hai’s Mobile Interconnect Products Business Group headquartered in DaYuan, Taiwan. This is the world’s largest printed circuit manufacturer, with over 10 new facilities operating in Asia. Prior to joining Foxconn, he was the Senior PCB Technologist for Mentor Graphics System Design Division, the Advanced Technology Manager at Westwood Associates, and Merix Corporations. He retired from Hewlett-Packard after 28 years. Mr. Holden formally managed Hewlett-Packard's application organizations in Taiwan and Hong Kong. His prior assignments with HP had been as director of PCB R&D and PCB Engineering Mgr. He holds degrees in Chemical Engineering and Computer Science. He is a member of the IPC, TPCA, SMTA, IMAPS and the IEEE.

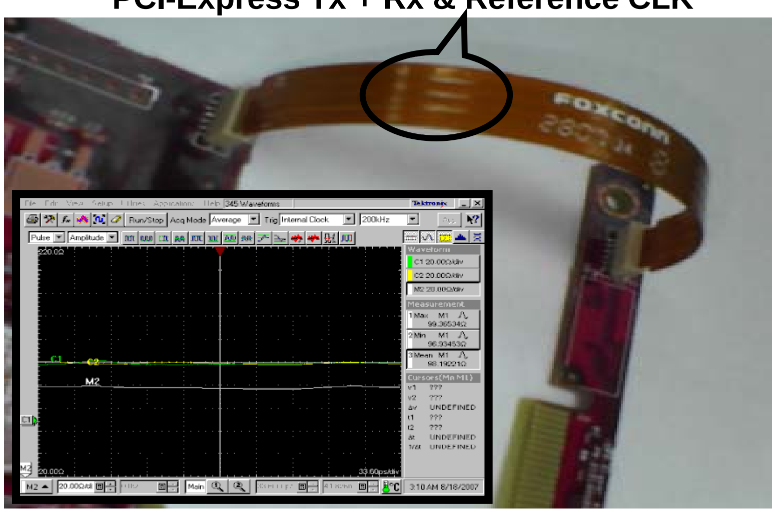
Flexible Printed Circuit Board

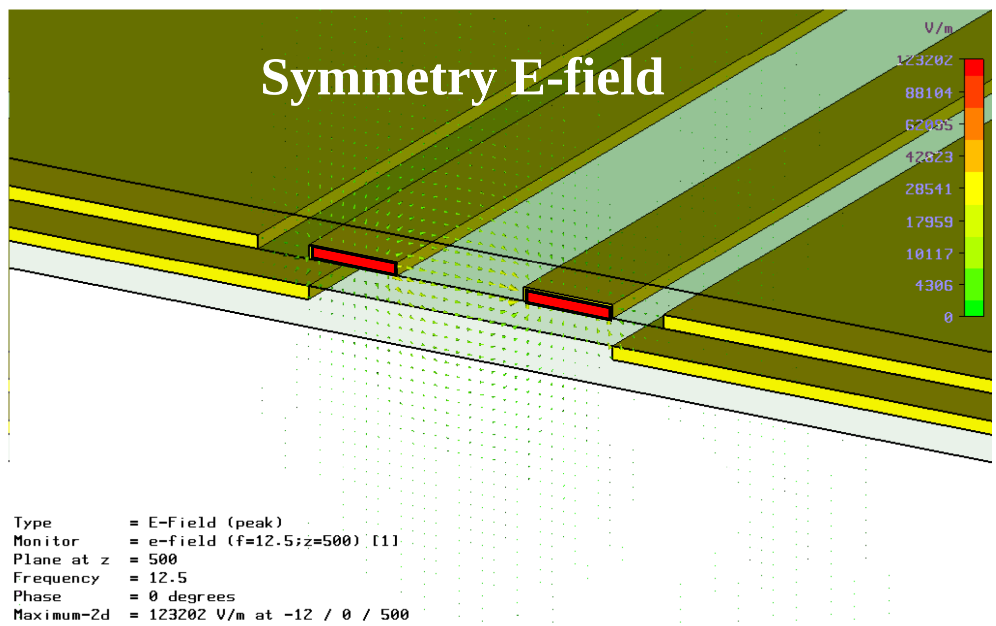
Impedance For Flex: Challenges For Dynamic Flex





PCI-Express Tx + Rx & Reference CLK





OUTLINE

Impedance Requirements

- Conventional Stripline, Mesh and Coplanar Structures
  - Microstrips (SE & Diff)
  - Coated & Embedded (SE & Diff)
  - Mesh Ground
  - Coplanar
- Dynamic Flex - Hinges
- Novel Structures
  - Mesh Ground
  - Coplanar w/o Ground
  - Broadside Coupled w/o Ground

High-Speed Flex Transmission Lines

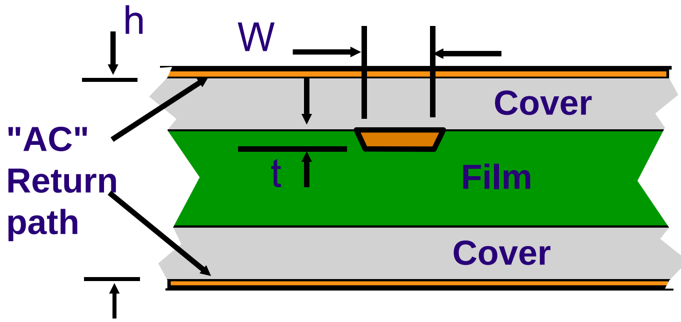
Signal Interfaces

- 1394 (110 Ohm)
- PCI-Express Gen1 (100 Ohm)
- USB2.0 (90 Ohm)
- Video (90 Ohm)
- PCI-Express Gen2 (85 Ohm)
- Memory (75 Ohm)

Conventional Stripline

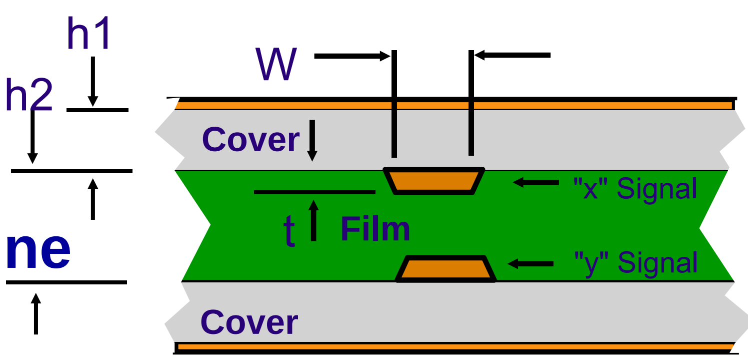
Stripline Transmission Lines

Stripline



$$Zo=\frac{60}{\sqrt{Er}}\ln\left[\frac{4h}{0.067\pi\, w\,(0.8+t/w)}\right]$$

Dual Stripline



Zo=80*ln[1.9*(2h1+t)/(0.8w+t)]*(1-B/(4(h1+h2+t)))/Er^0.5

Design Consideration

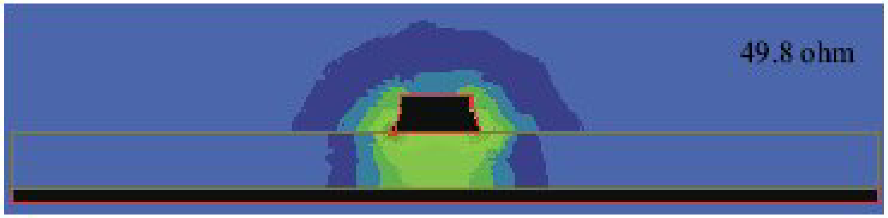
General

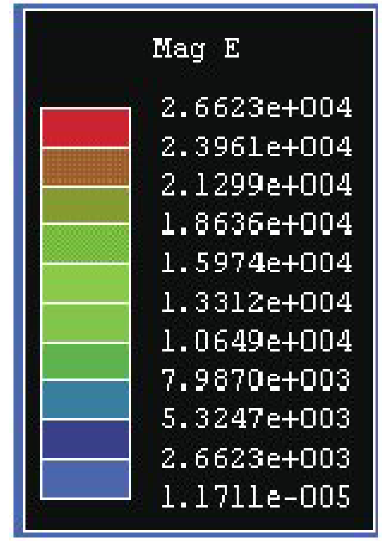
- **Propagation velocity dependant on dielectric constant of laminate**
- **Signal has two effective return paths**

Dual Stripline

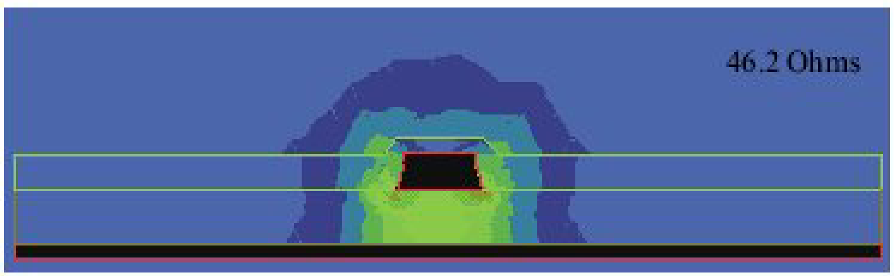
- **Parallel circuits traces between layers can result in excessive croasstalk**
- **Orthagonal crossings can result in:**
  - **Coupling in sensitive circuits (analog)**
  - **Reduced impedance**
  - **Increased propagation delay**

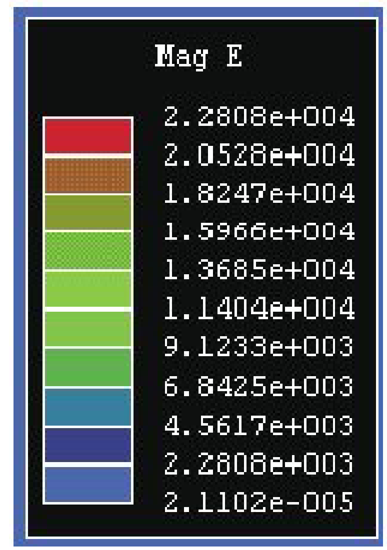
Field Distribution: Microstrip





Microstrip: Surfacce trace 0.005" wide, 0.002" thick

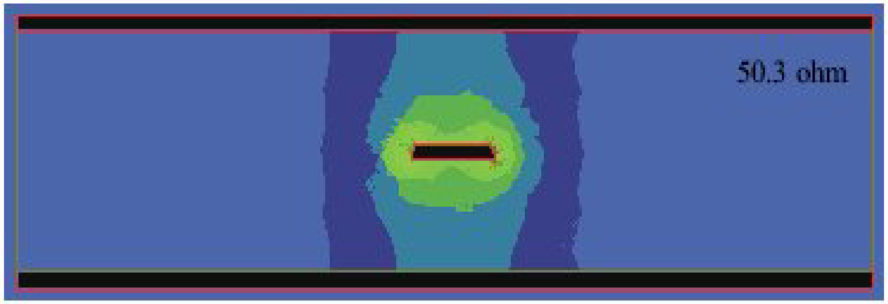


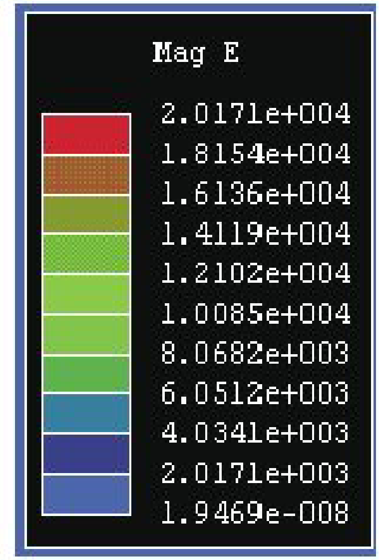


Embedded Microstrip (Soldermask): Surfacce trace 0.005" wide, 0.002" thick

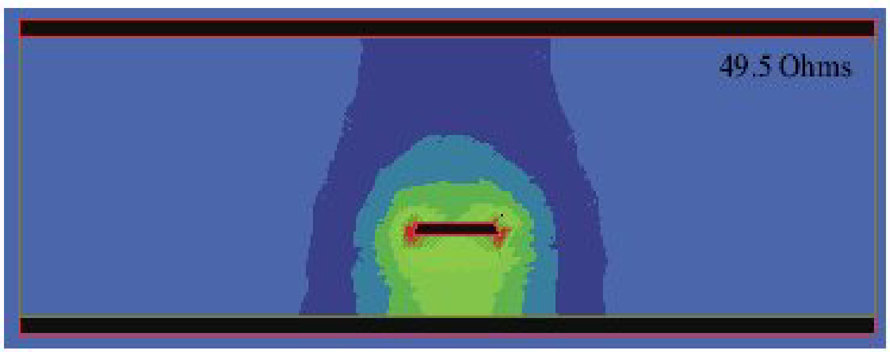
Circuit construction: FR-4. er=4.3

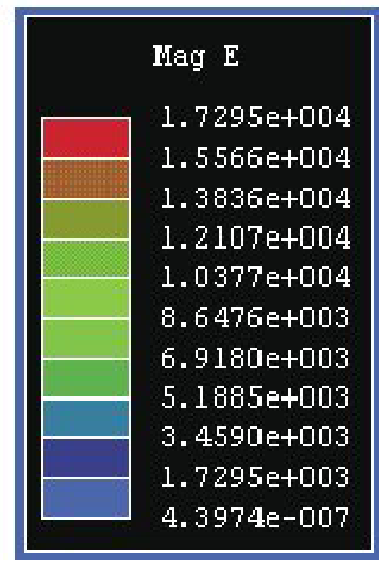
Field Distribution: Stripline





Stripline: 0.005” trace ½ oz copper

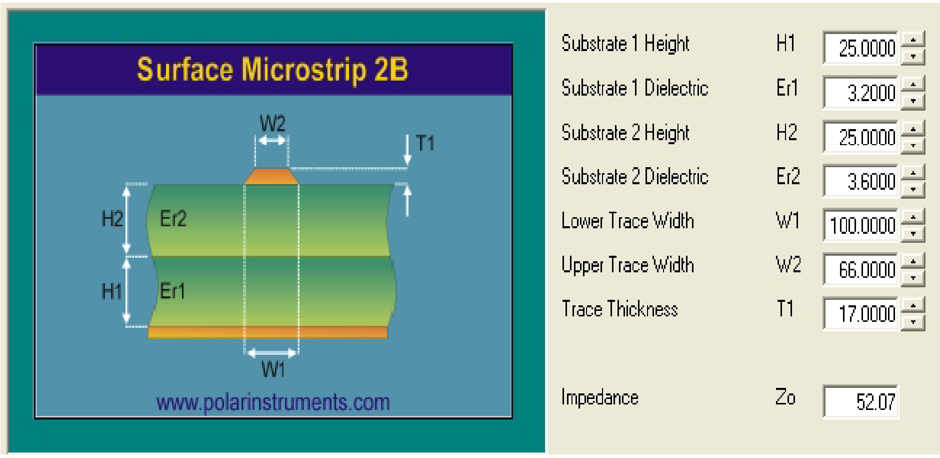


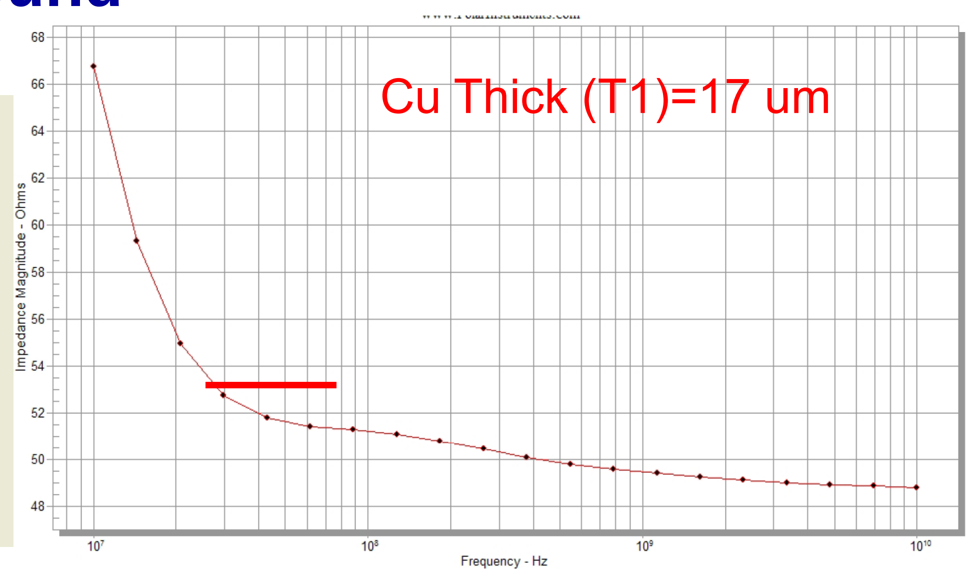


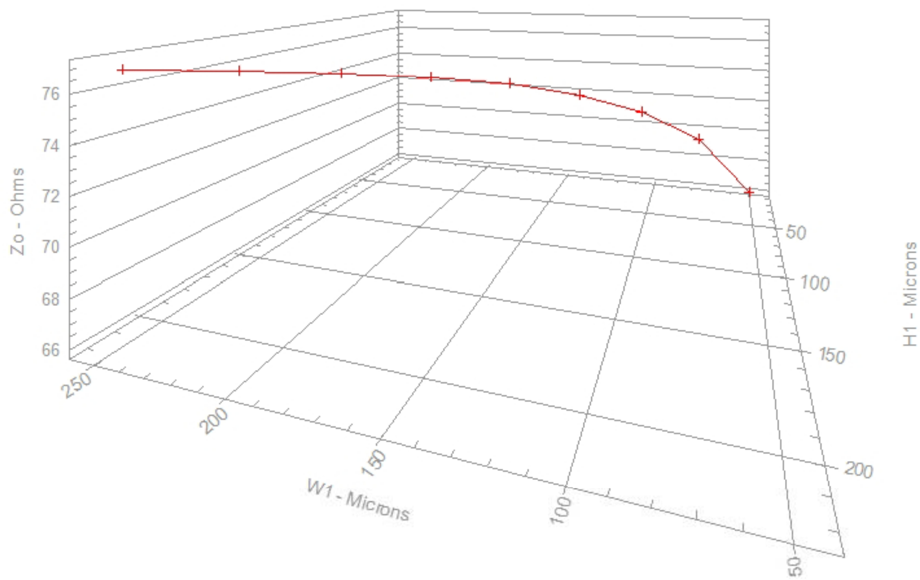
Asymmetric Stripline: 0.005” trace ½ oz copper

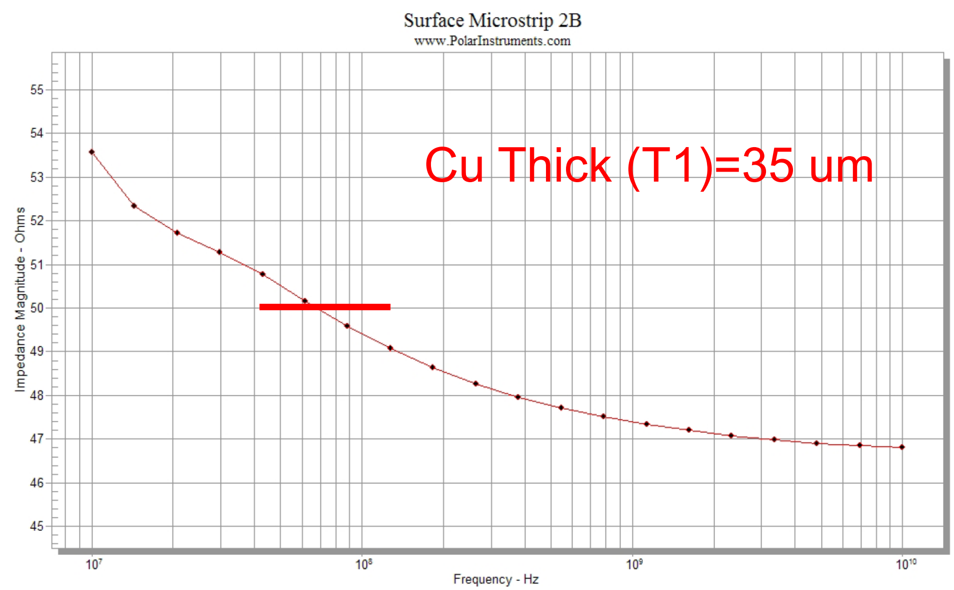
Circuit construction: FR-4, er=4.3

Surface MicroStrip With Ground

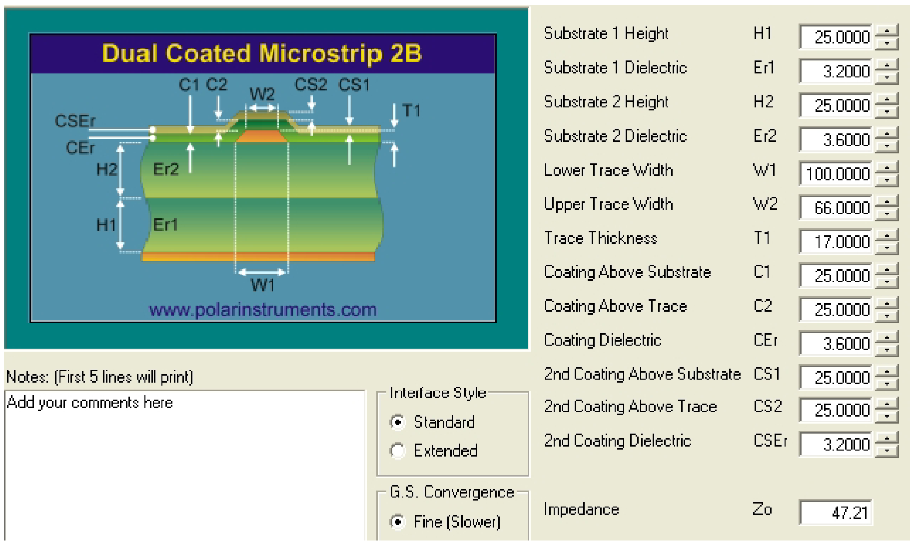


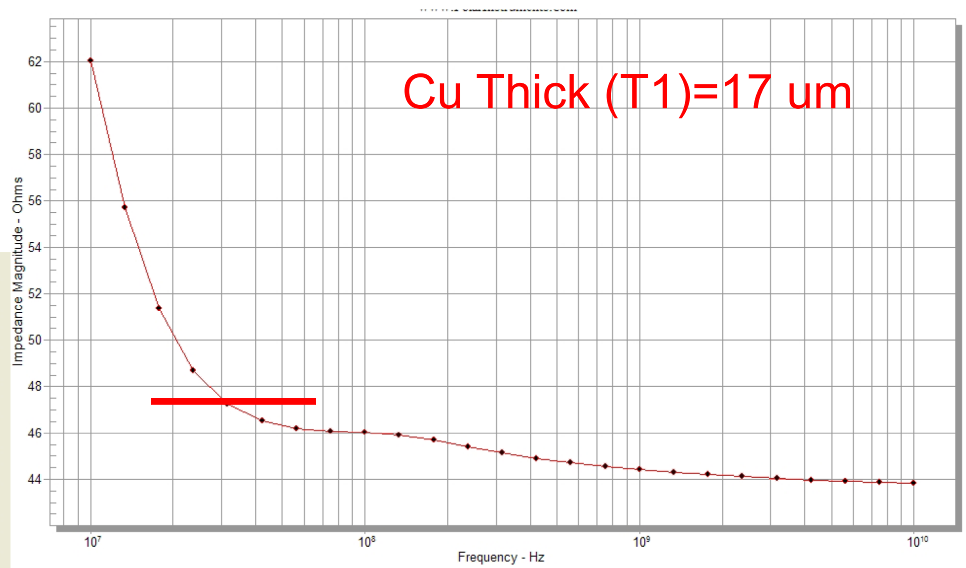


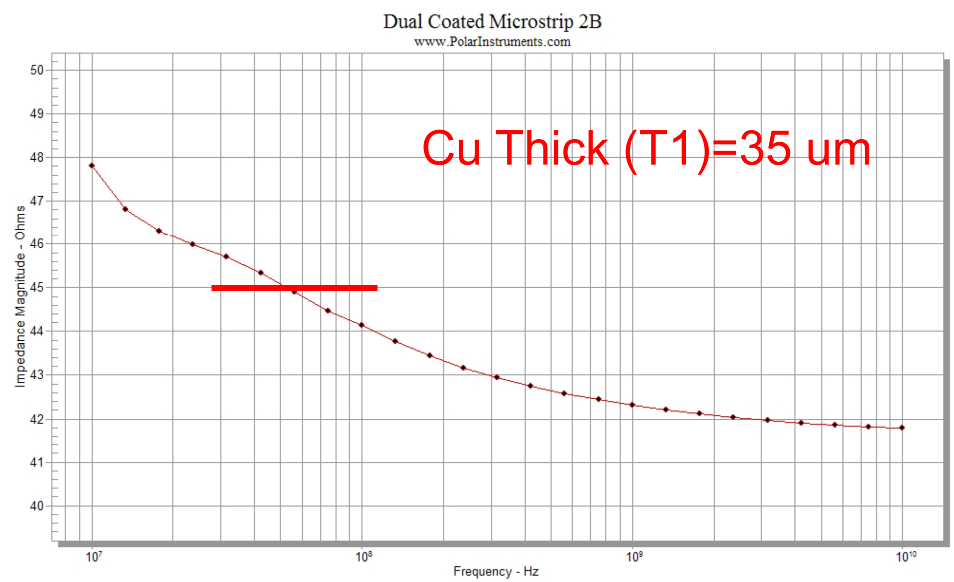




Coated MicroStrip With Ground

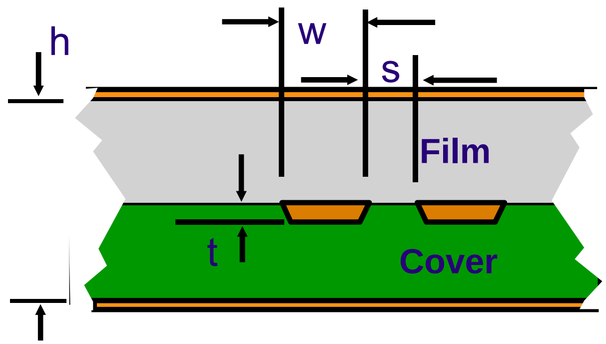




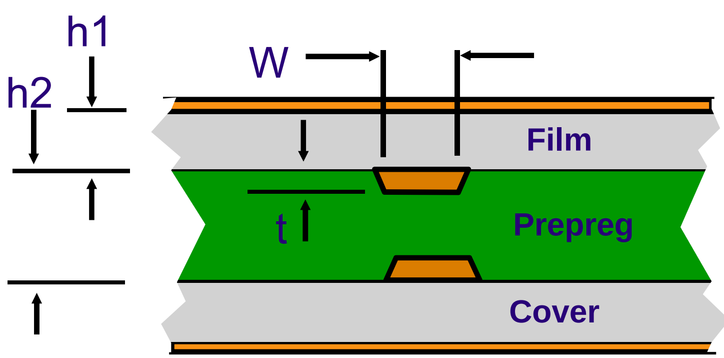


Differential Lines

Design Consideration



Edge Coupled Stripline



Broadside Coupled Stripline

General

- Excellent common mode rejection.
- Matched length required to maintain low common mode currents.
- Lower crosstalk and EMI

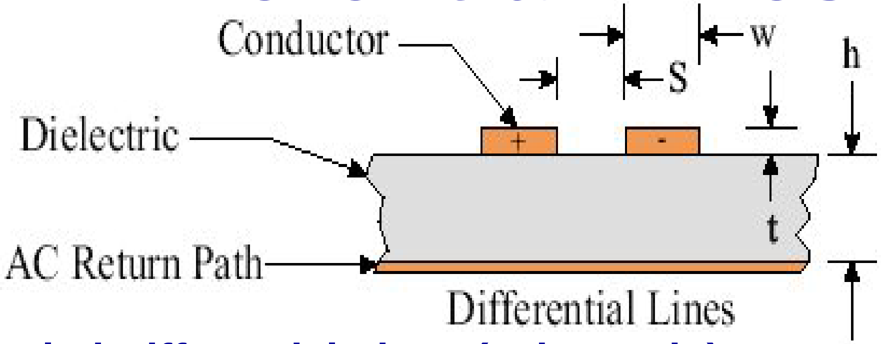
Edge Coupled

- Geometry and spacing defined by artwork
- Can achieve high differential impedance
- Impedance drop as "S" is reduced
- Impedance approaches 2x the single ended impedance as "S" is increased
- Different routing through fine pitch holes

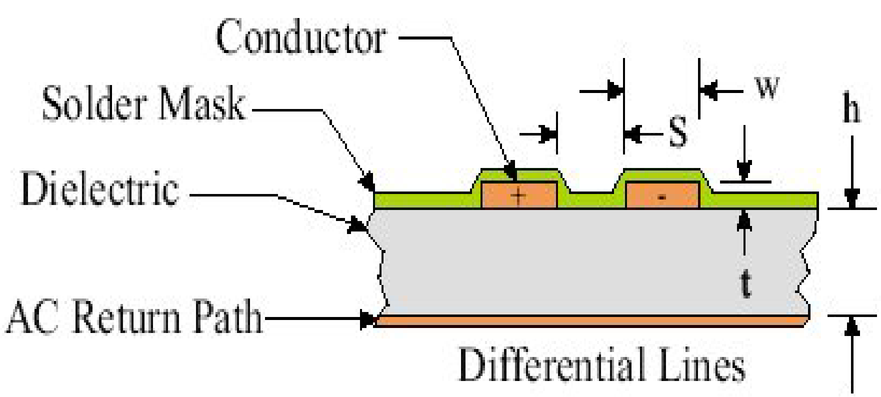
Broadside Coupled

- Geometry affected by layer registration
- Construct differential pairs on "core" layers
- Can achieve very low differential impedance
- Easy to route while maintaining propagation skew

Differential Lines (Microstrip)



Edge-coupled Differential Lines (Microstrip)



Edge-coupled Differential Lines (Coated Microstrip)

Impedance (ohms)

	t	Zo	Zodd	Zeven
Microstrip	0.0017"	72.9	61.0	83.8
Coated Microstrip	0.0017"	67.0	54.6	78.5
CM-Over Plated	0.0027"	63.1	49.5	75.5

Design Consideration

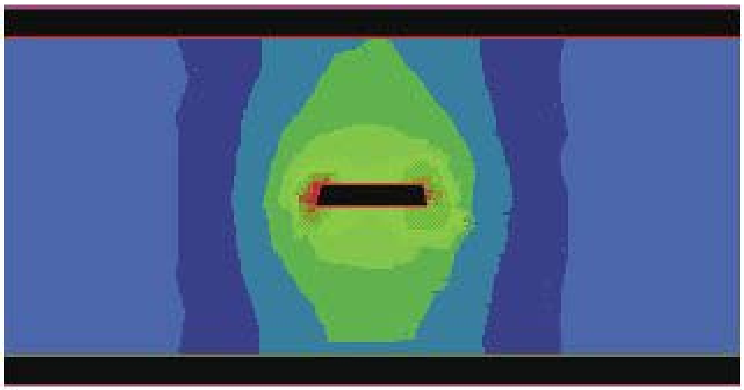
- **Conductor thickness is a function of copper plating (Cu Foil + Cu Plating)**
- **Copper thickness impacts coupling**
- **Mixed dielectric: Laminate + Air**
- **Higher propagation velocity than Stripline using equivalent laminates**
- **Solder Mask lowers the effective dielectric constant between lines**
- **Solder Mask increased mutual capacitance between lines**
- **Conductor edge geometry (plating & etch factor) affect coupling**
- **Non masked lines can provide better control (optional surface finish gold, ect.)**

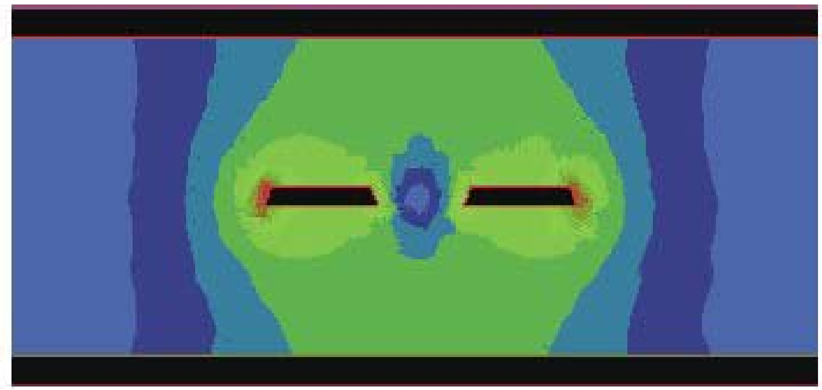
Example

Where:

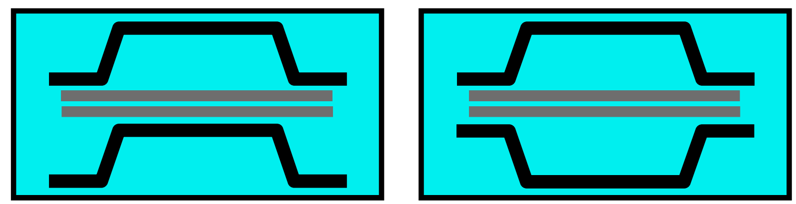
Dielectric (εr)	= 4.3, 3.6 mask
Line width (w)	= 0.005"
Plane distance (h)	= 0.0065"
SM thickness (h1)	= 0.0005"
Cu thickness (t)	= 0.0007" +
Cu plating	0.001" to 0.002"
Spacing (s)	= 0.008"

Modal Impedances





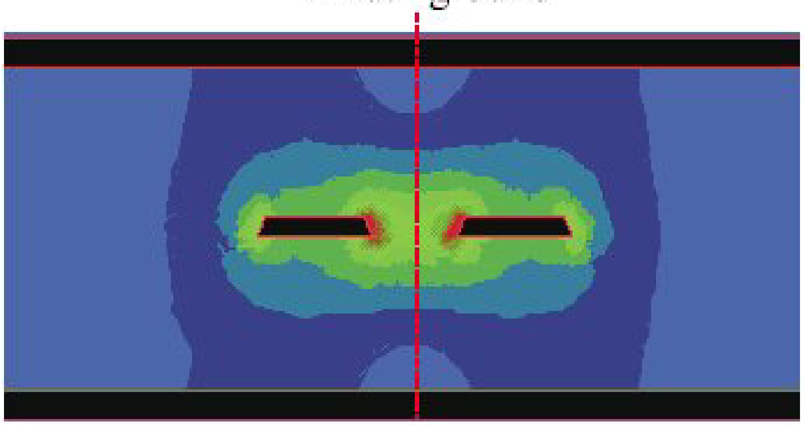
Single ended (Zo)



Even Mode

Odd Mode

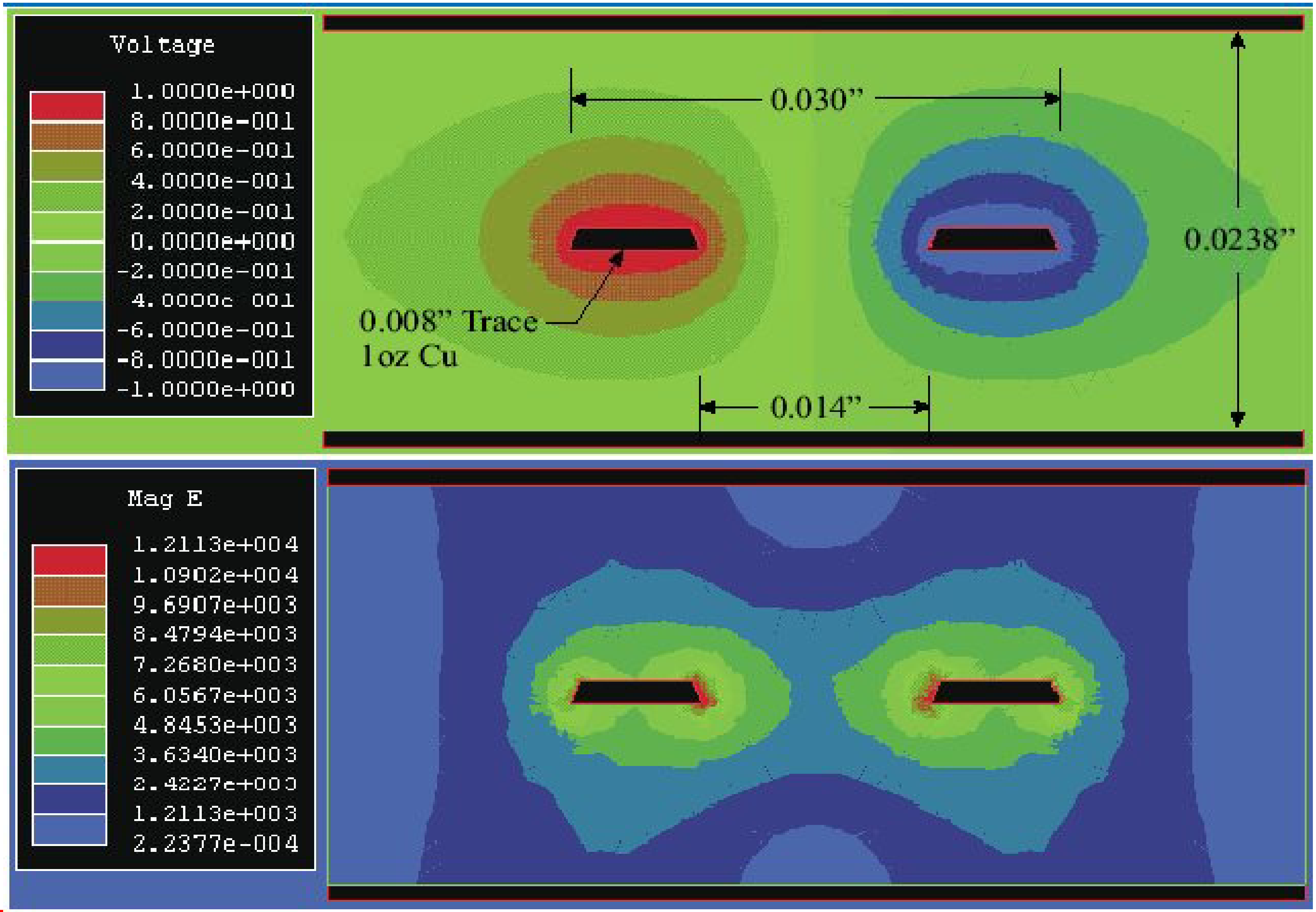
Virtual ground



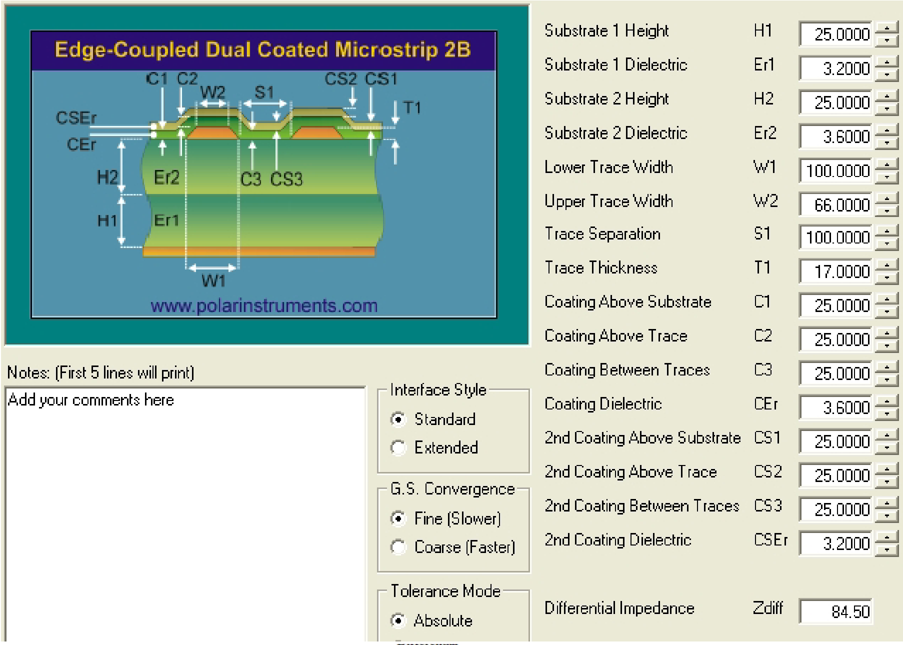
Odd mode

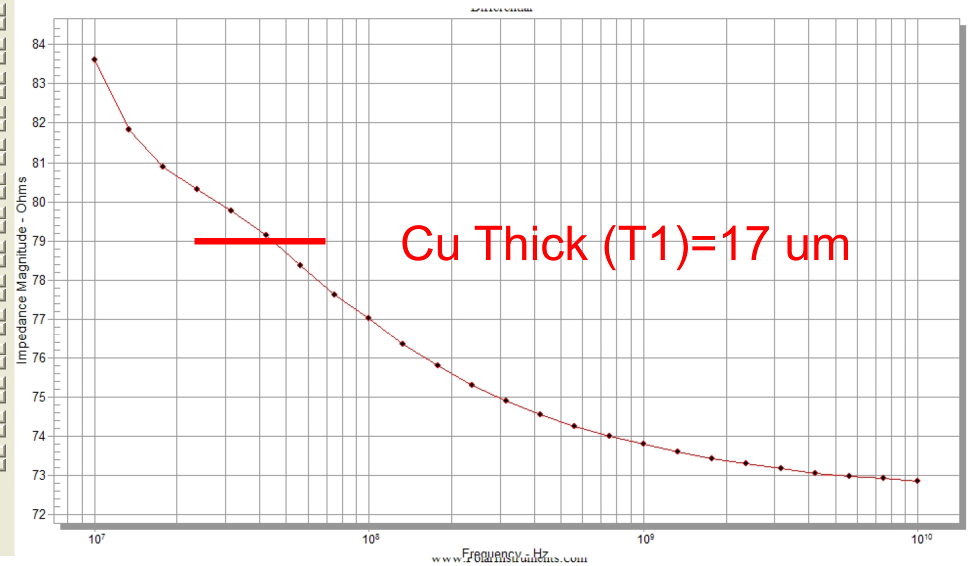
- Differential impedance is twice the odd mode impedance
- Odd Mode os not the same as Differential Mode
- Only differential signals work on the difference between values
- Differential lines set up a virtual ground at the axes of symmetry between the two signal lines

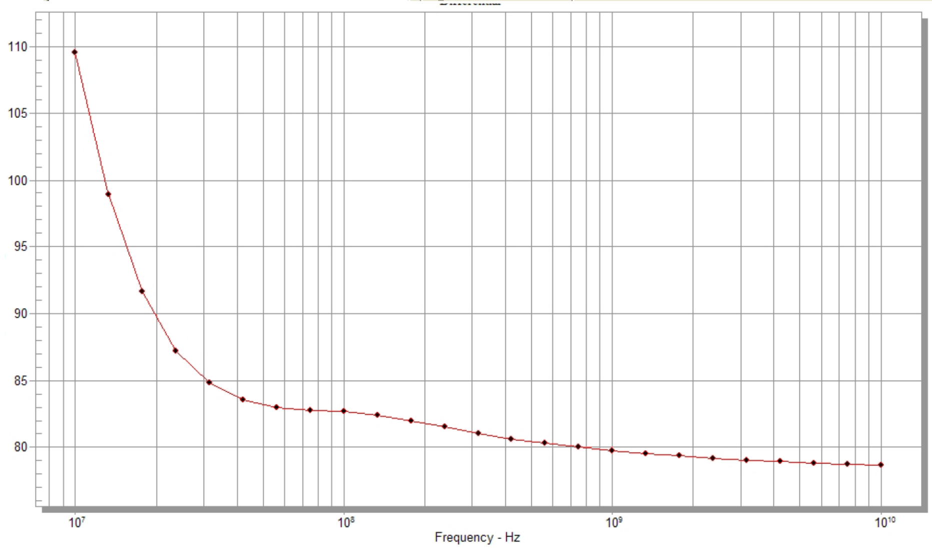
Zo= 52.6 Zodd 48.8 Zeven= 56.2

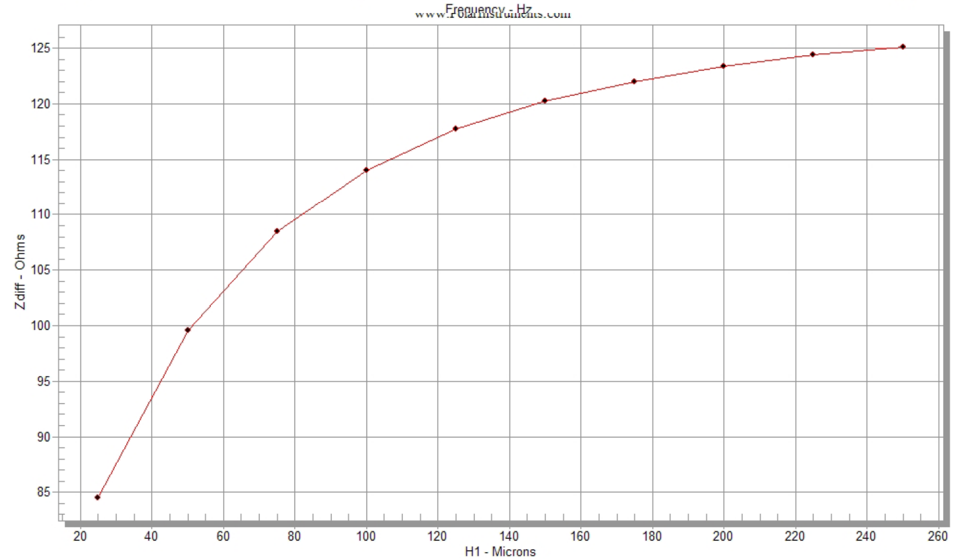


Differential Coated MicroStrip

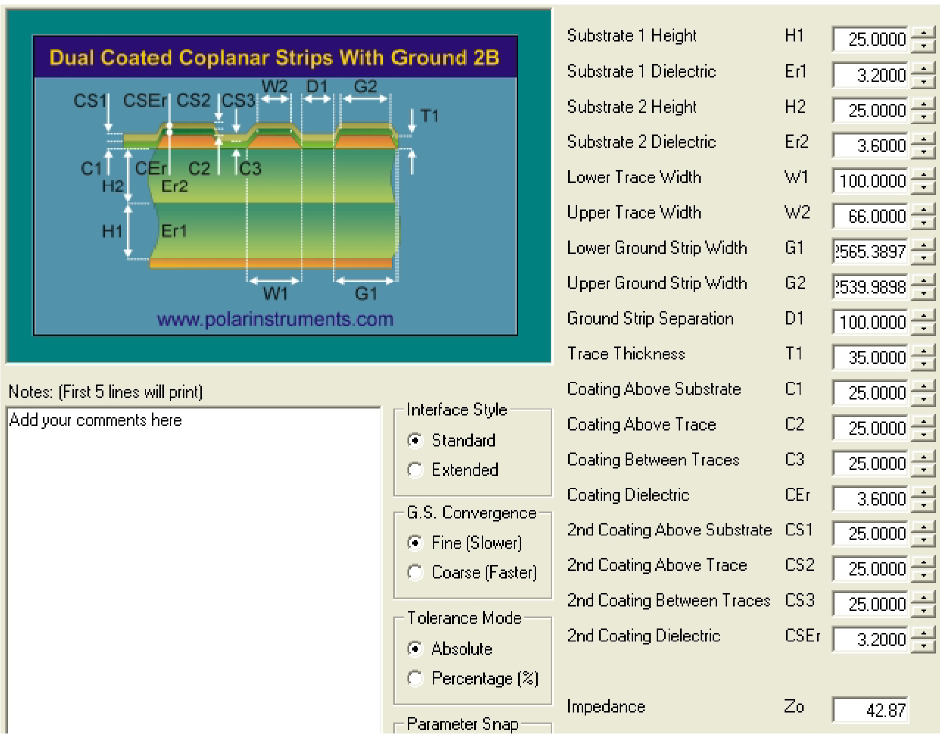


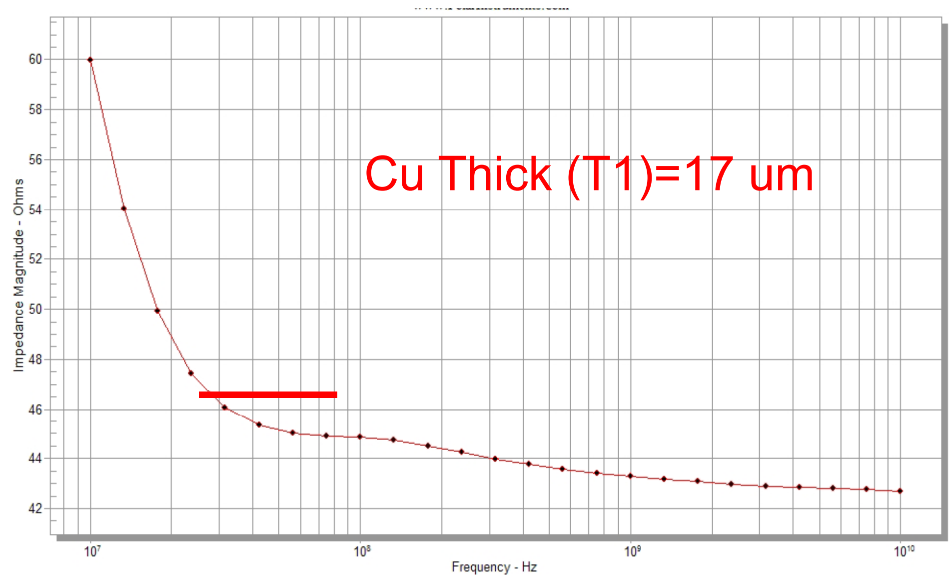


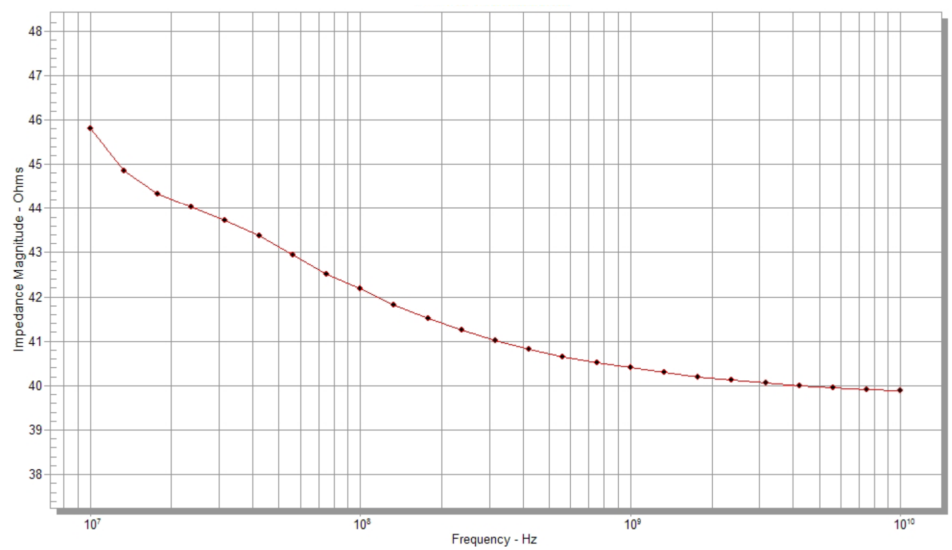




Differential Coated Stripline With Ground

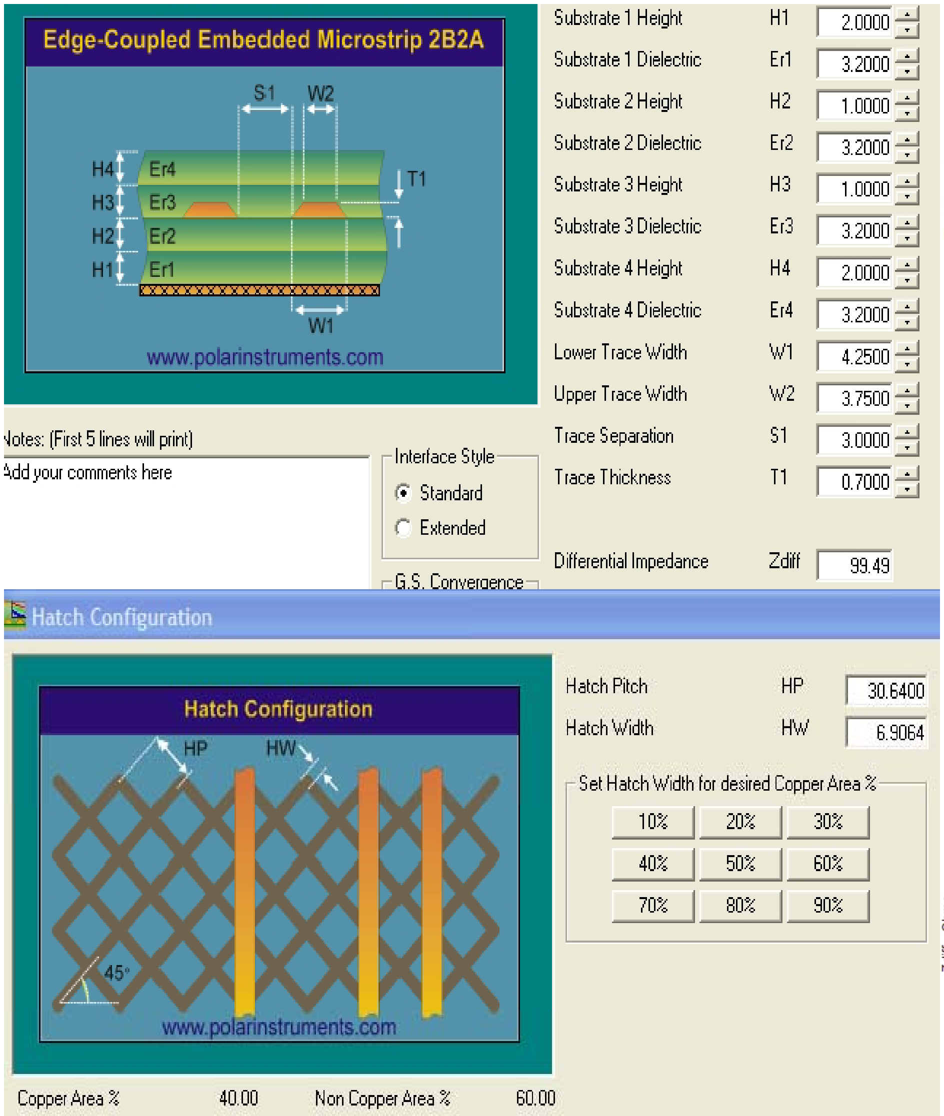


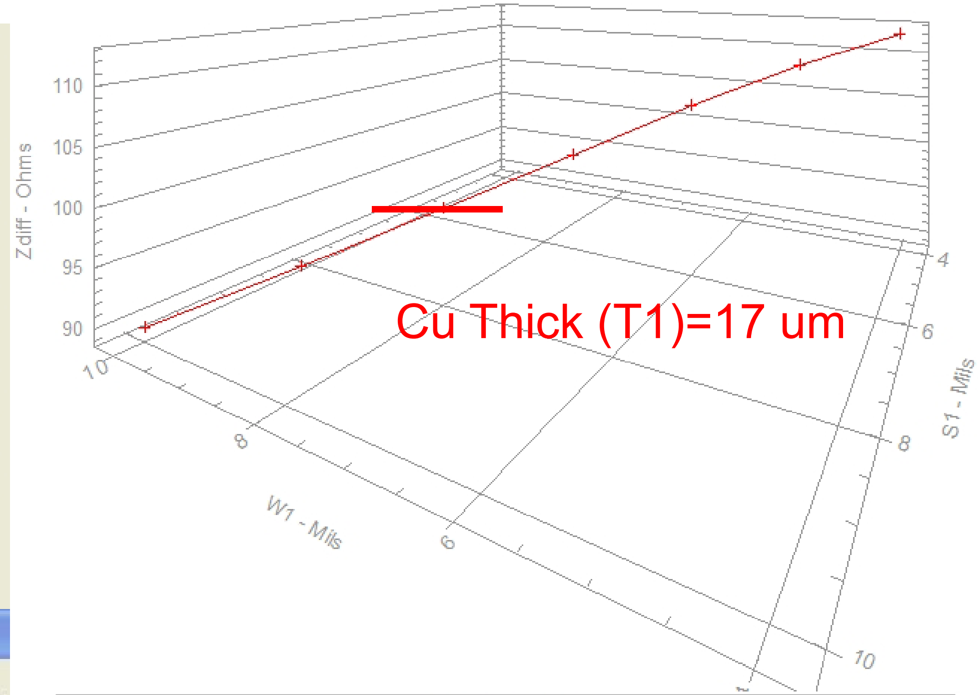


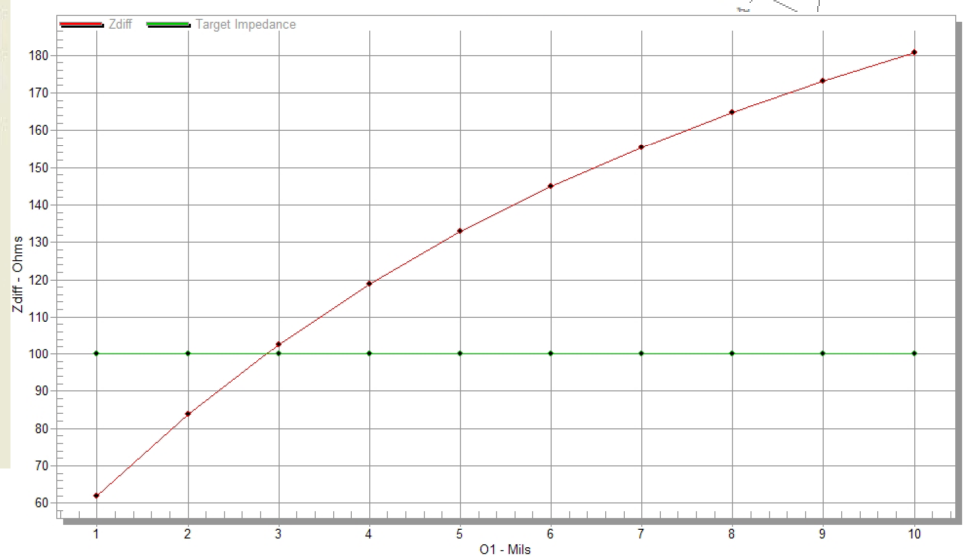


Mesh Ground

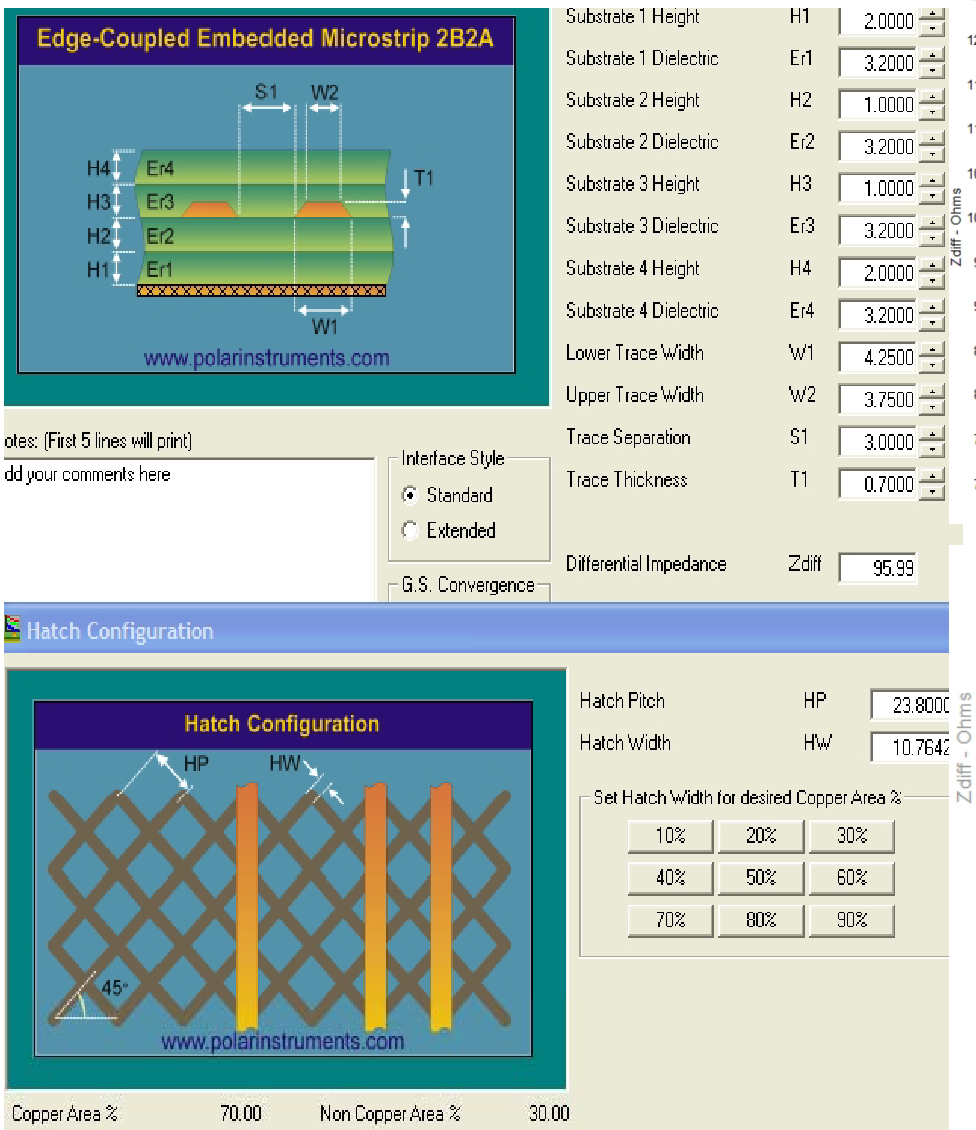
Differential Embedded MicroStrip w/Mesh (40%) GND

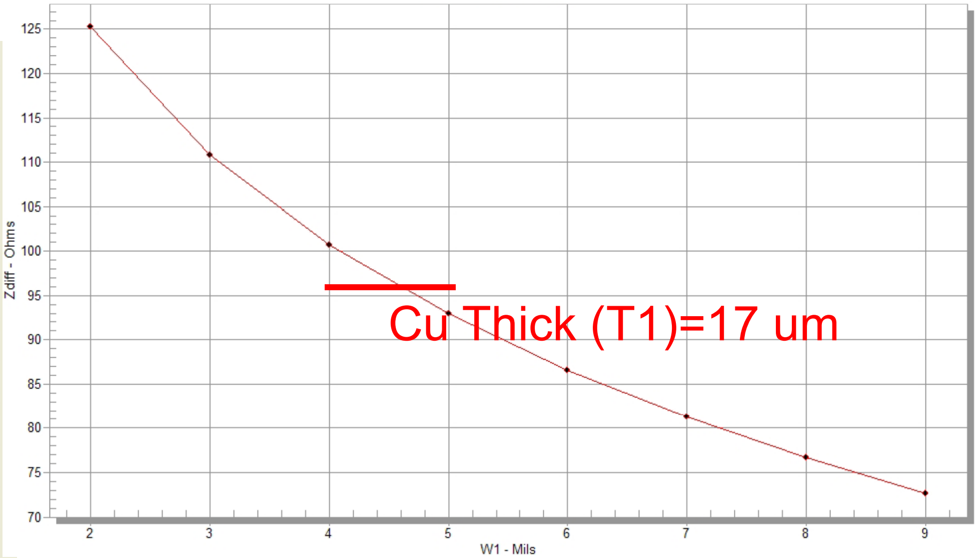


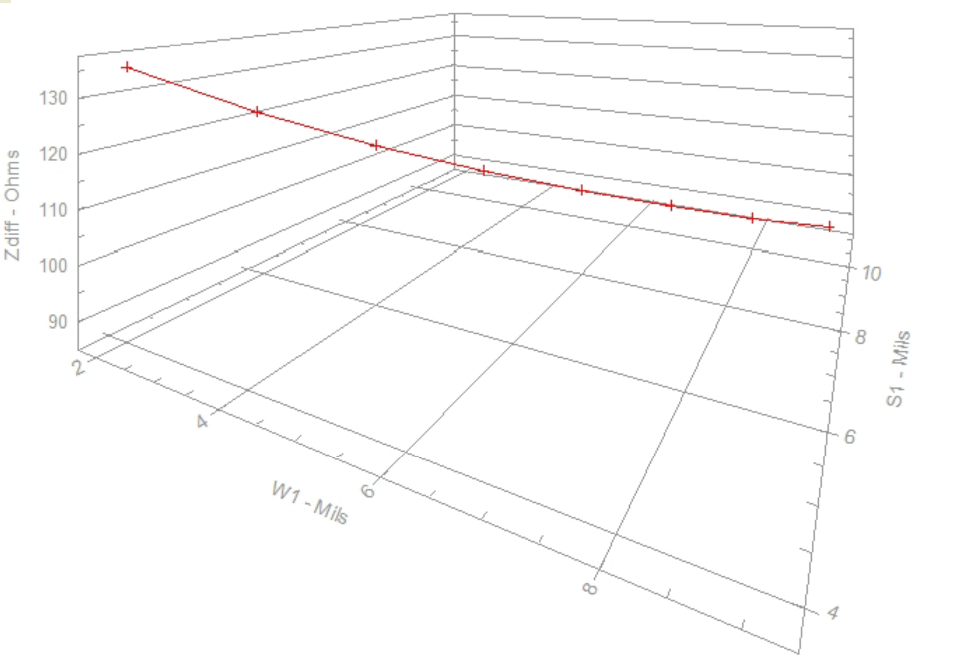




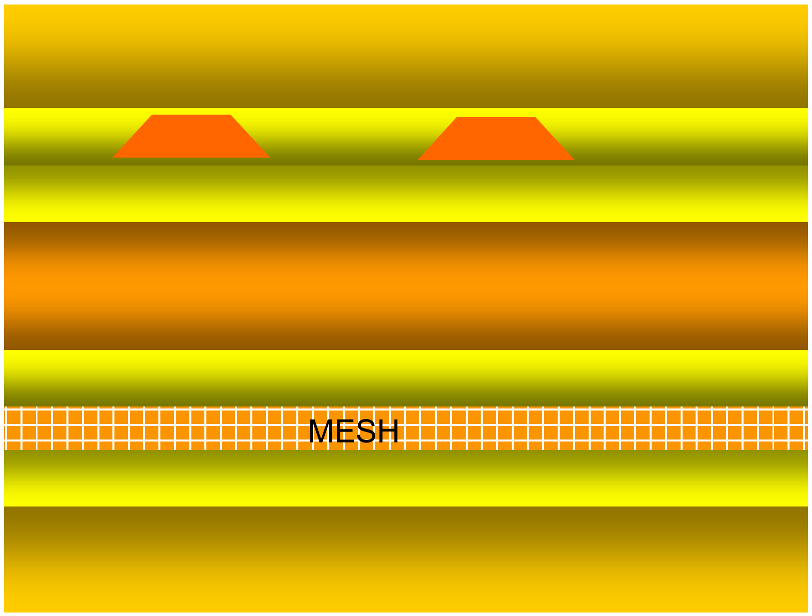
Differential Embedded MicroStrip w/Mesh (70%) GND

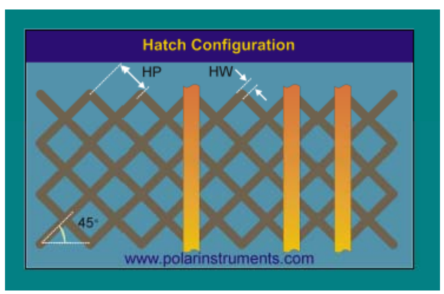






Actual Flex Material Stackup

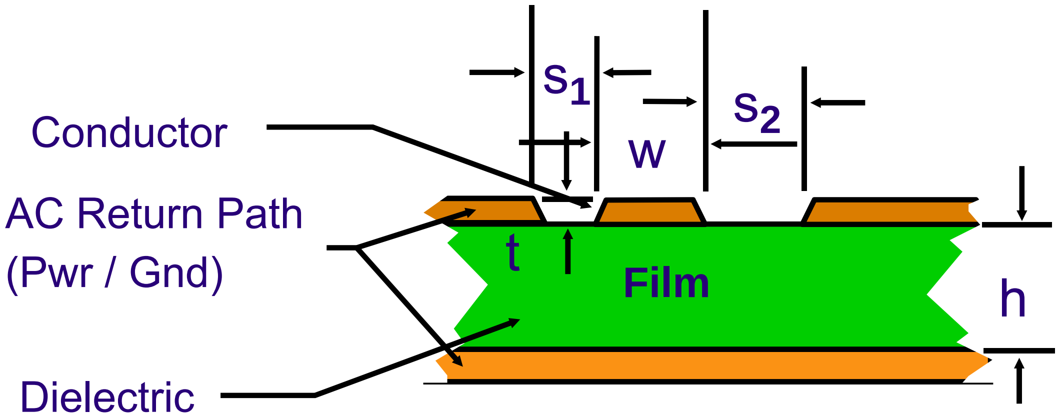




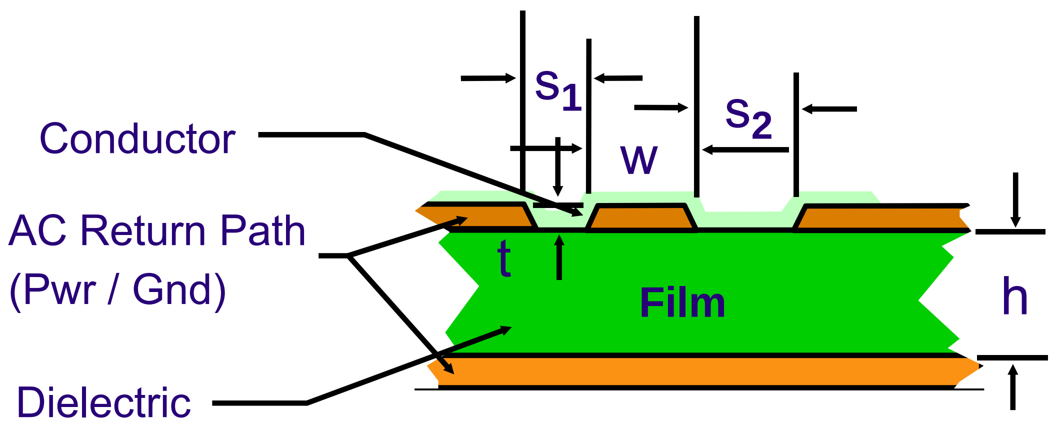
Need 5 materials and different angles on the MESH

Coplanar Structures

Coplanar Transmission Lines



Coplanar Microstrip

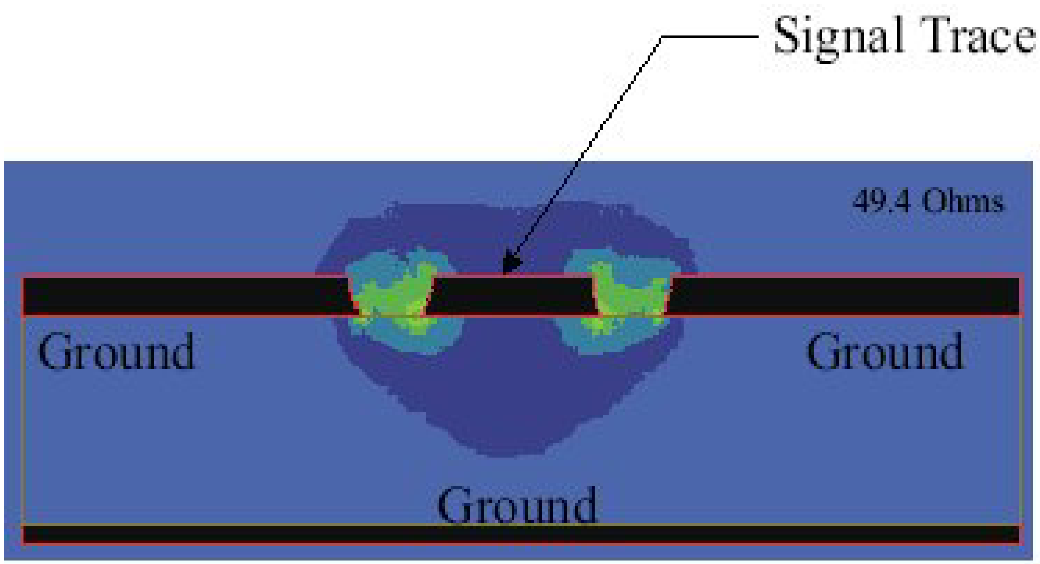


Coated Coplanar Microstrip

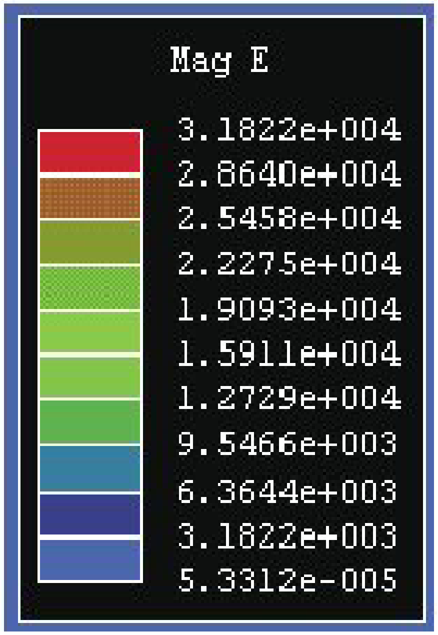
Design Consideration

- **Conductor thickness is a function of copper plating (Cu foil + Cu plating)**
- **S1 & S2 critical for impedance control**
- **Mixed dielectric: Laminate + Air**
- **Higher propagation velocity than stripline using equivalent laminates**
- **Solder Mask lowers the effective dielectric constant**

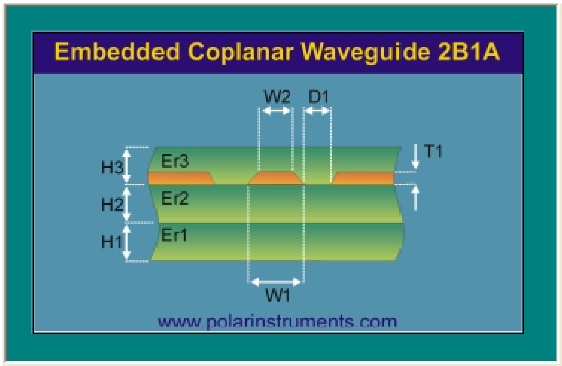
Field Distribution: Coplanar



Coplanar Stripline: Surface Trace
0.009" trace 0.002" thick



Embedded Coplanar Stripline With-Out Ground



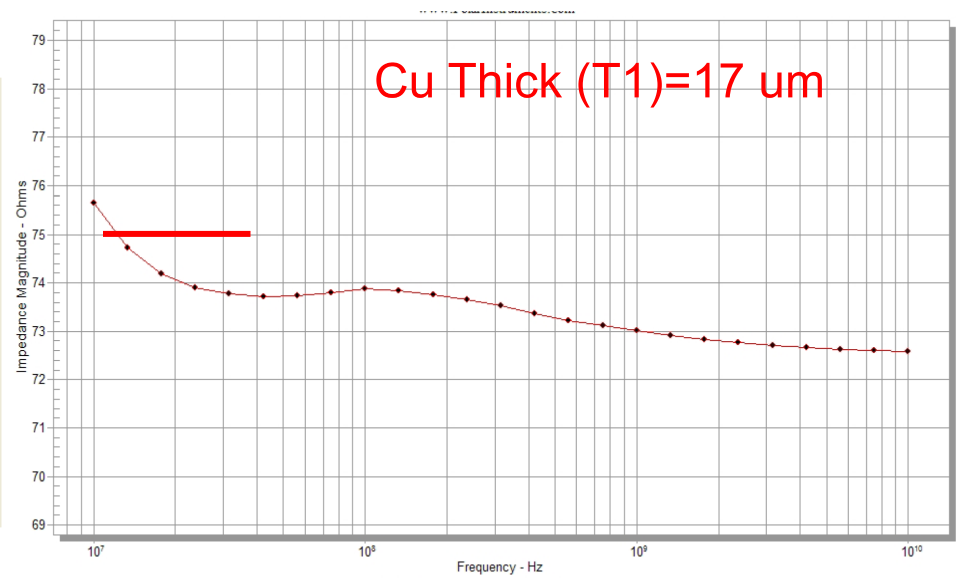
Substrate 1 Height	H1	25.0012
Substrate 1 Dielectric	Er1	3.2000
Substrate 2 Height	H2	25.0012
Substrate 2 Dielectric	Er2	3.6000
Substrate 3 Height	H3	25.0012
Substrate 3 Dielectric	Er3	3.2000
Lower Trace Width	W1	300.0000
Upper Trace Width	W2	266.0000
Ground Strip Separation	D1	100.0000
Trace Thickness	T1	17.0002
Impedance	Zo	75.06

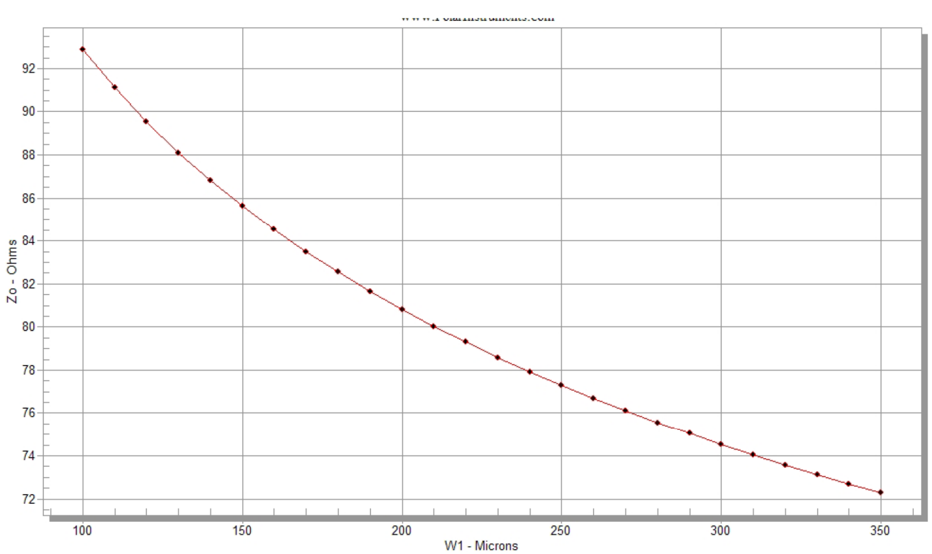
Notes: (First 5 lines will print)

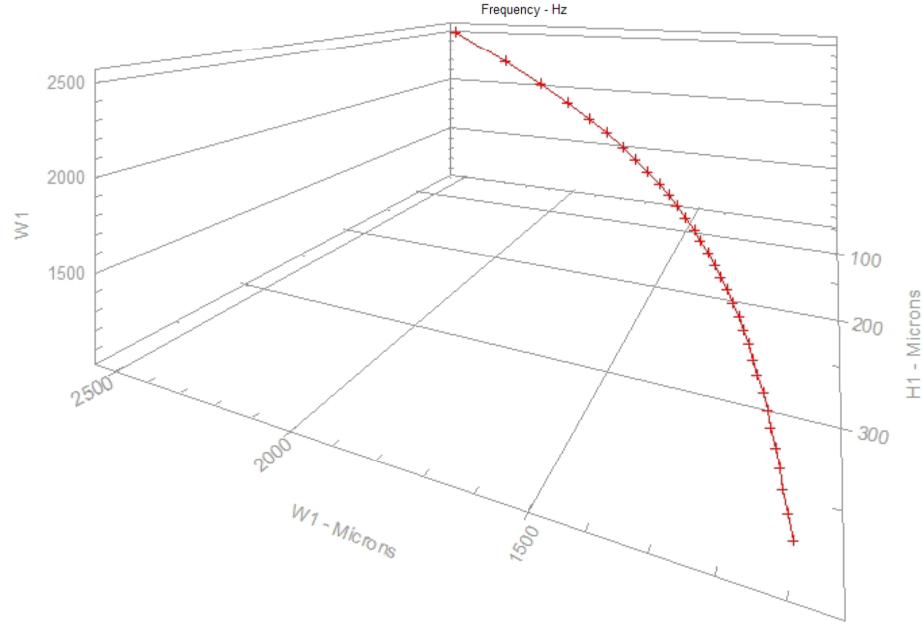
Add your comments here

Interface Style

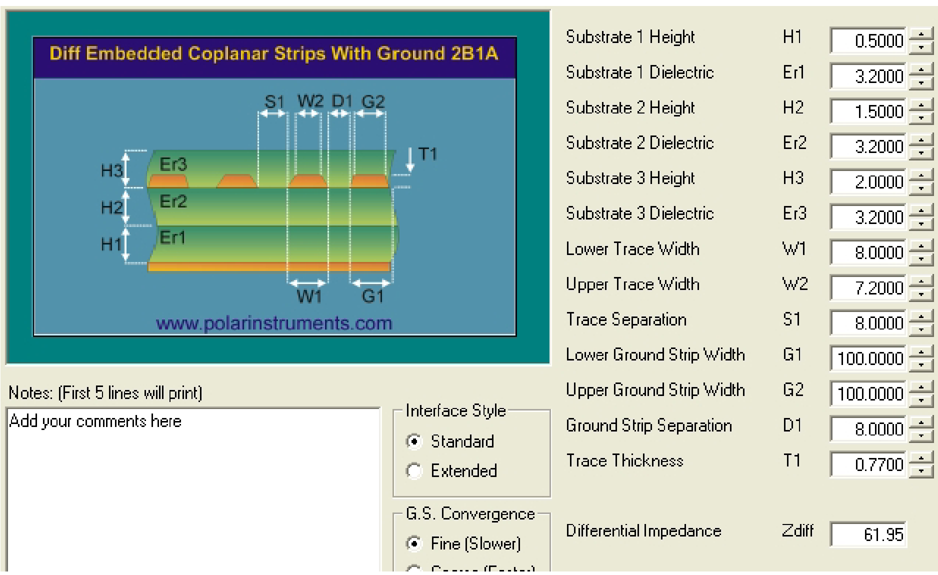
Standard

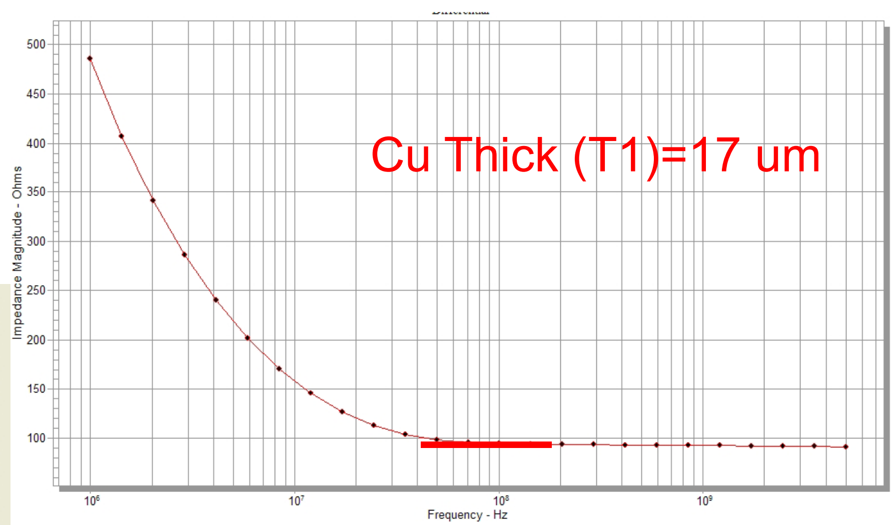


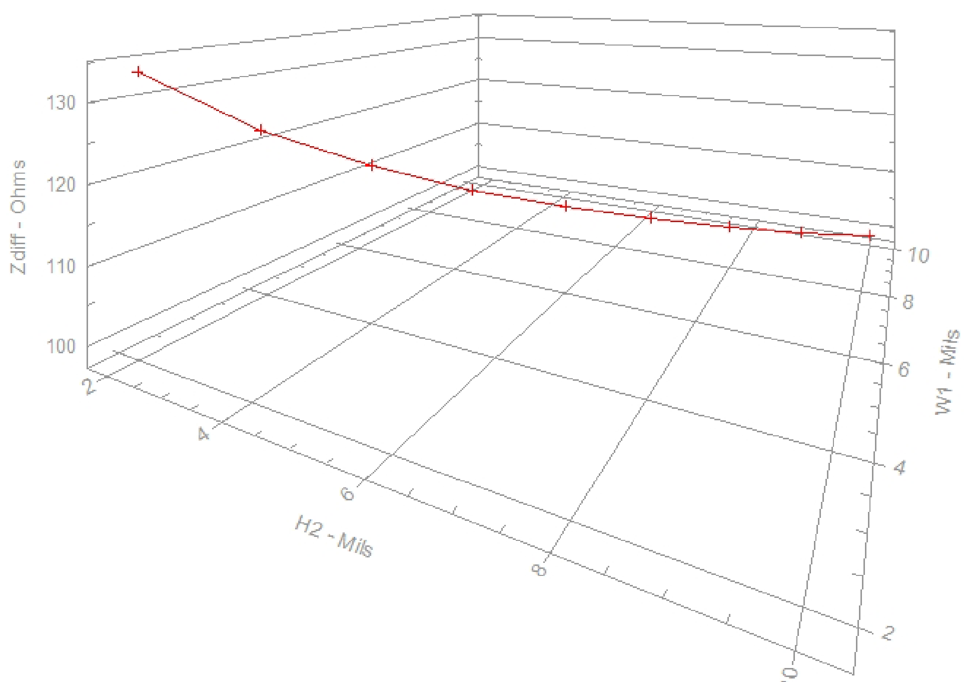




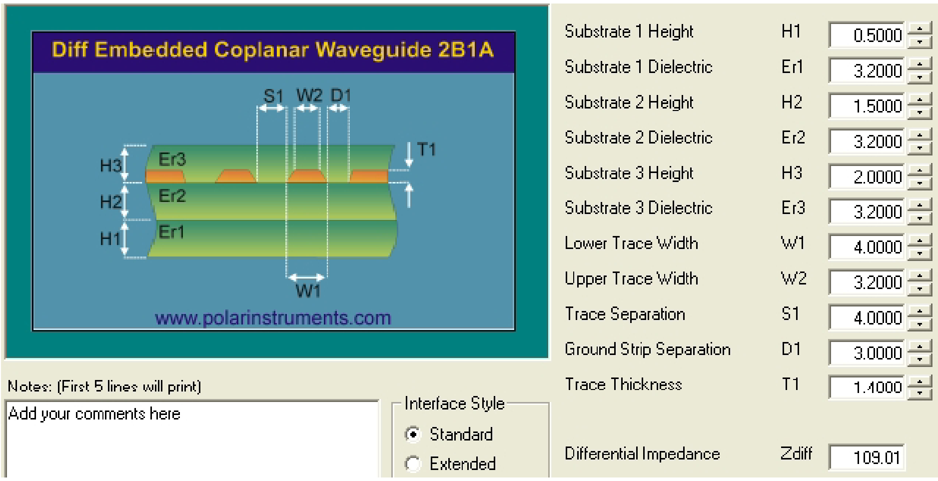
Differential Embedded Coplanar Stripline With Ground

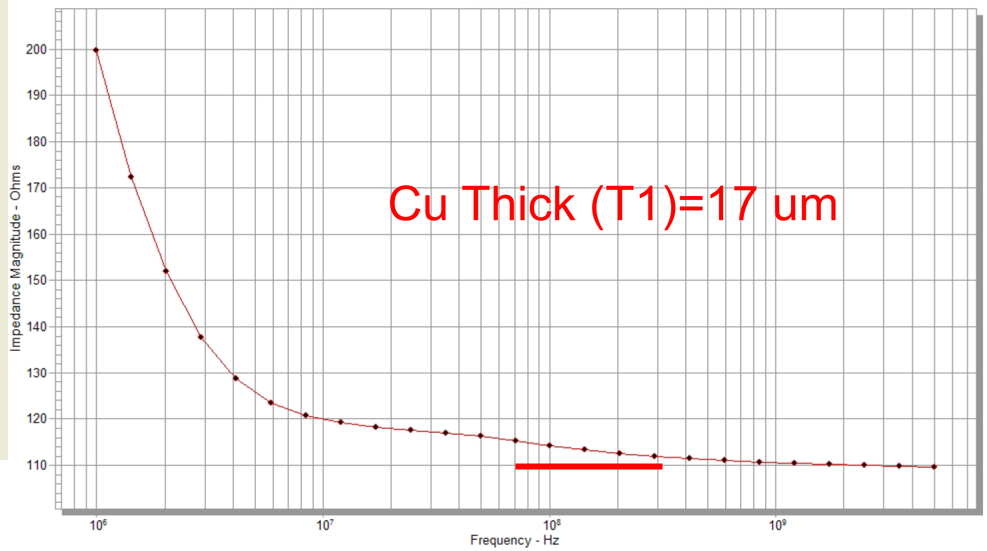


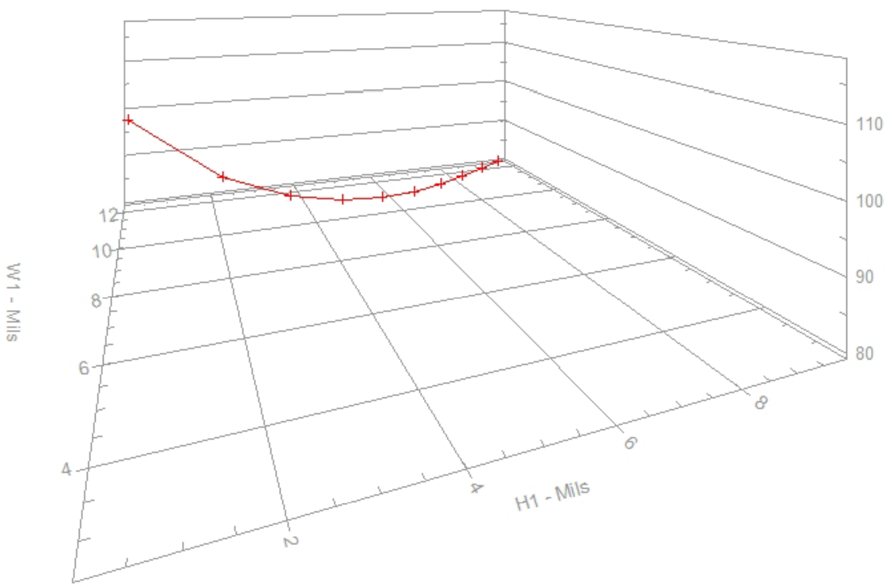


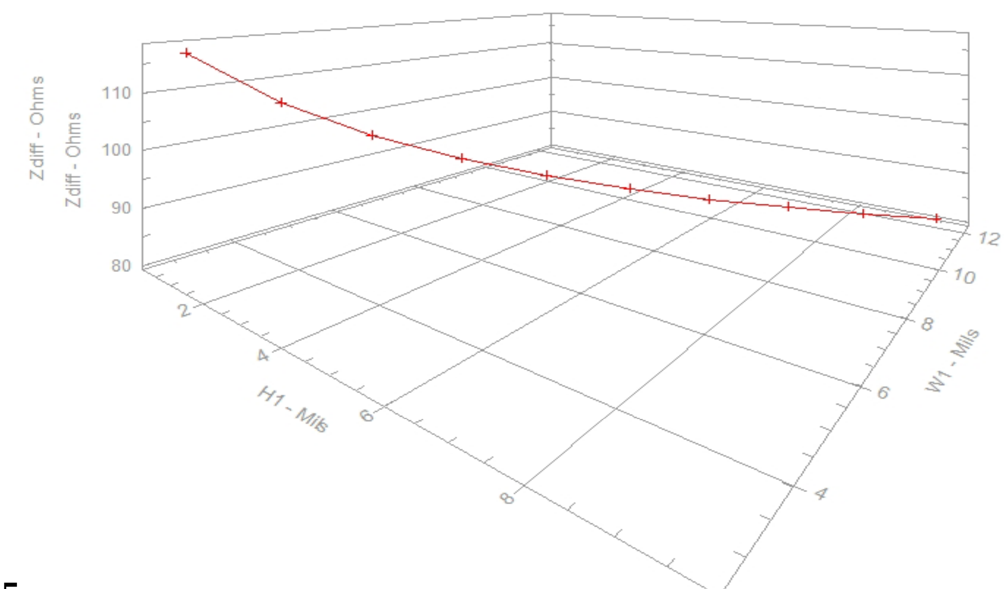


Diff. Embedded Coplanar Waveguide With-Out Ground



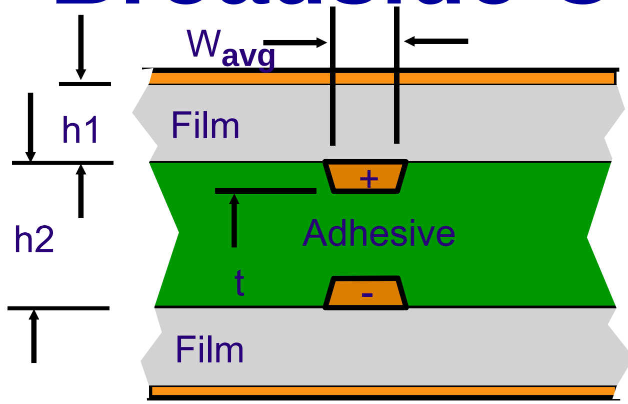




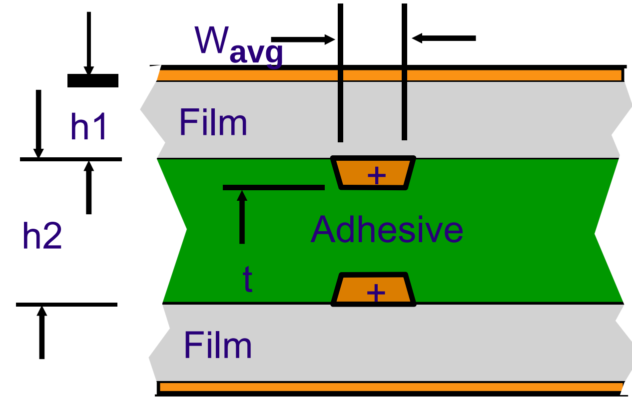


Diff. Broadside Coupled

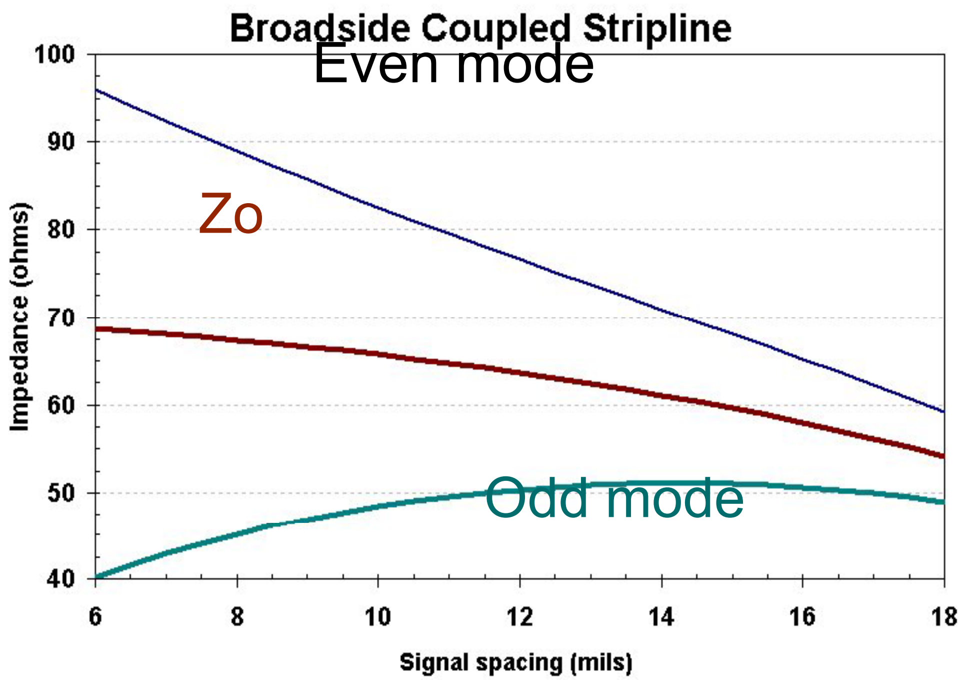
Broadside Coupled Lines



Broadside Coupled Stripline (odd mode)



Broadside Coupled Stripline (even mode)

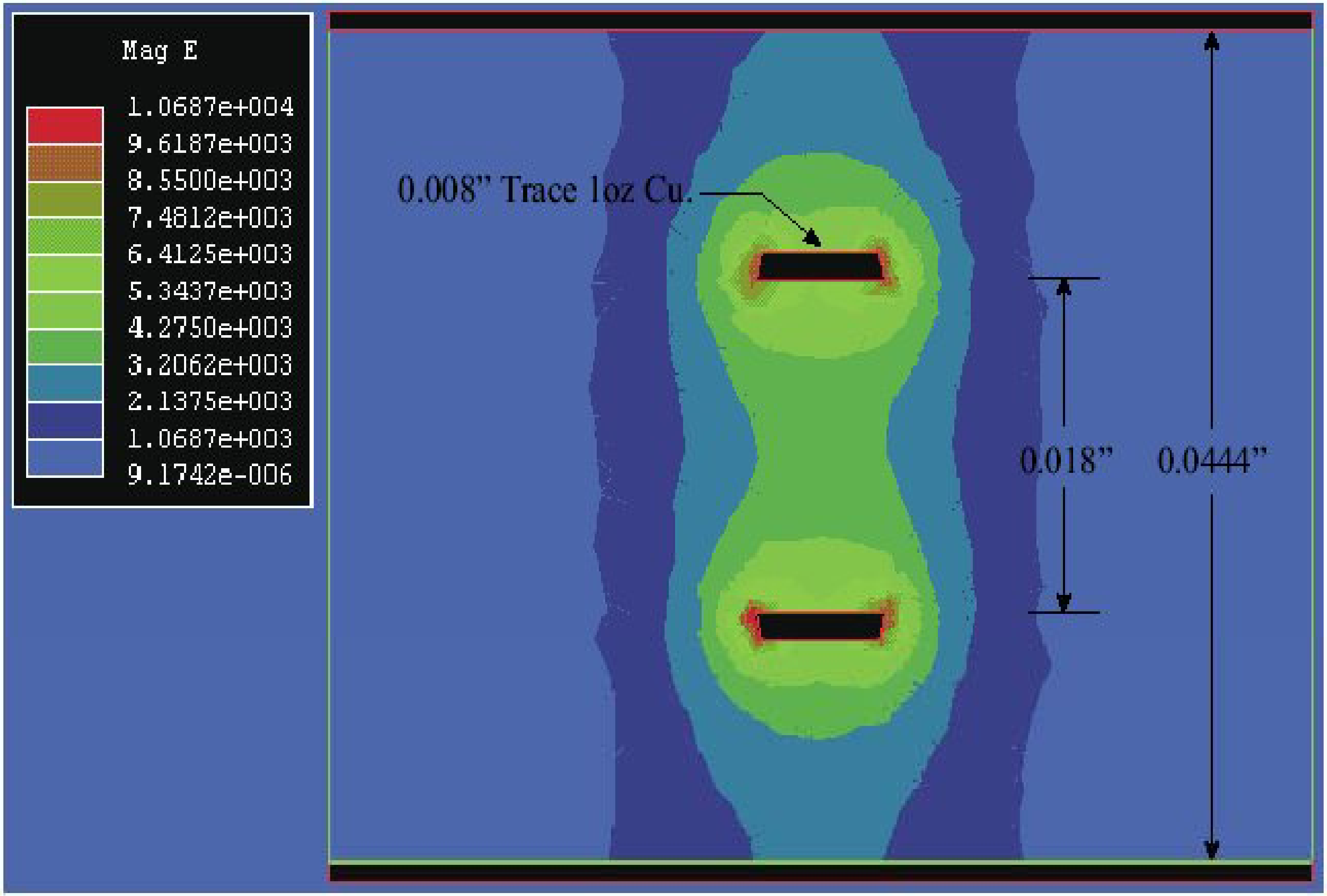


Broadside Example

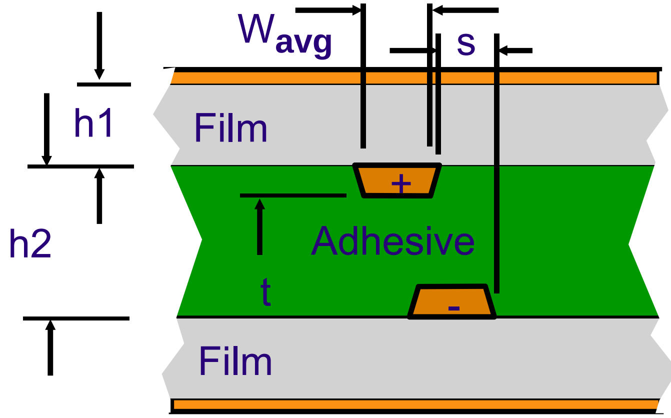
Where:

Dielectric (εr)	**= 4.3**
Line width (w)	**= 0.005"**
Plane distance (h)	**= 0.031"**
(2h1 + h2 + 2t)	
Cu thickness (t)	**= 0.0013"**
Diel. Spacing (h2)	**= 0.006" to 0.018"**

Zo= 63.0 Zodd= 50.1 Zeven= 75.6

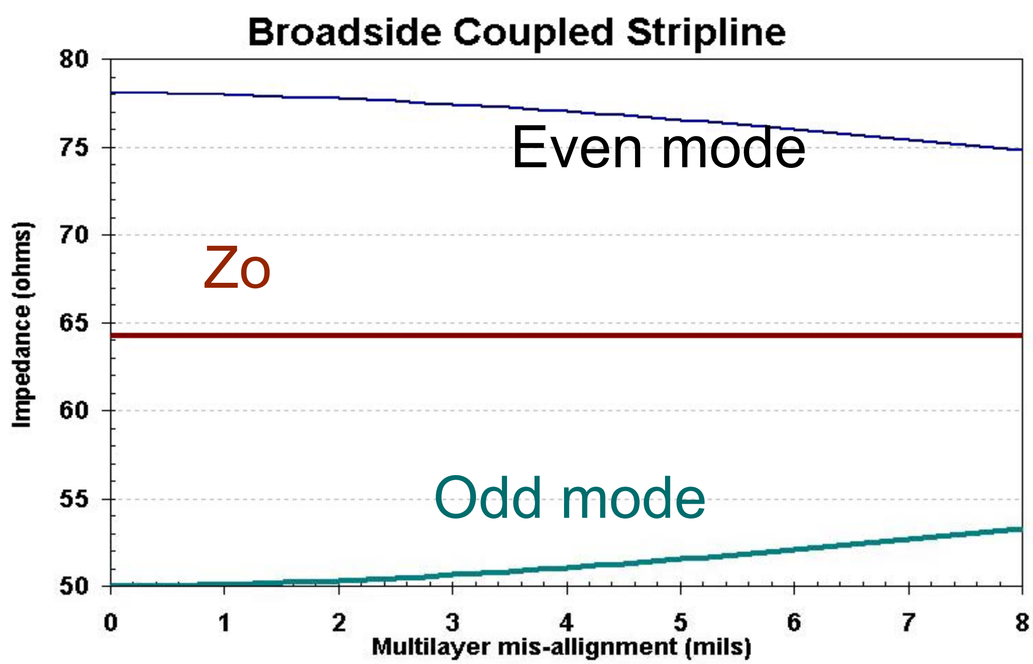


Broadside Coupled Lines



Design Consideration

- Signals opposite the same core provide better impedance control.
- Signal to plane distance control is less critical than signal - signal.



Broadside Example

Where:

Dielectric (εr) = 4.3

Line width (w) = 0.005"

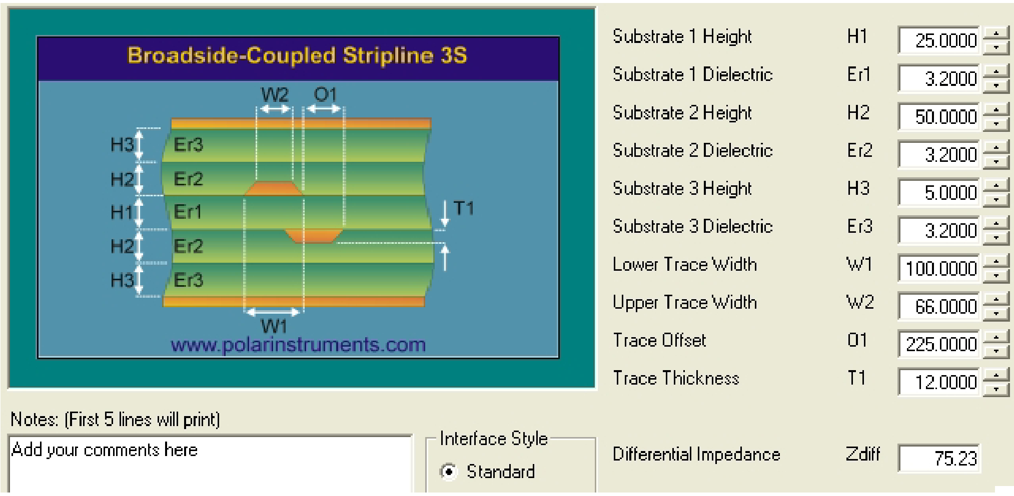
Plane distance (h) = 0.031" (2h1 + h2 + 2t)

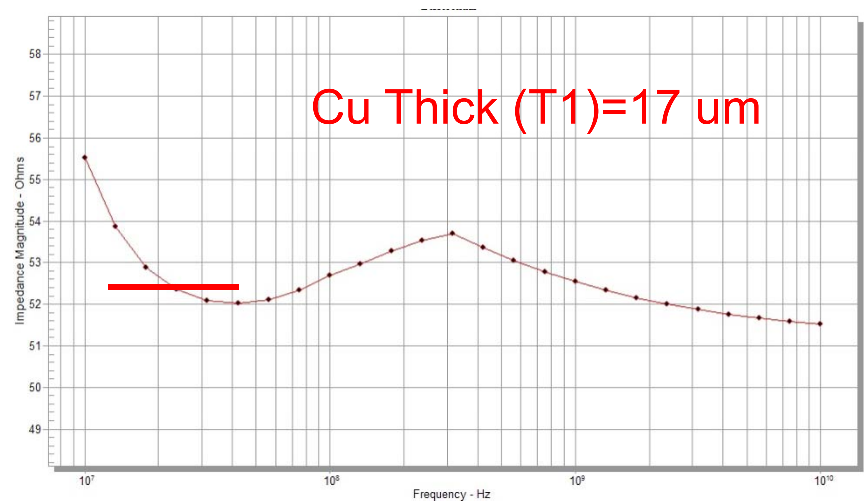
Diel. Spacing (h2) = 0.00115"

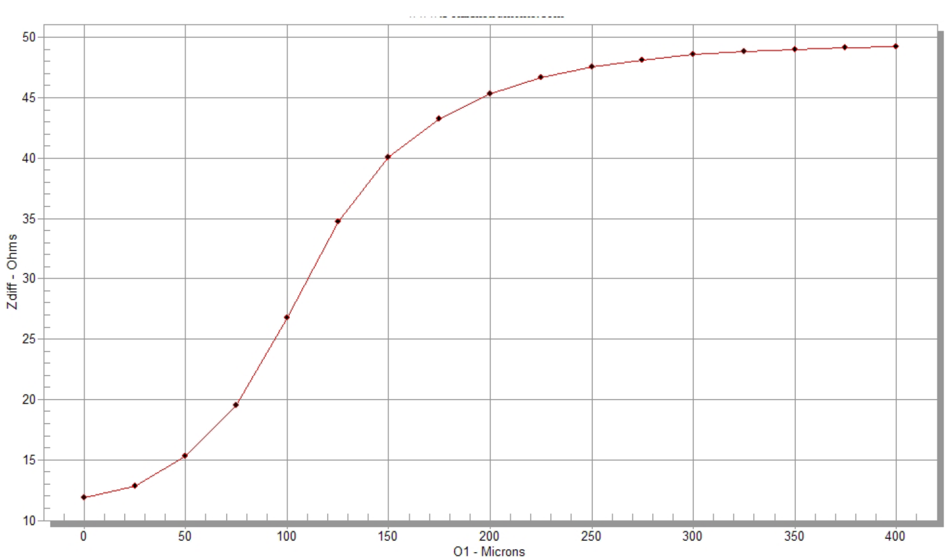
Cu thickness (t) = 0.00137"

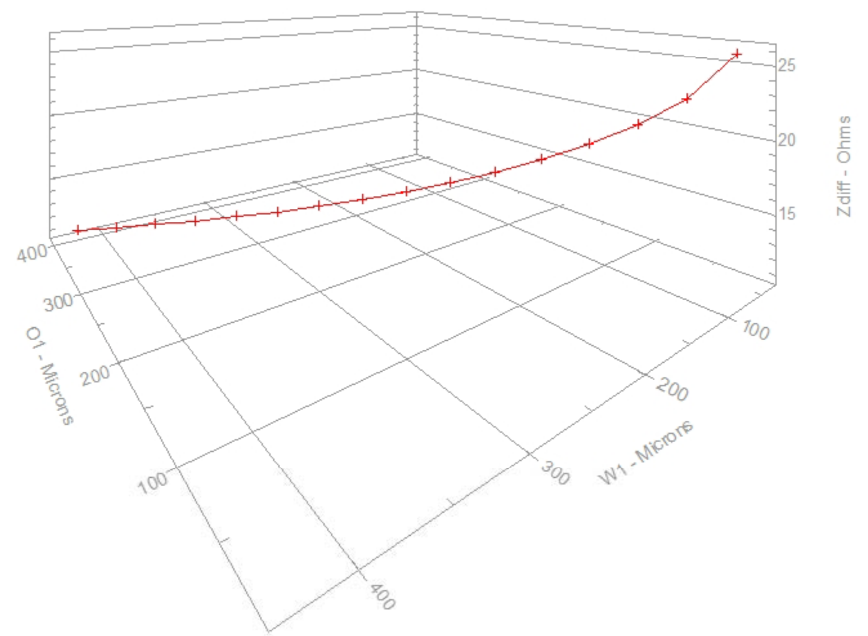
Trace allignment(s) =0 to 0.008"

Broadside Differential Stripline With Ground

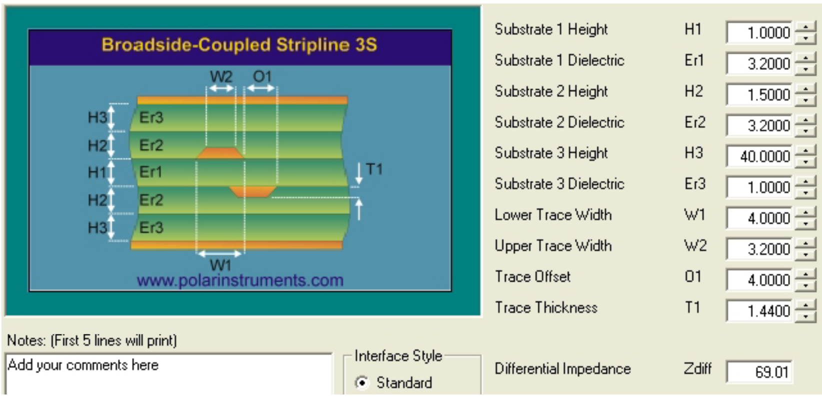


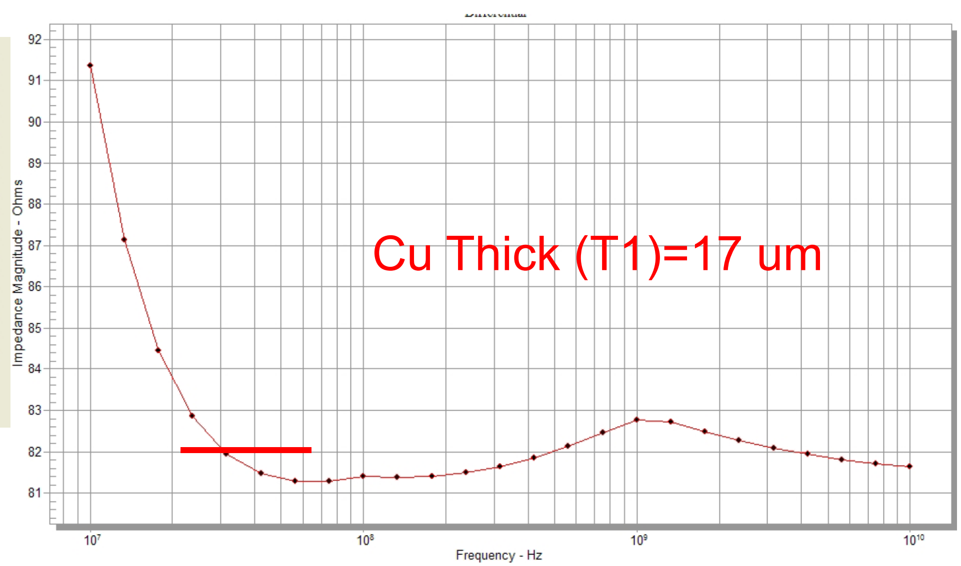


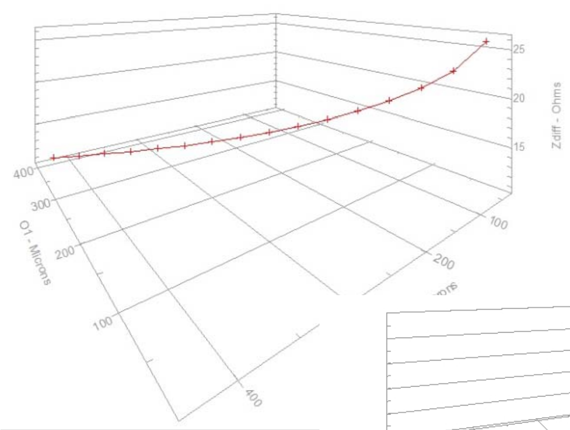


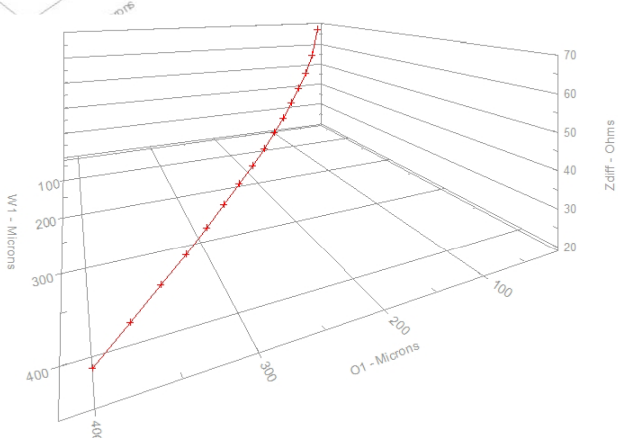


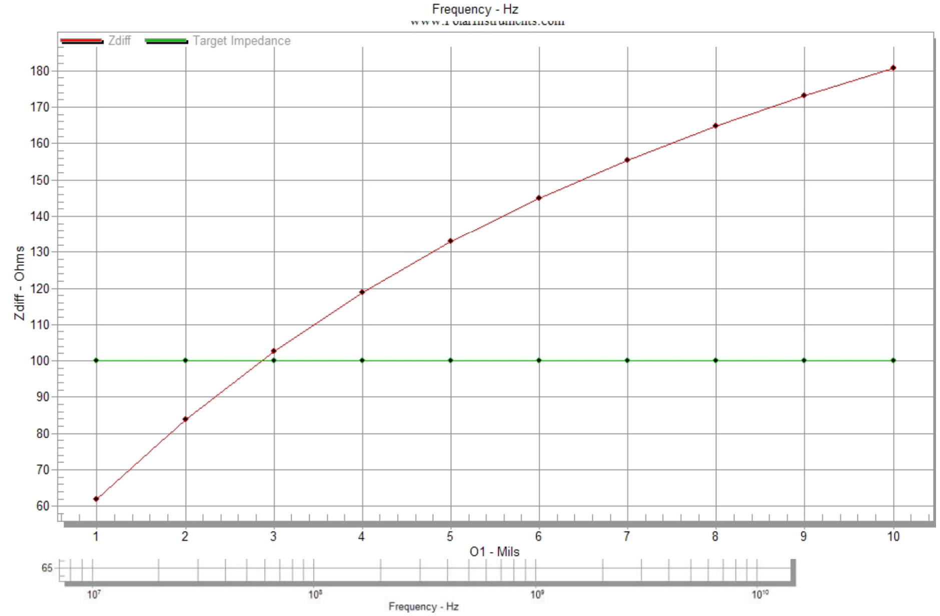
Broadside Differential Stripline With-Out Ground







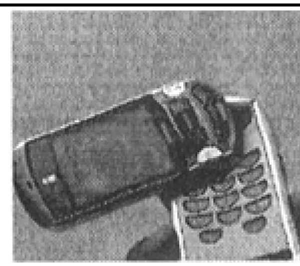
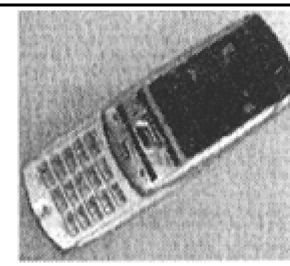
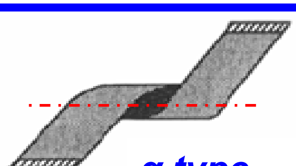
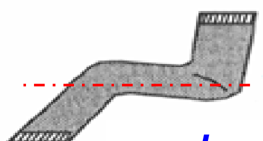
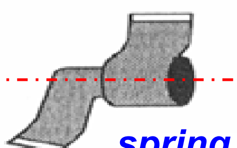
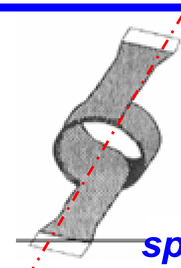
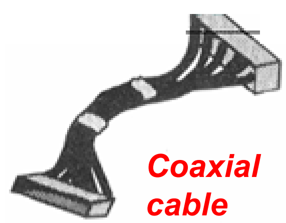
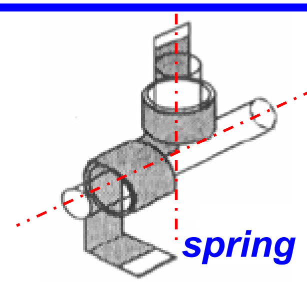
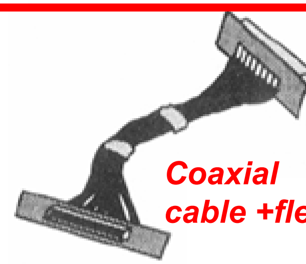
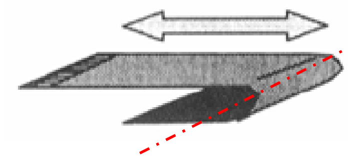




Foxconn Proprie

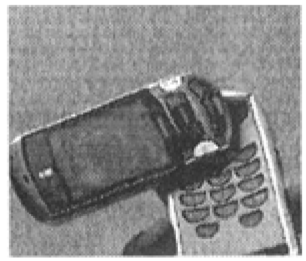
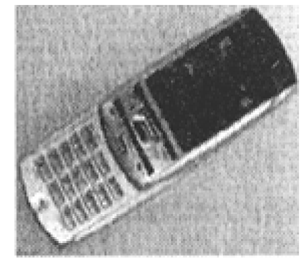
Dynamic Bend for Different Stackup

Hinge Type

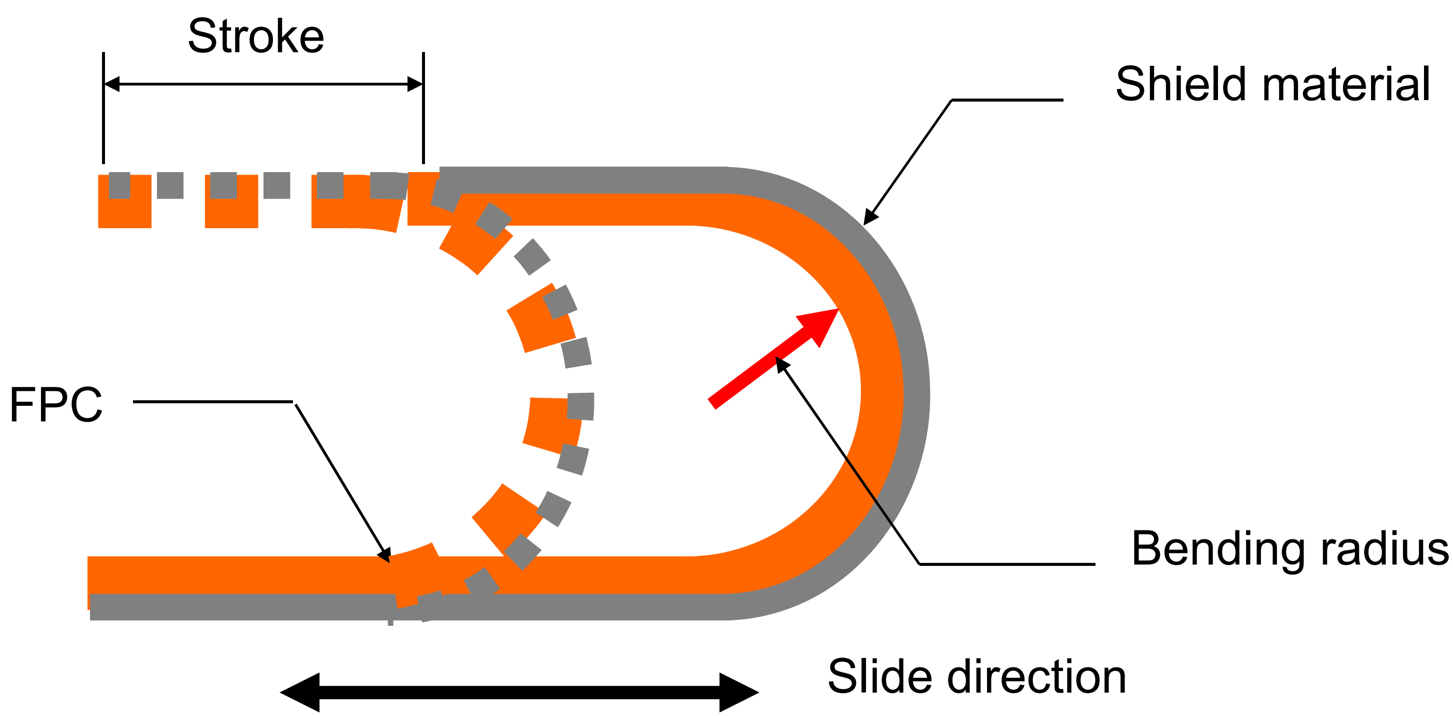
Hinge Type	Shell	Revolve	Universal	Slide
Diagram				
Application	 α type  crank  spring	 spring  Coaxial cable	 spring  Coaxial cable +flex	 slide

Blue box: :flex   Red box: :coaxial cable

Hinge Type Structure

Hinge Type	Shell	Revolve	Universal	Slide
Diagram				
Copper Thickness	RA copper ½ oz	RA copper ½ oz	RA copper ½ oz	HTE copper 1/3 oz
PI	1mil or ½ mil	1mil or ½ mil	1mil or ½ mil	0.7mil,0.8mil or ½ mil
Structure suggestion	1.Single side flex 2.Double sided flex with one side trace 3.Multi-layer with air gap	1.Single side flex 2.Double sided flex with one side trace	1.Single side flex 2.Double sided flex with one side trace	1.Single side flex is preferred 2.Double sided flex with one side trace 3.Due to very tight space, air gap is not preferred.

Test Sample Description



Key message:

.**Bending radius:** 1.15mm

.**Stroke:**25mm

.**Frequency:** 0.5 Cycle/s

.Shield material: Outward

.**Pass criteria**: **Resistance variation less than 10%**

Static Bending Radius

Stack	Thickness	Bending angle	Bending radius (Mandrel)	Remark
1-Layer	0.11	180	0.6	
2-Layer	0.165	180	0.6	Mesh ground copper is much better than solid ground copper
3-Layer	0.2	180	0.6	Layer transition is much better for folding
4-Layer	0.29	180	1.25	Layer transition is much better for folding
5-Layer	-	Not recommended	-	Layer transition is much better for folding

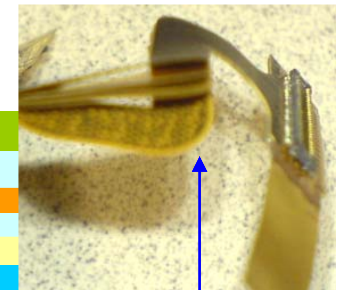
Key message

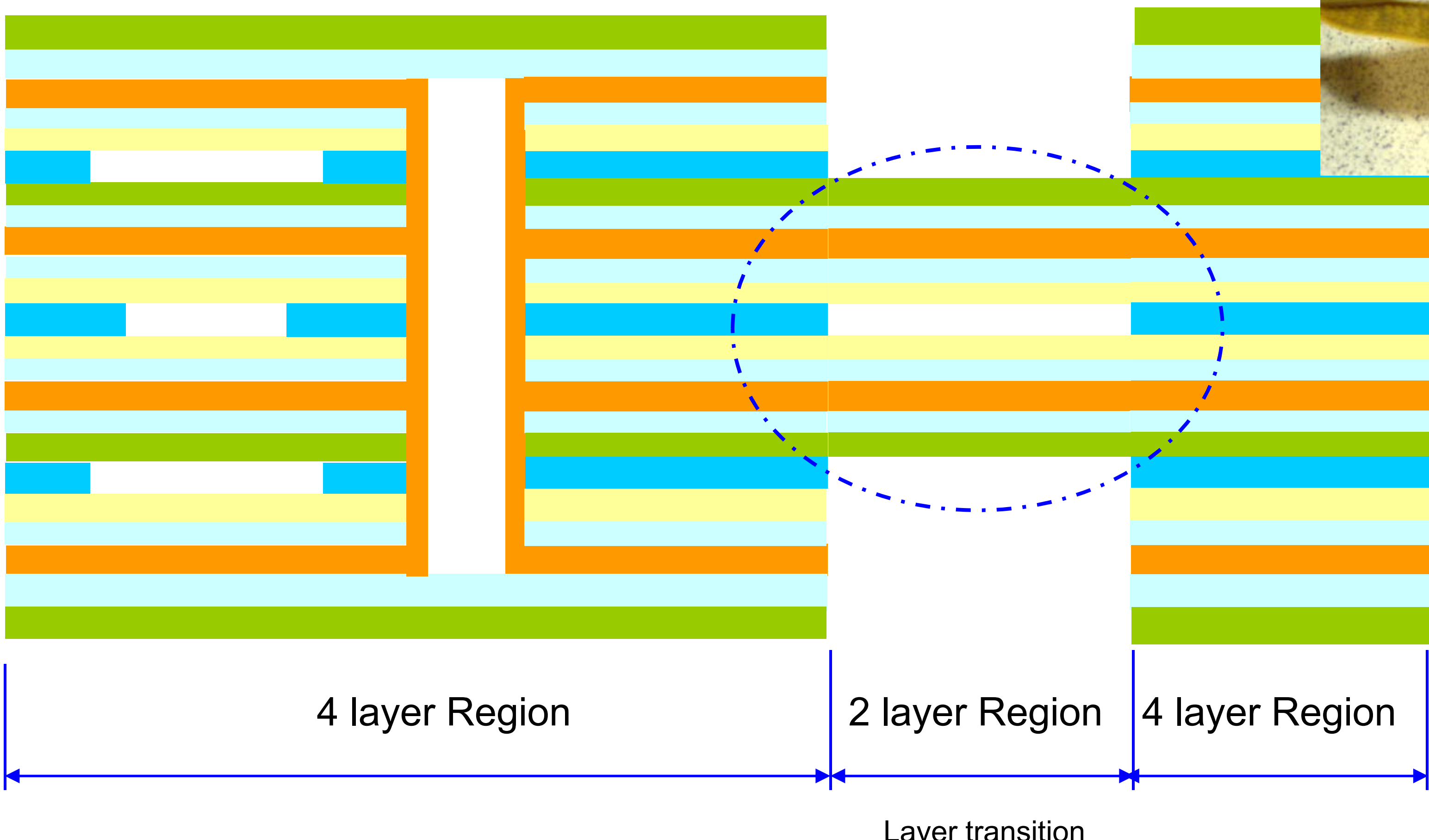
. The bending radius is minimum on the table.

Remove Partial Layer (Layer Transition)

For multi-layer flex, partial layers could be removed to increase the flexibility.

Air gap is another way to increase the flexibility. Impedance at this region will vary dramatically. However, the interval is very short.

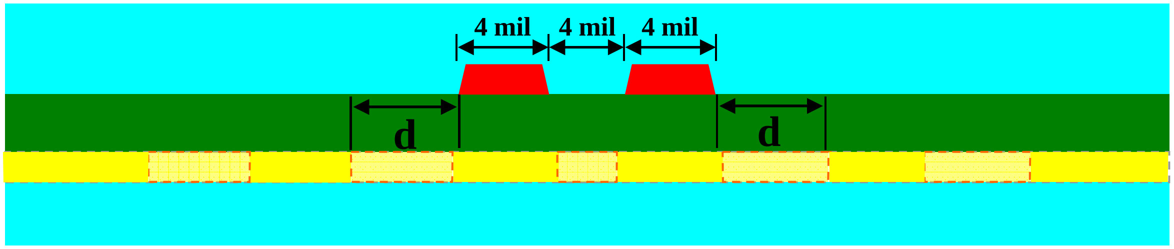




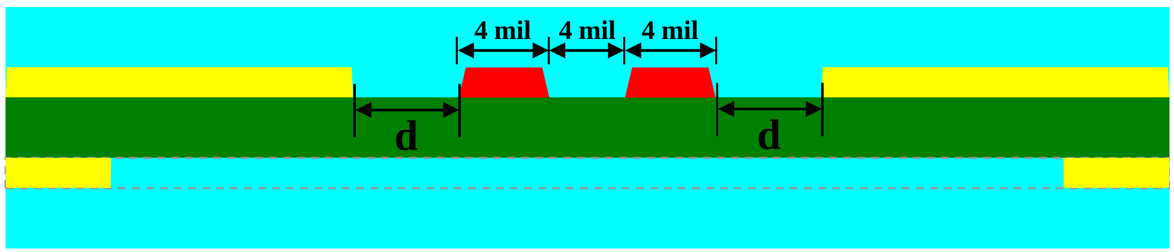
Novel Flex IMP Structures

Dynamic Flex Transmission Lines

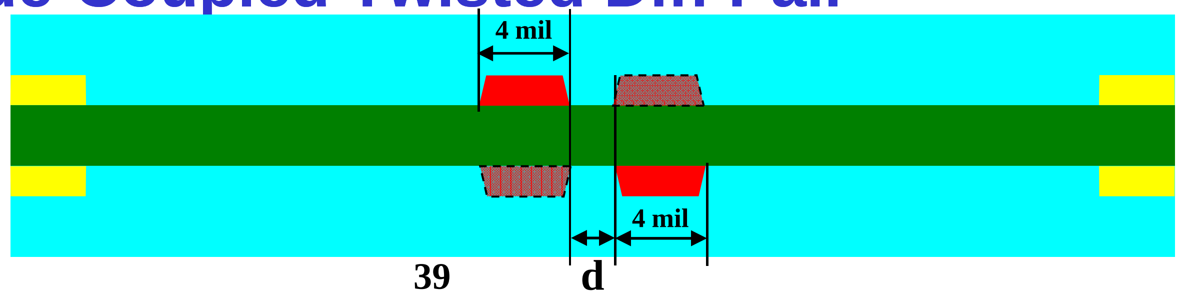
- Mesh GND Differential Microstrip



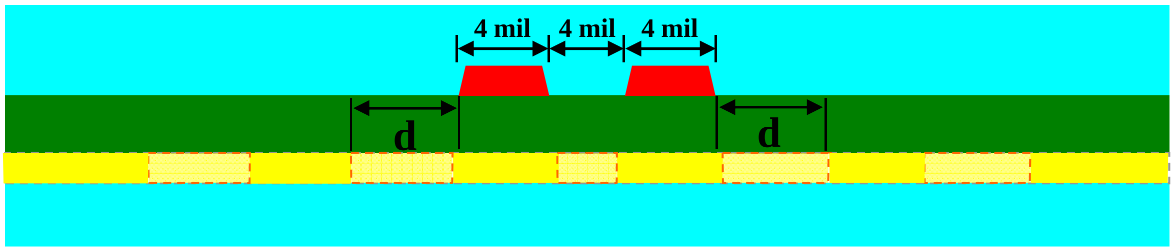
- Coplanar Differential Microstrip

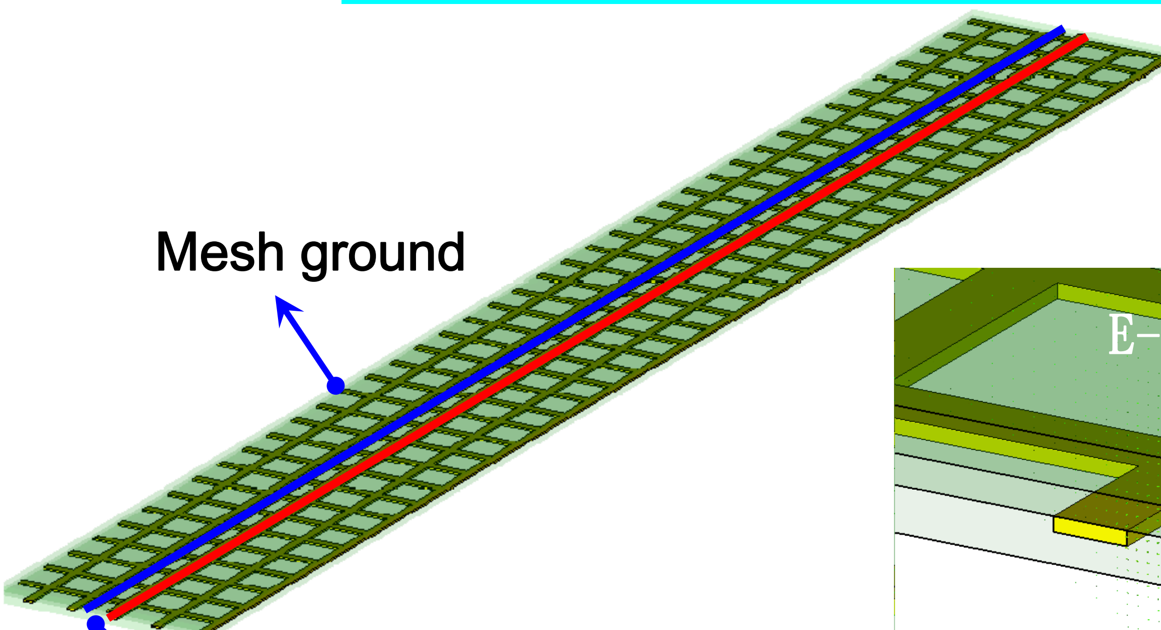


- Broadside-Coupled Twisted Diff Pair



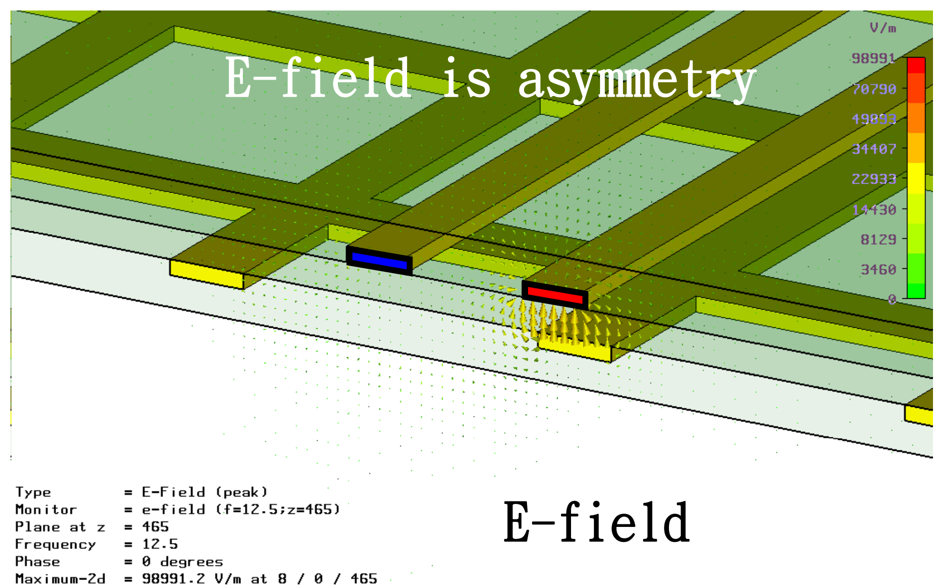
Mesh Ground Diff Microstrip





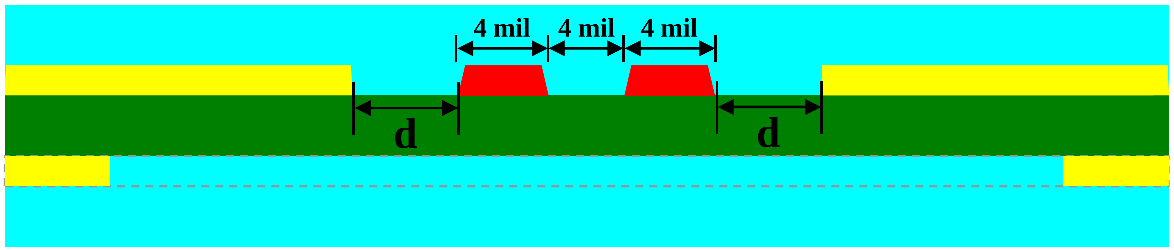
Mesh ground

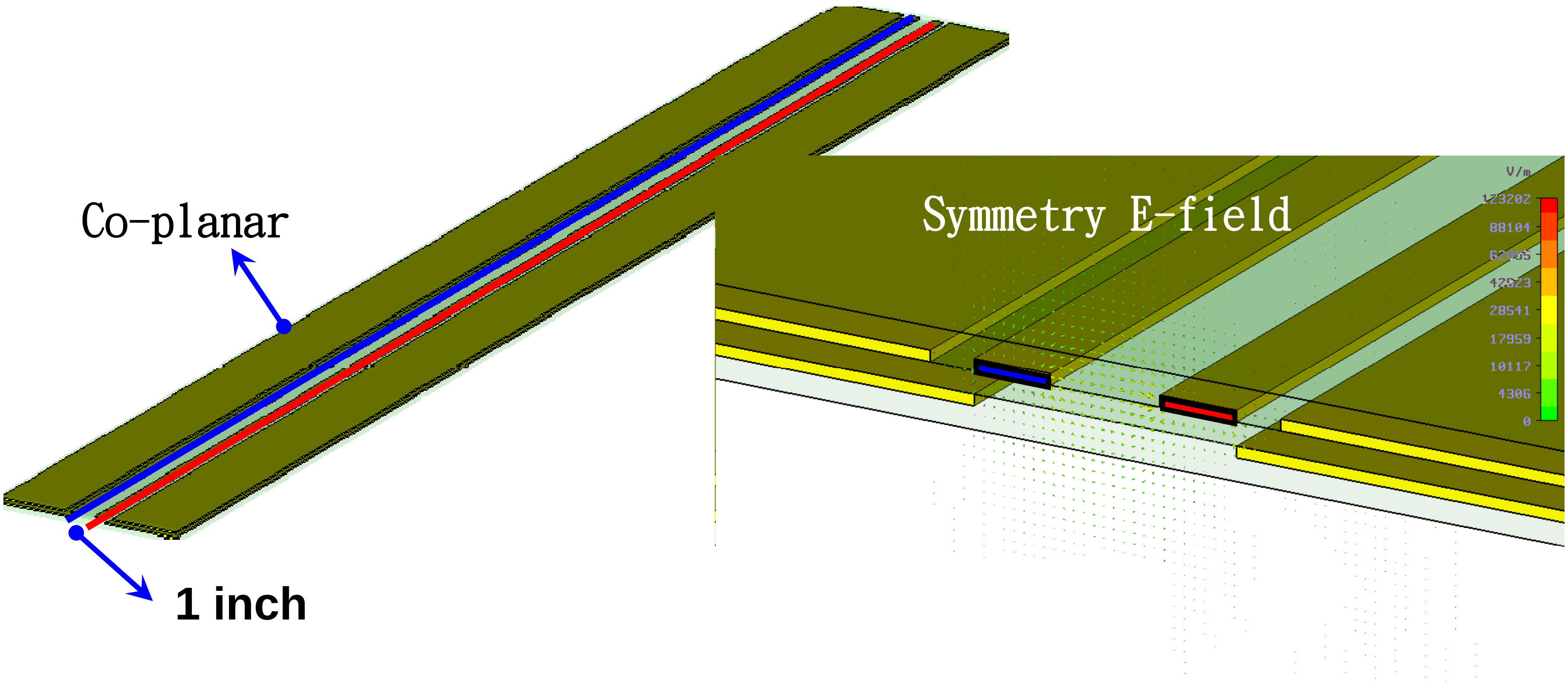
1 inch differential-pair



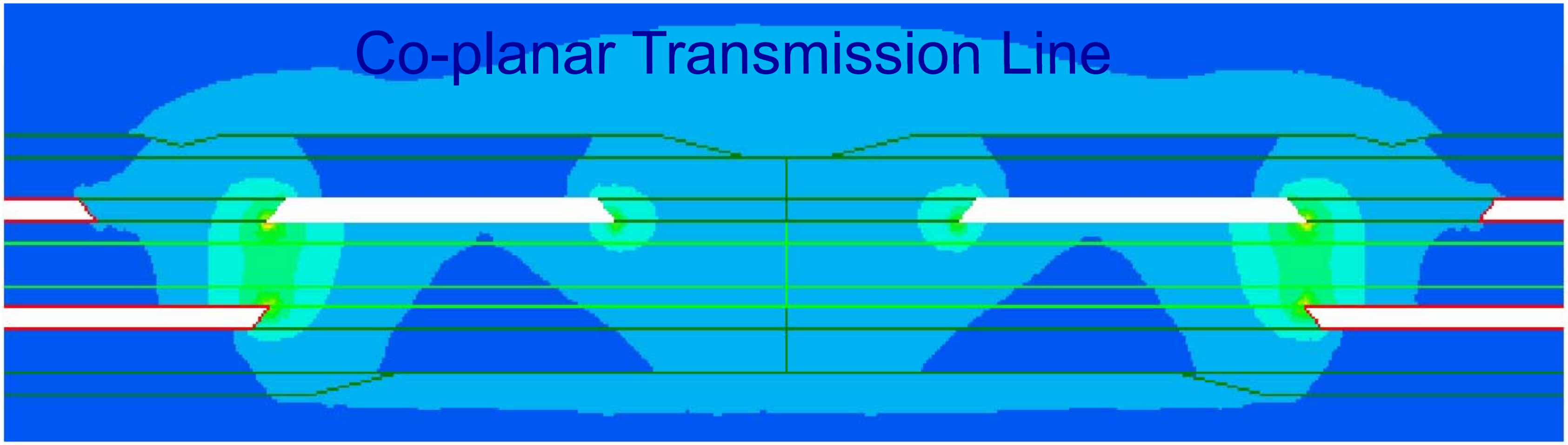
E-field

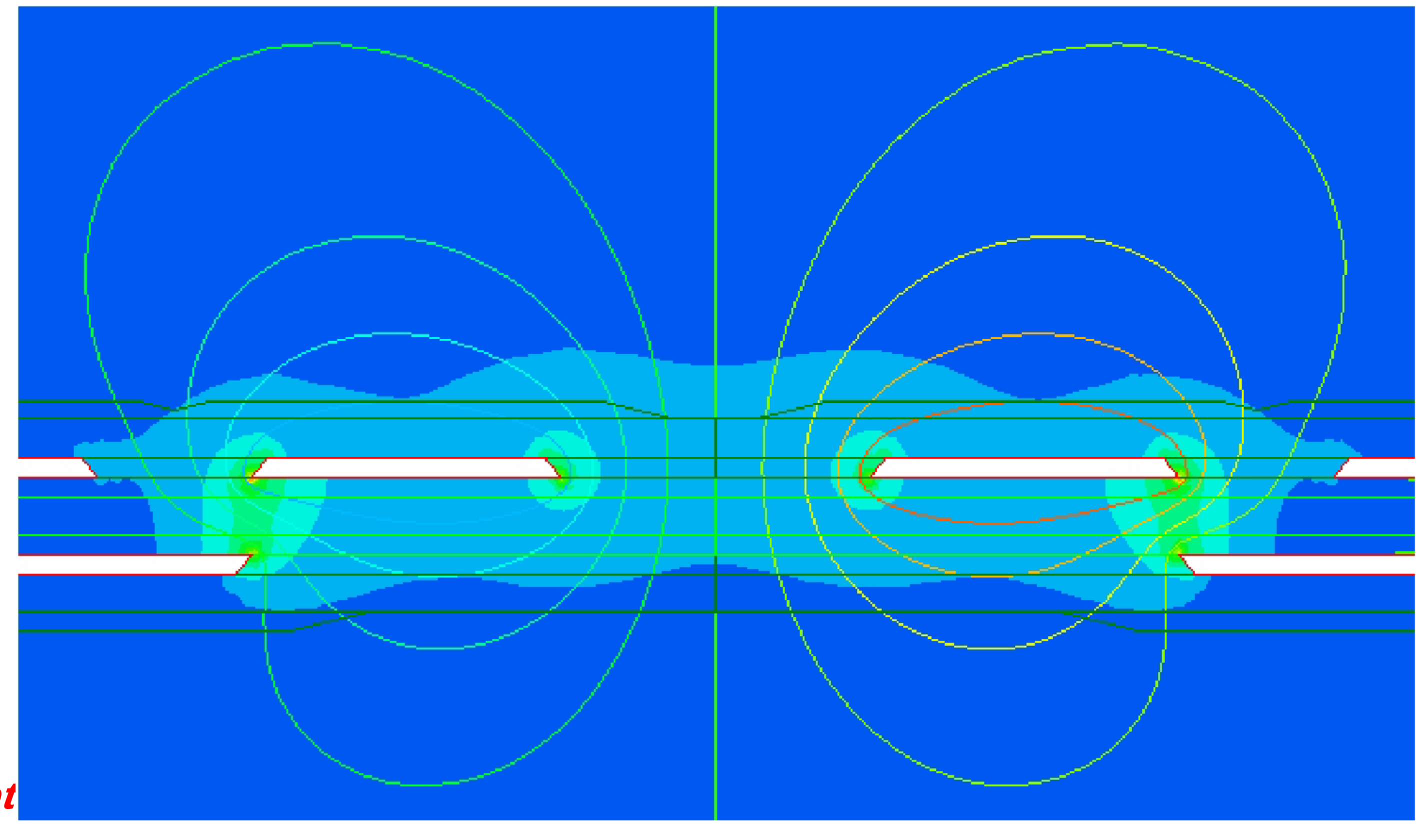
Co-planar Transmission Line



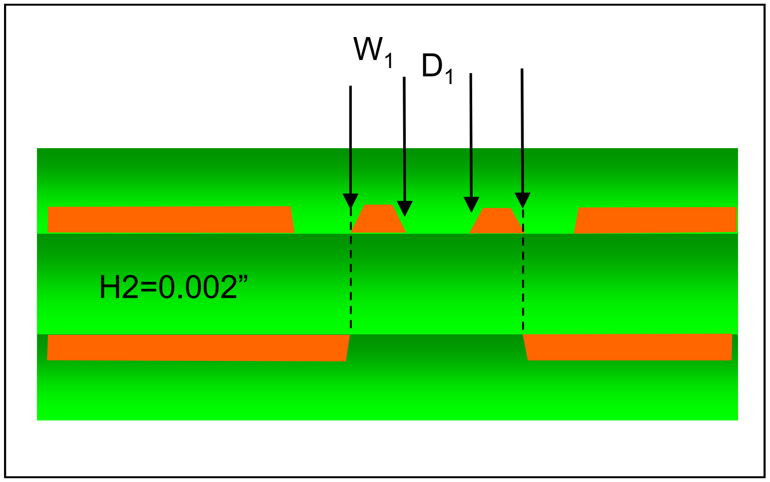


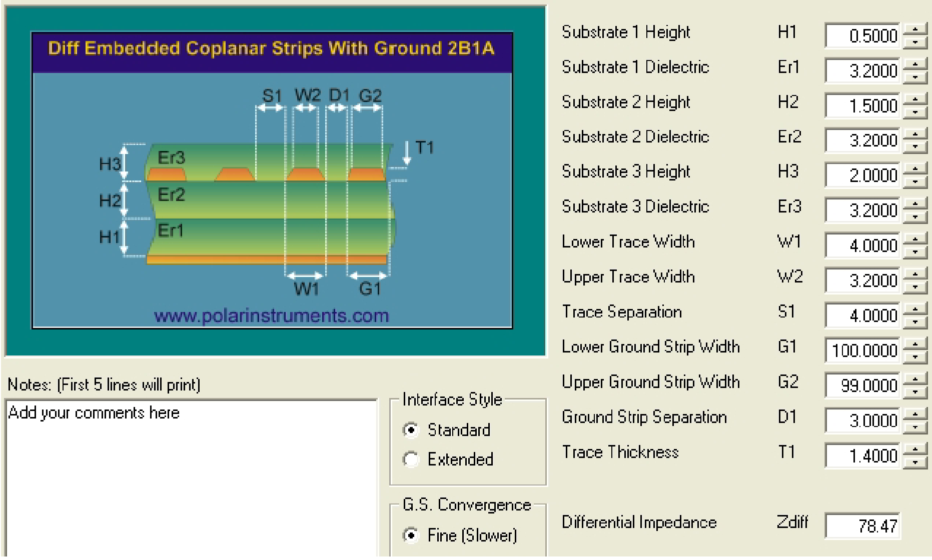
Co-planar Transmission Line



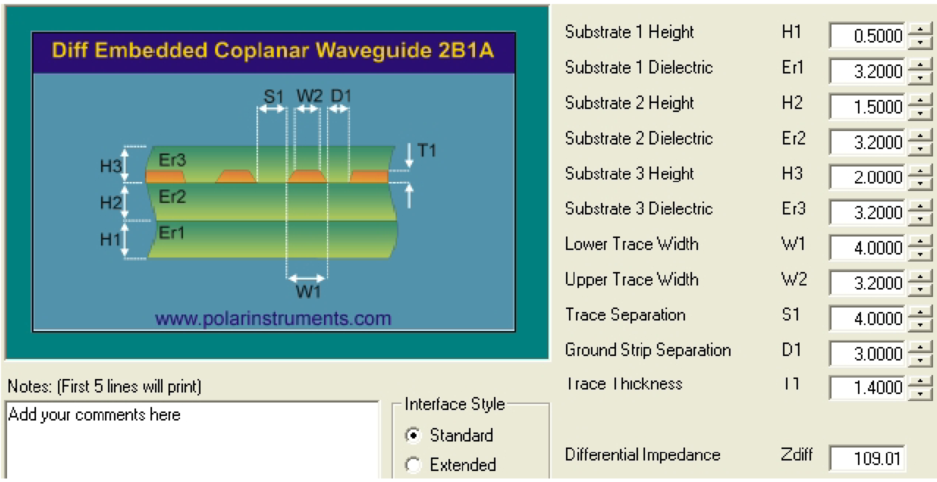


Embedded Coplanar Differential Stripline With-Ground (moved)



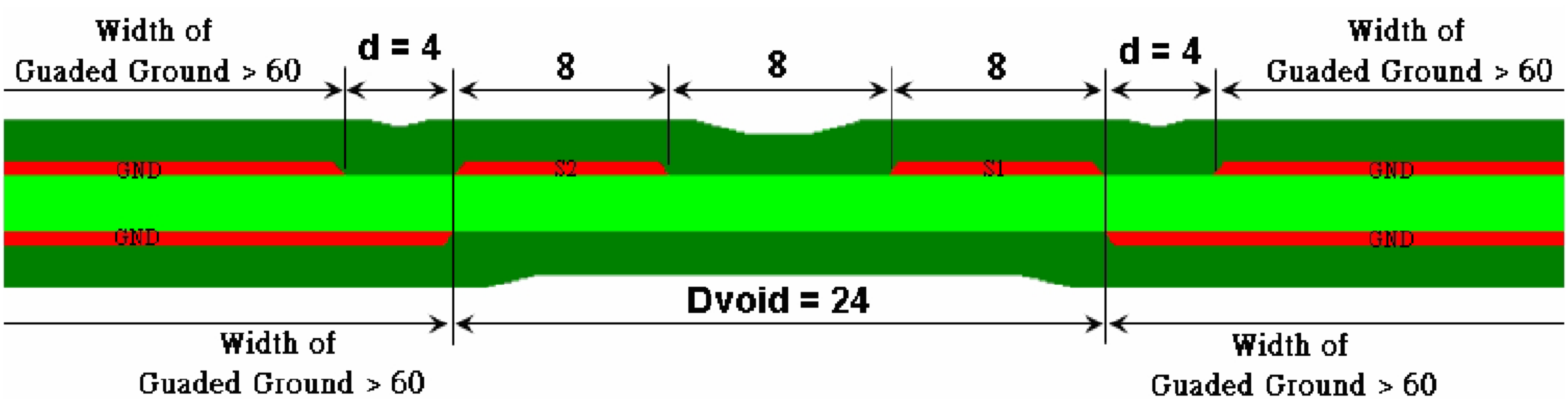


No Polar Model yet

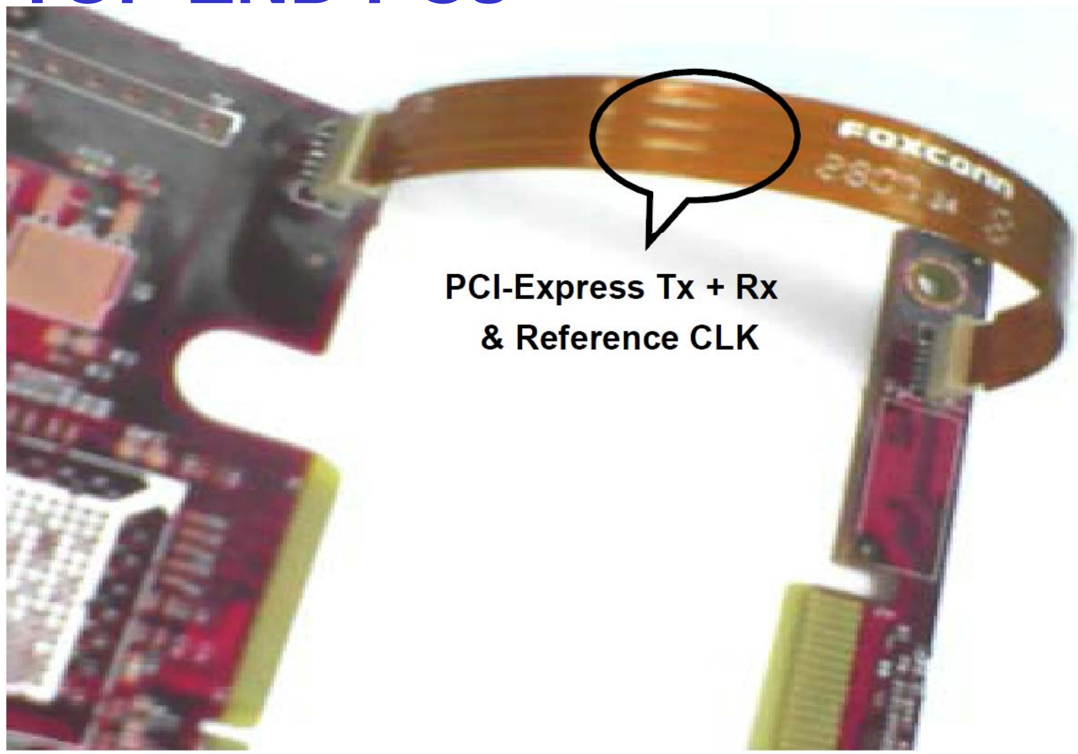


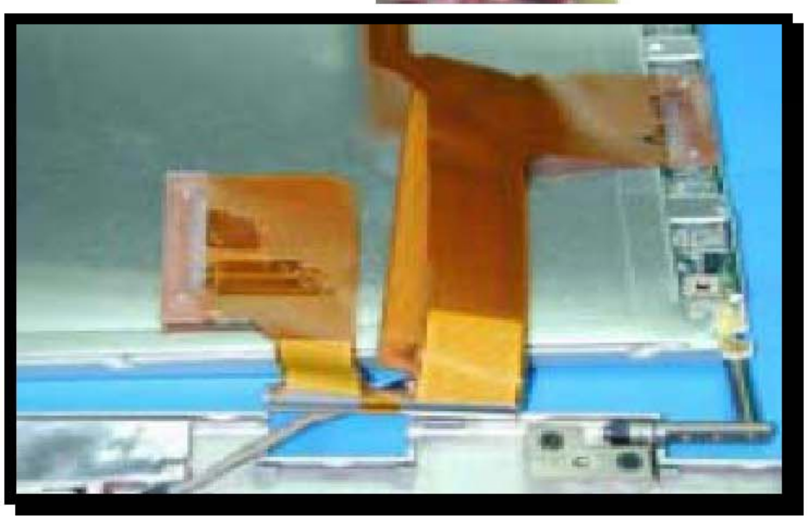
Stackup for Application

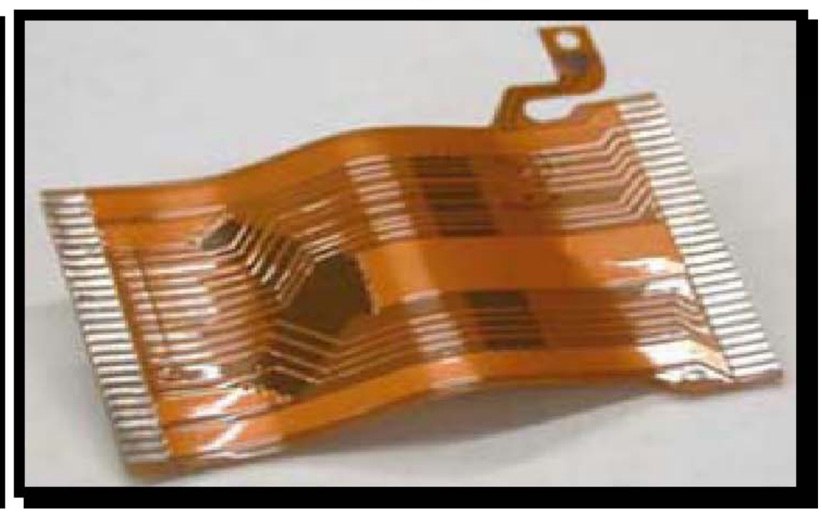
Name	Material	Copper Weight (oz)	Thickness (mil)	介電常數
CVL	AD		1.0	3.2
	PI		1.0	
Cu Plating	E-Cu		0.3	
CCL	Cu	0.5	0.7	
	AD		0.5	3.2
	PI		1.0	
	AD		0.5	
	Cu	0.5	0.7	
Cu Plating	E-Cu		0.3	
CVL	PI		1.0	3.2
	AD		1.0	
Total Thickness			8.0	



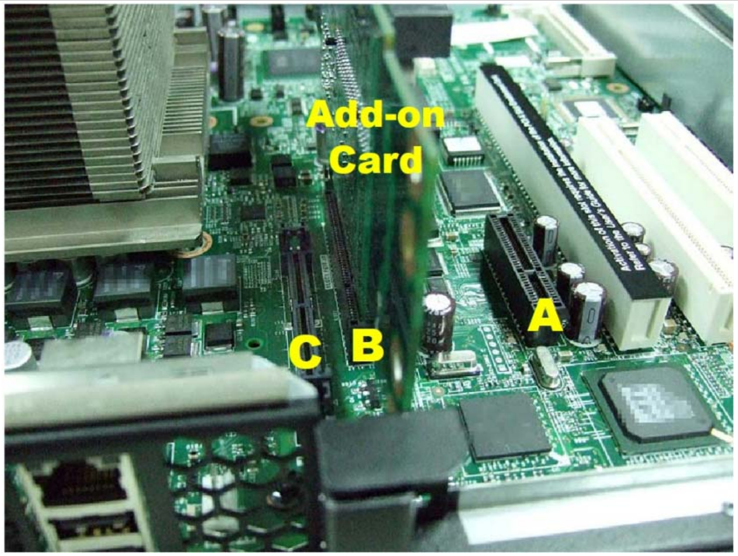
Application to TOP END PCs

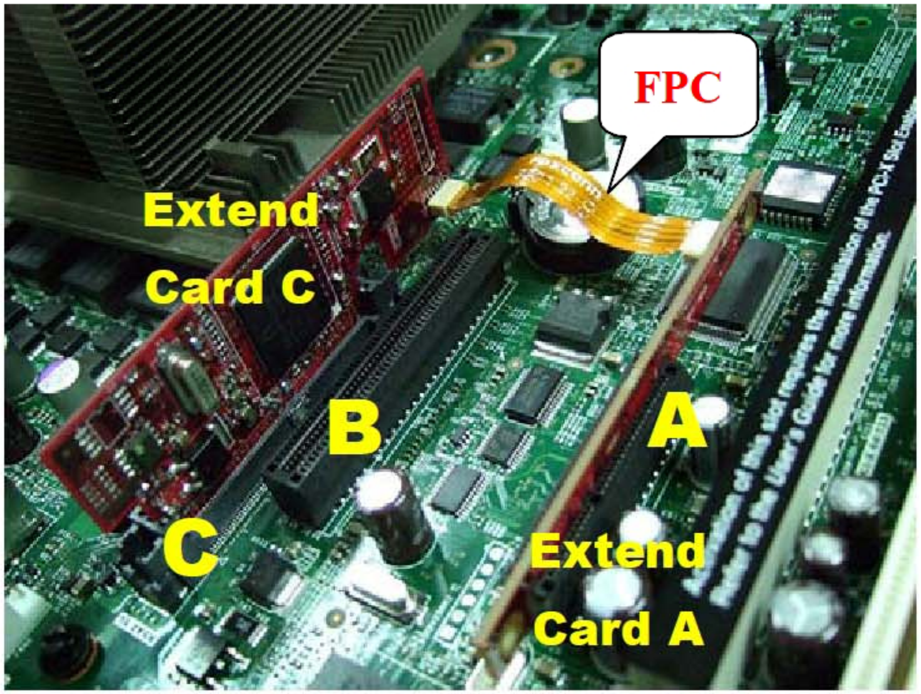


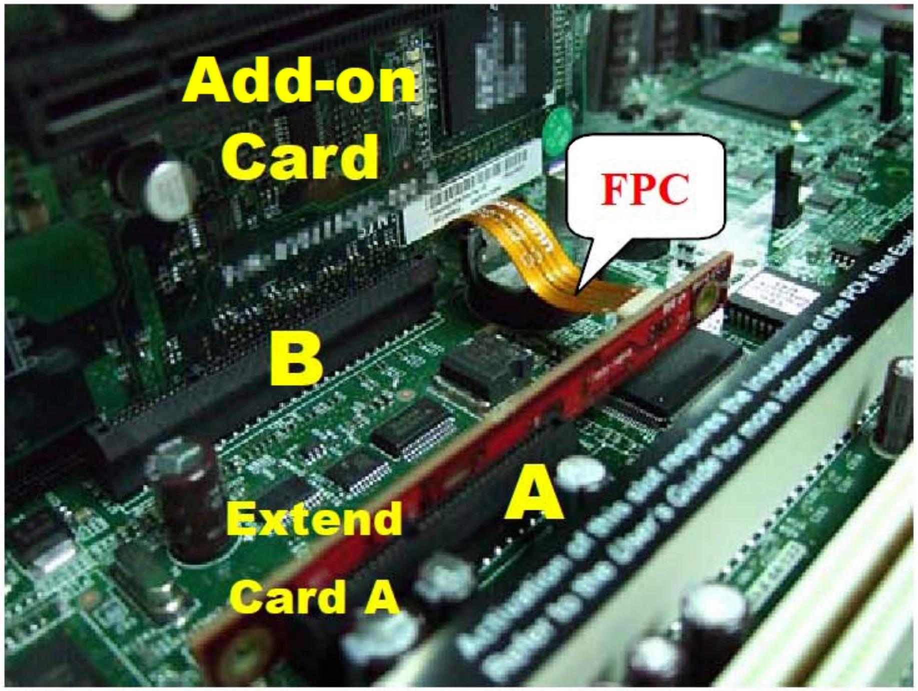




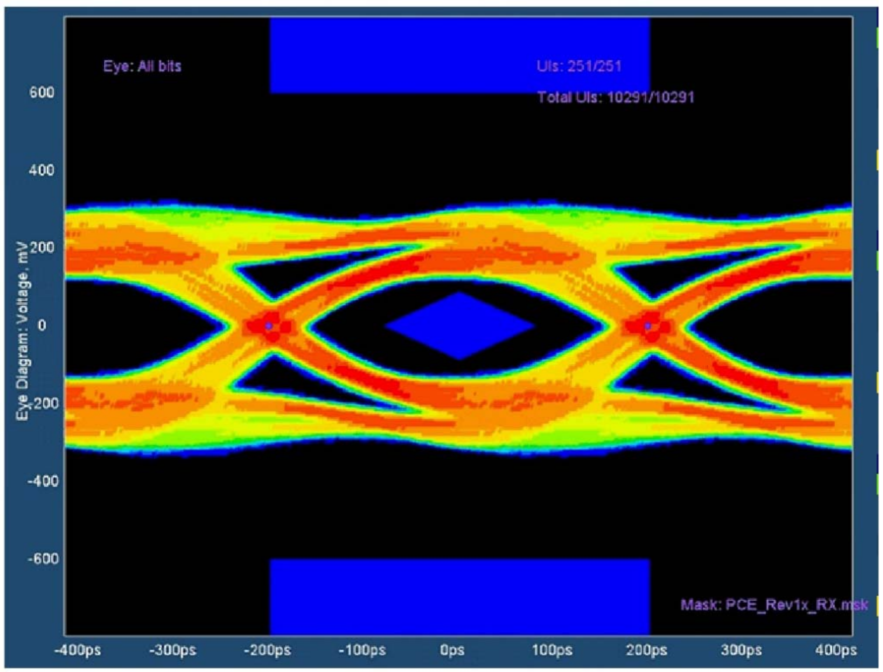
Application to TOP END PCs



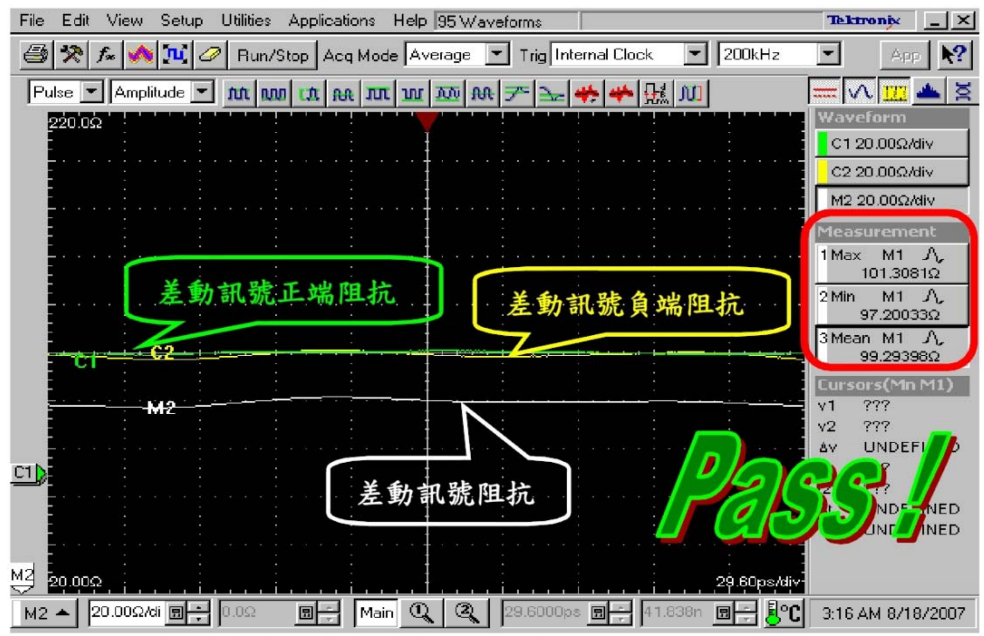




Performance Validation for TOP END PCs



十三、基於共平面傳輸線差動訊號軟性基板之 PCI-Express 訊號



圖二十一、時域反射阻抗量測 — 實際應用軟性基板之設計案例

Patented New Design Scheme — Broadside Coupling Differential Pair

- In order to avoid too close of the distance between the upper signal layer and the lower ground plane which leads to lower trace impedance, the design which claims removing the ground plane is proposed. Besides, P/N signal lines are routed on the upper and lower plane respectively.

- Change the offset distance d of the P/N signal lines to match the requirement of the target differential impedance.

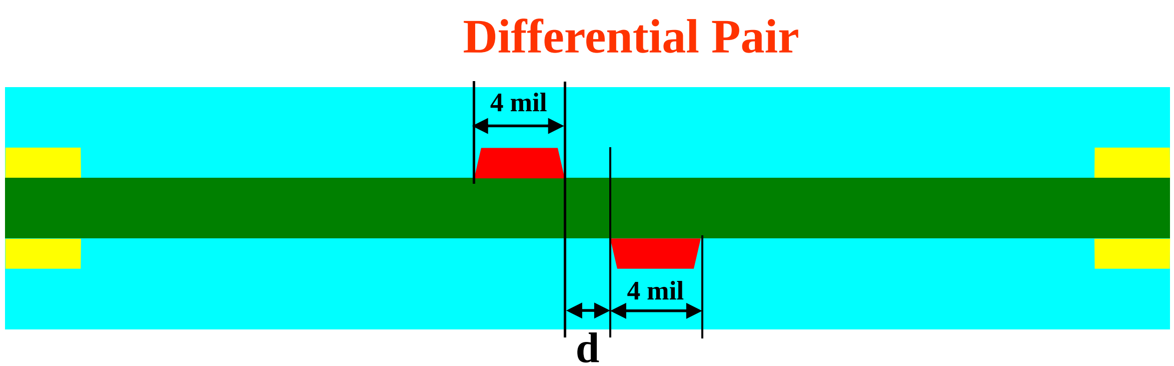


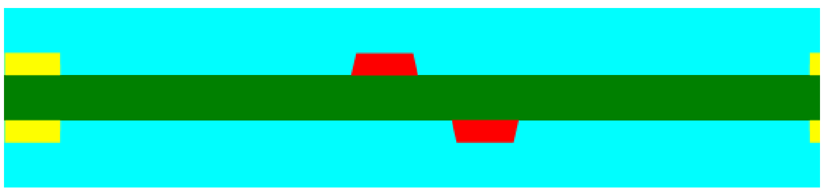
Figure. New Design Scheme：Change the offset distance (d) of the P/N signal lines on different layers to match the requirement of the target differential impedance.

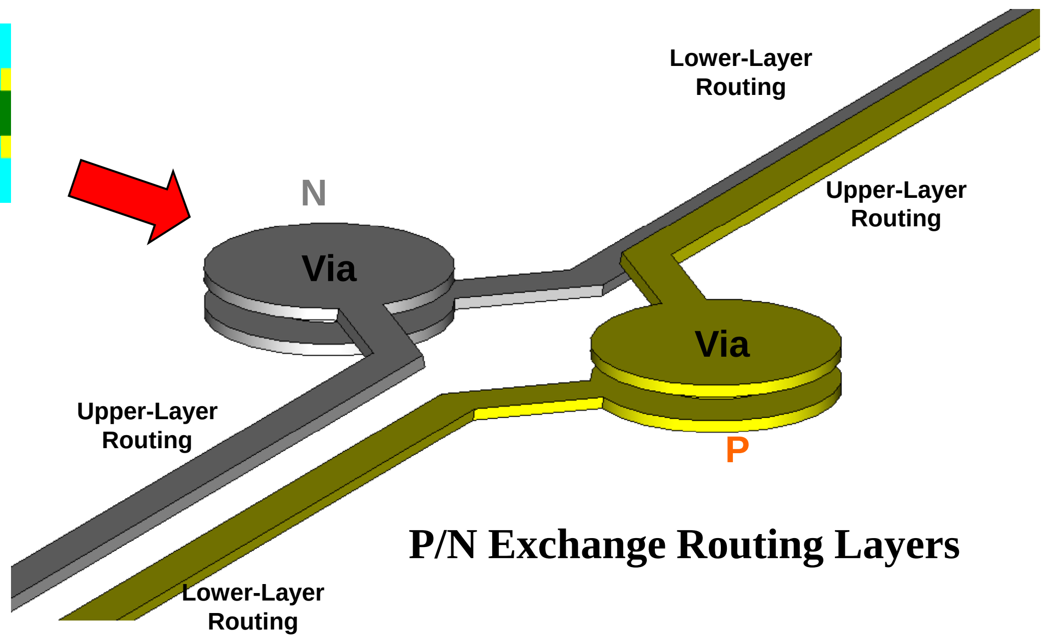
d (mil)	Zdiff (Ohm)	Signal Interfaces
5	120.847	
4.5	116.744	
4	112.339	
3.5	107.789	1394 (110 Ohm) (between 4 and 3.5)
3	103.115	
2.5	98.337	PCI-Express Gen1 (100 Ohm) (between 3 and 2.5)
2	93.298	
1.5	88.198	USB2.0 (90 Ohm) (between 2 and 1.5)
1	82.83	PCI-Express Gen2 (85 Ohm) (between 1.5 and 1)
0.5	77.564	
0	72.096	

Table. The relationship of offset distance d and Zdiff.

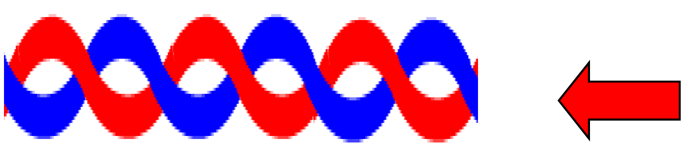
New Design Scheme —Twisted Broadside Coupling Differential Pair

- Based on the proposed broadside-coupled differential signal design, an extended design scheme called twisted differential pair signals is proposed.
- Characteristic: P/N signals continually exchange the routing layers at a specific distance, which mimics the twisted-cable-like routing.



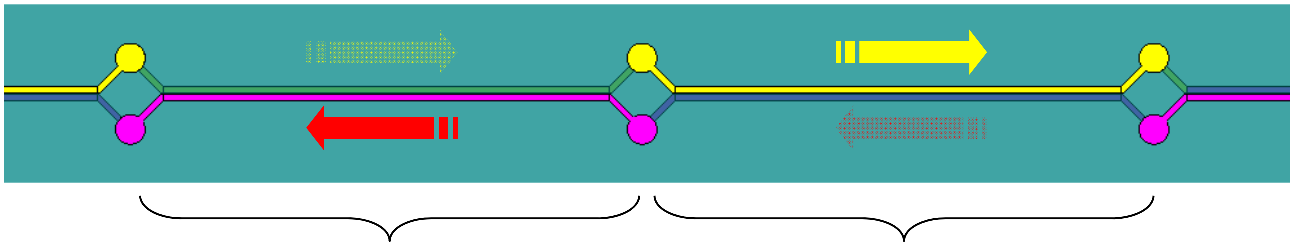


P/N Exchange Routing Layers

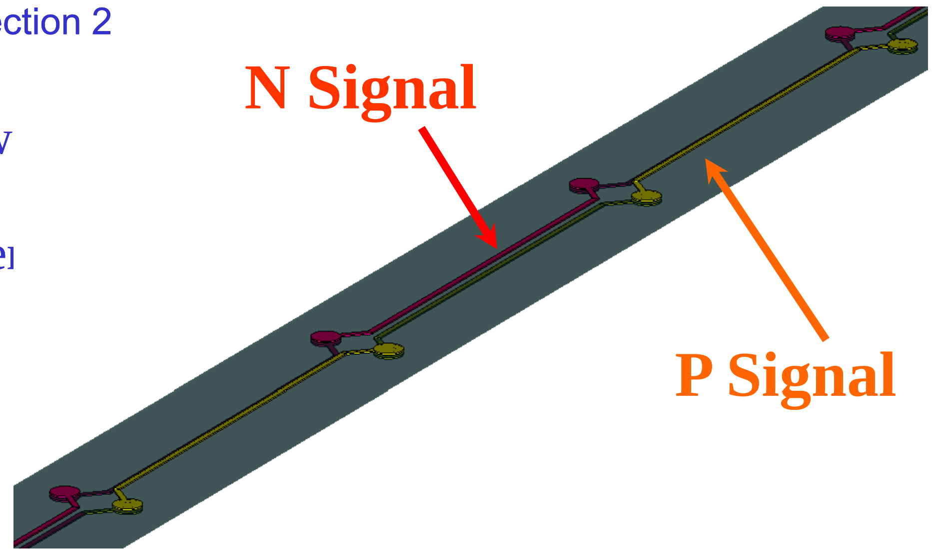


Conventional Twisted Cable

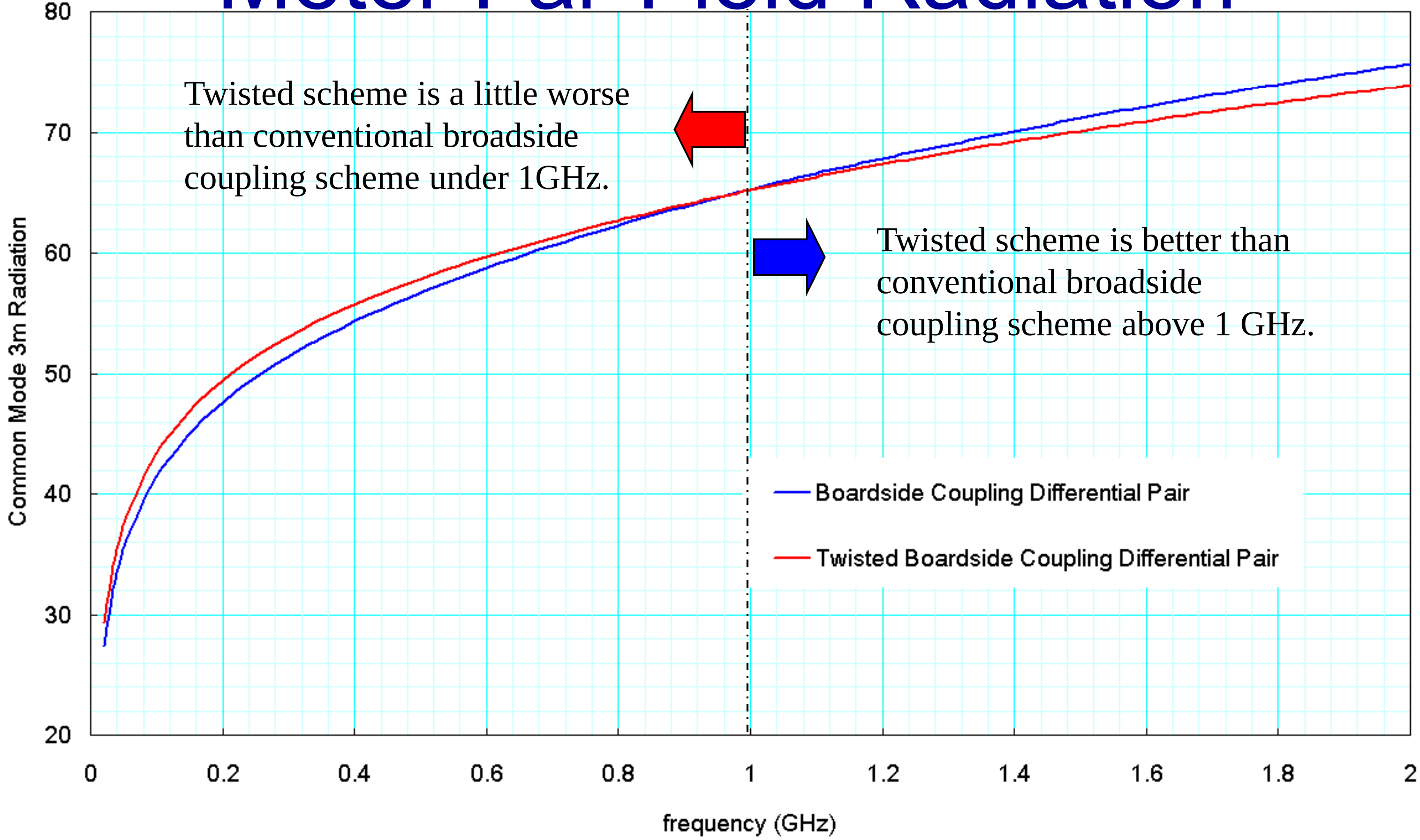
New Design Scheme —Twisted Broadside Coupling Differential Pair



- On the same layer, the current flow direction through Section 1 and Section 2 is inverse and can further suppress EM radiation.



Simulation Results — Common Mode 3-Meter Far-Field Radiation



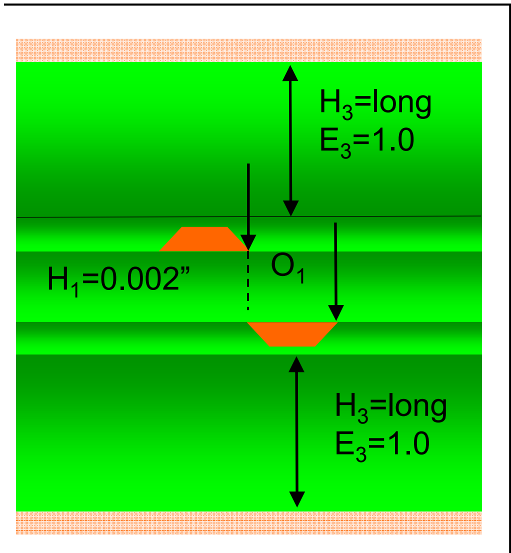
Comparison of Different Design Schemes to Fix Low Impedances in FPC

	Conventional Mesh Ground	Twisted Broadside Coupling Diff. Pair
EMI Suppression	Acceptable	Good
Measured TDR Impedance	Bad	Good
Common-mode Noise Suppression	Bad	Good
Routing Feasibility	Acceptable	Good

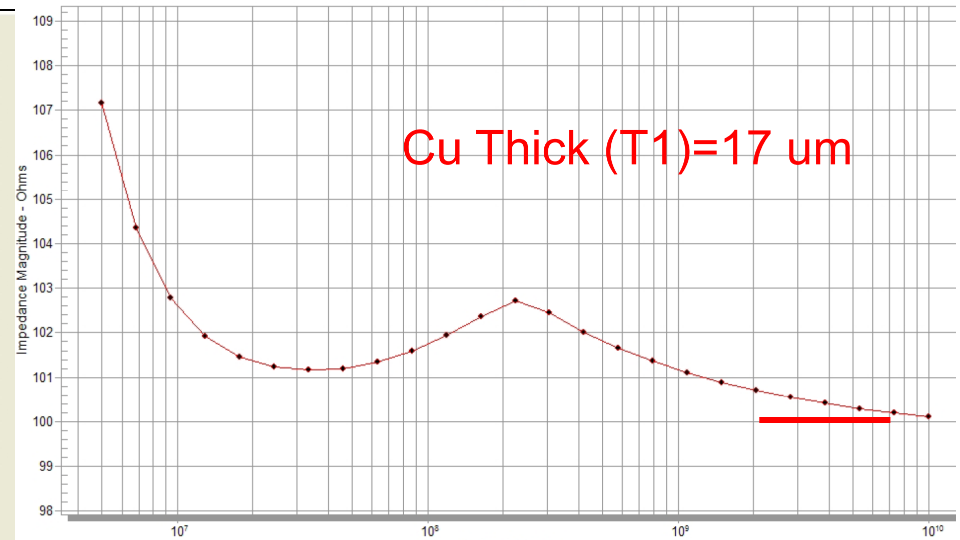
*** The proposed twisted broadside coupling differential pair, which combines the concept of the previously-published broadside coupling differential scheme, not only can precisely control Transmission line impedance but also can further suppress EMI effect.**

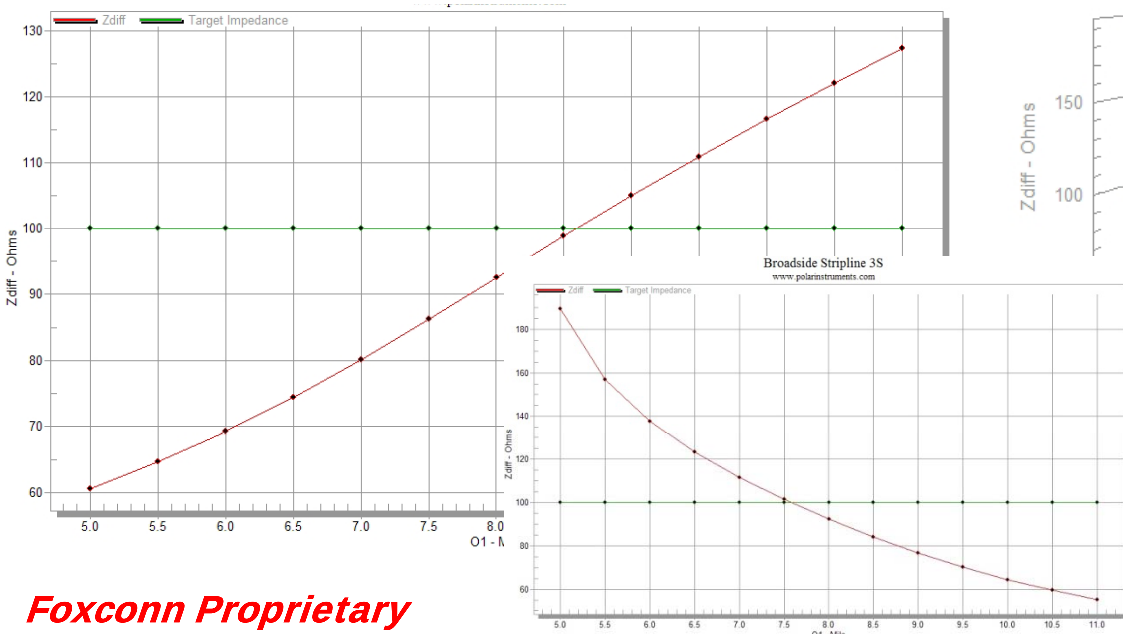
*** This new proposal only adjusts the routing manner without extra cost, and is therefore considered an cost-effective and novel solution.**

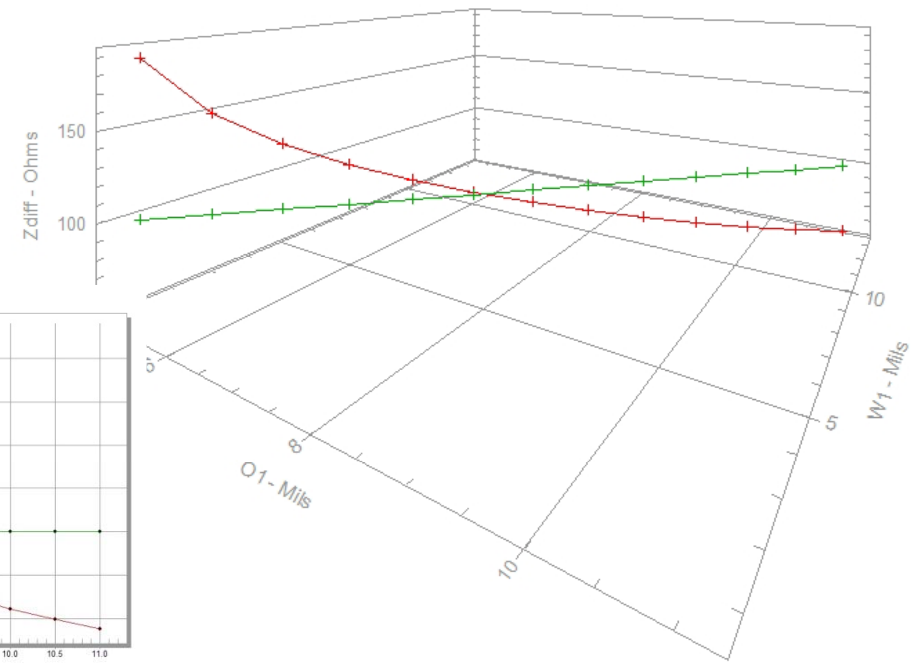
Broadside Differential Stripline With-Out Ground (moved)



Substrate 1 Height	H1	2.0000
Substrate 1 Dielectric	Er1	3.2000
Substrate 2 Height	H2	2.0000
Substrate 2 Dielectric	Er2	3.2000
Substrate 3 Height	H3	49.9998
Substrate 3 Dielectric	Er3	1.0000
Lower Trace Width	W1	4.0000
Upper Trace Width	W2	3.2000
Trace Offset	O1	4.9000
Trace Thickness	T1	0.7700
Differential Impedance	Zdiff	100.57







Foxconn Proprietary

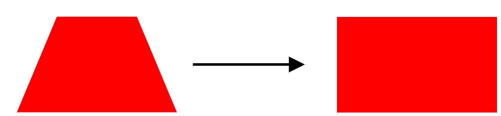
Conclusions

- **There is a number of novel ways to implement high-speed transmission lines, either single-ended or differential for dynamic flex applications.**
- **The highest performance is achieved when the Reference GND plane is moved, removed or de-populated (Mesh).**
- **All of these structures are being employed for the newer high-speed bus structures in computers.**

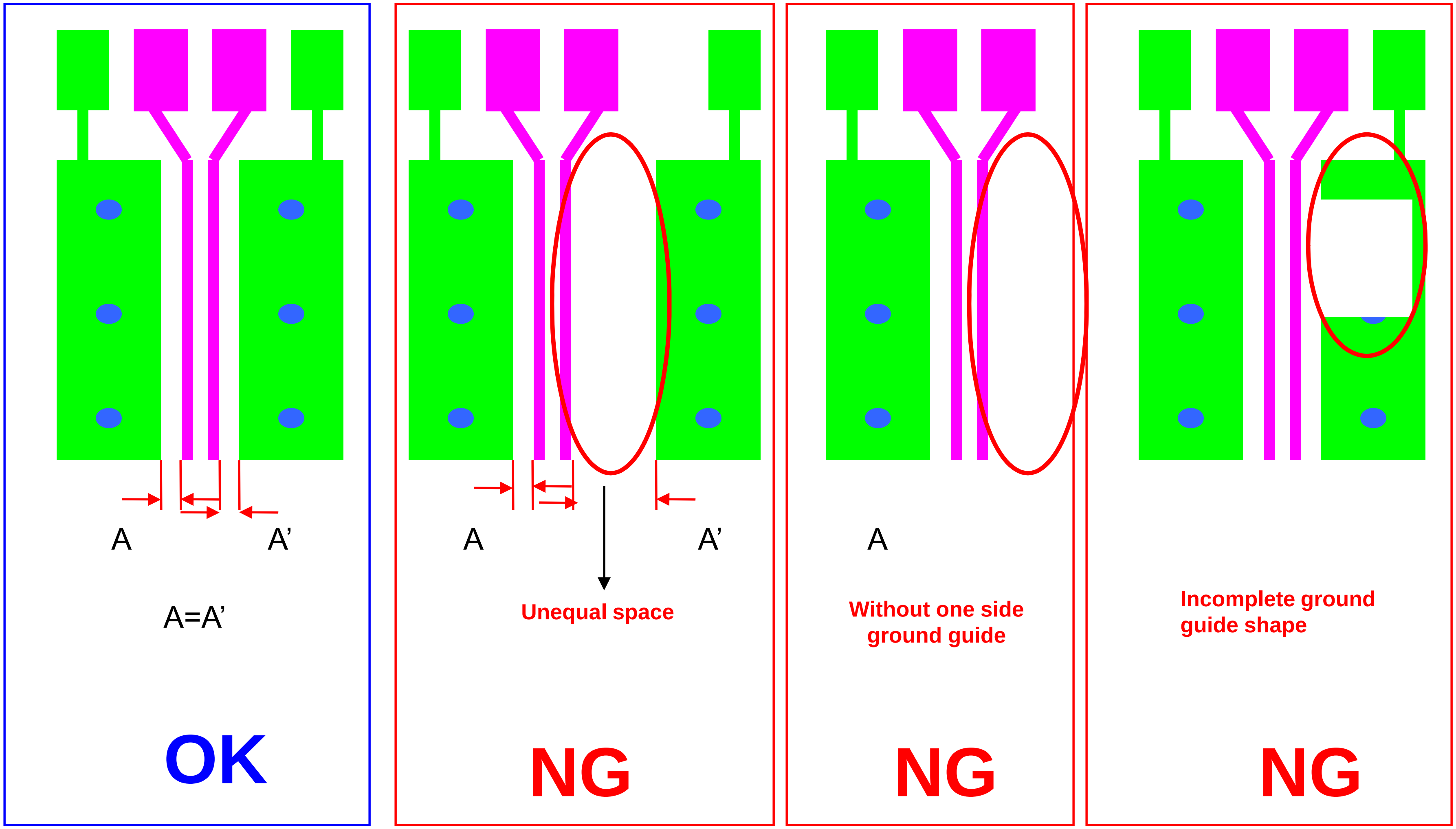
THANK-YOU FOR YOUR TIME TODAY

Impedance Design Suggestion

How to Improve Impedance Tolerance

Dielectric Constant	How accurate?
Dielectric Thickness	How accurate of the thickness? a. Adhesiveless vs. adhesive base
Trace width	How accurate of trace? a. Dry film selection b. Exposure machine parameter optimization c. Etching parameter optimization d. Width uniformity
Copper thickness	a. Incoming material inspection? b. Process control?(+/-100u”) c. Button plating?
Etching factor	a. How to increase etching factor? b. Semi-additive process implement? 
Cross mesh design	a. How to route or design trace? The angle between the signal trace and mesh.

Impedance Design Suggestion(I)-Equivalent Space

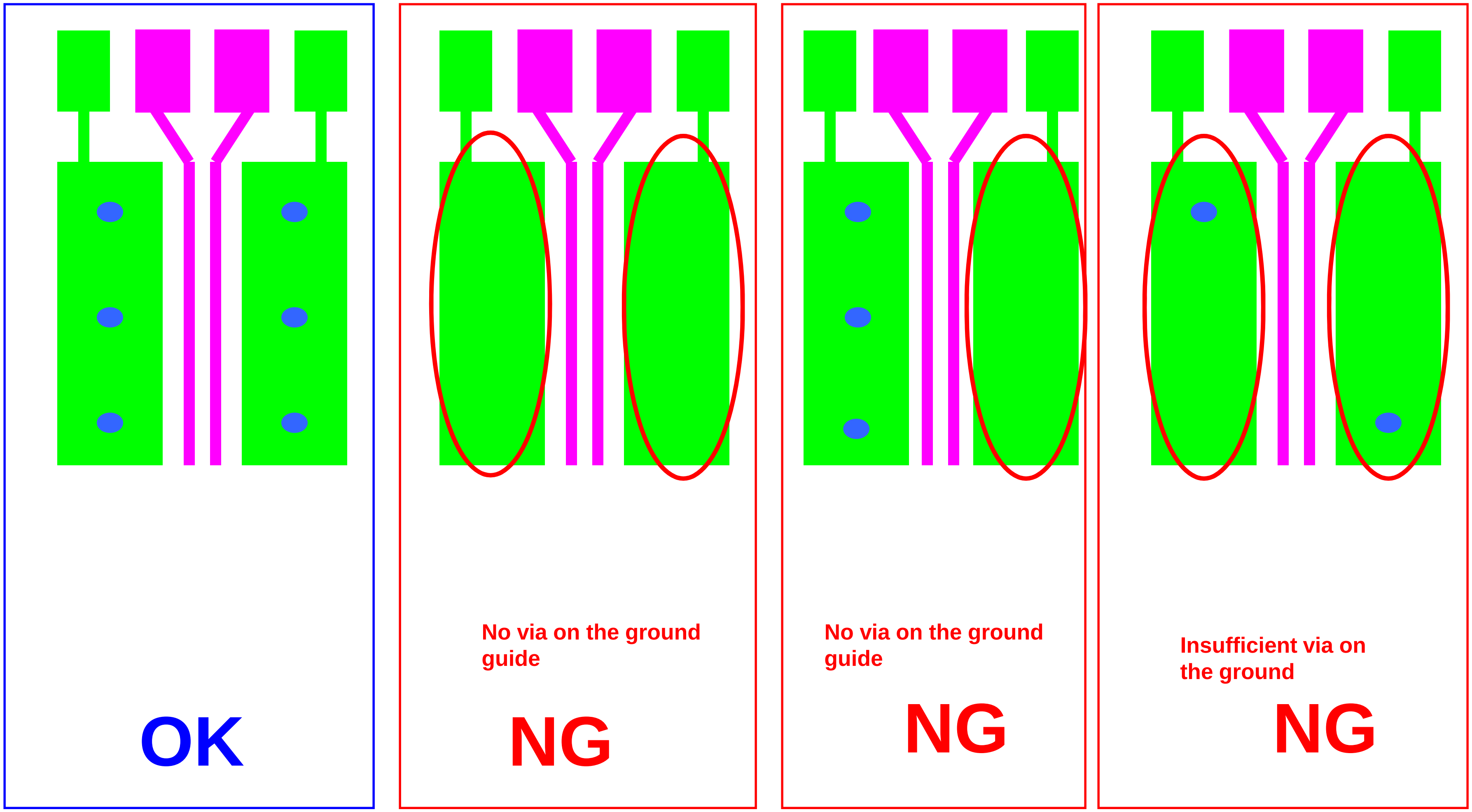


Key message:

. For differential pair,the two signal traces width would be the same.

. All spaces have to be equal.Otherwise, it will cause the impedance of signal trace without complete ground guide unstable.

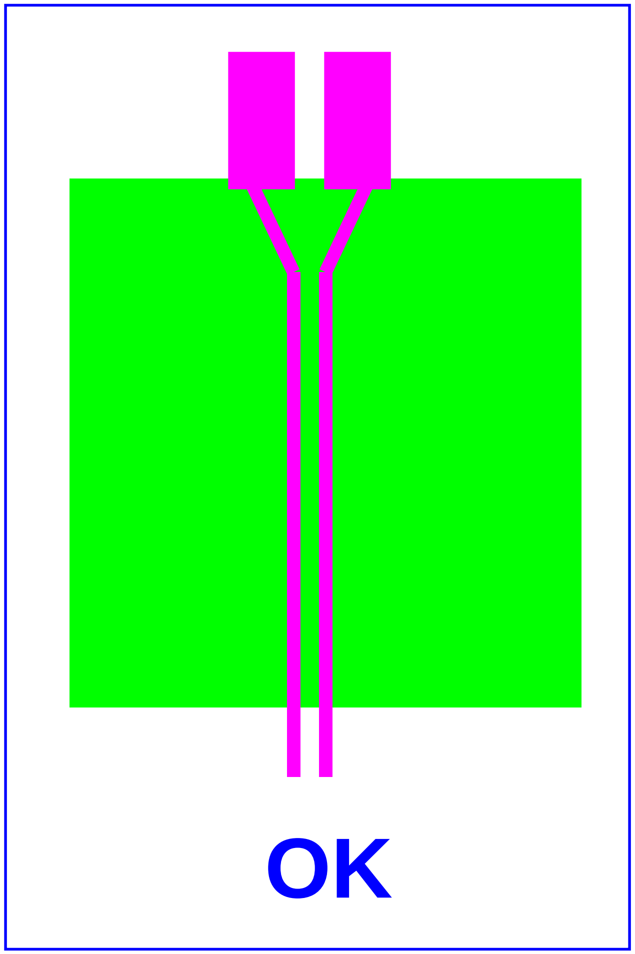
Impedance Design Suggestion(II)-Missed Ground Via

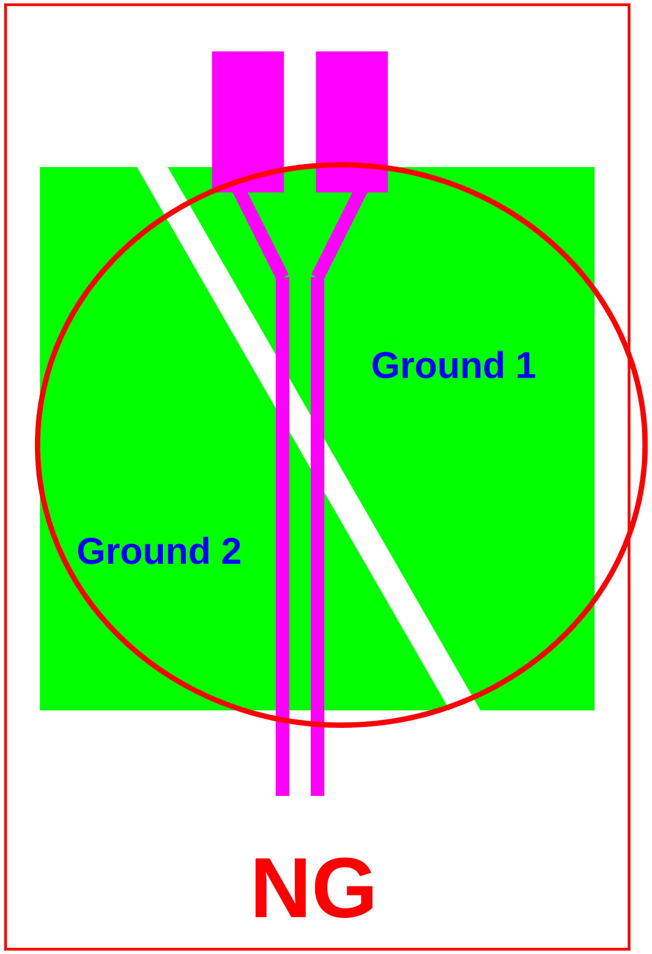


Key message:

. For ground guides, it needs vias to connect the ground plane to form a better performance.

Impedance Design Suggestion(III)-Across Two Ground Reference

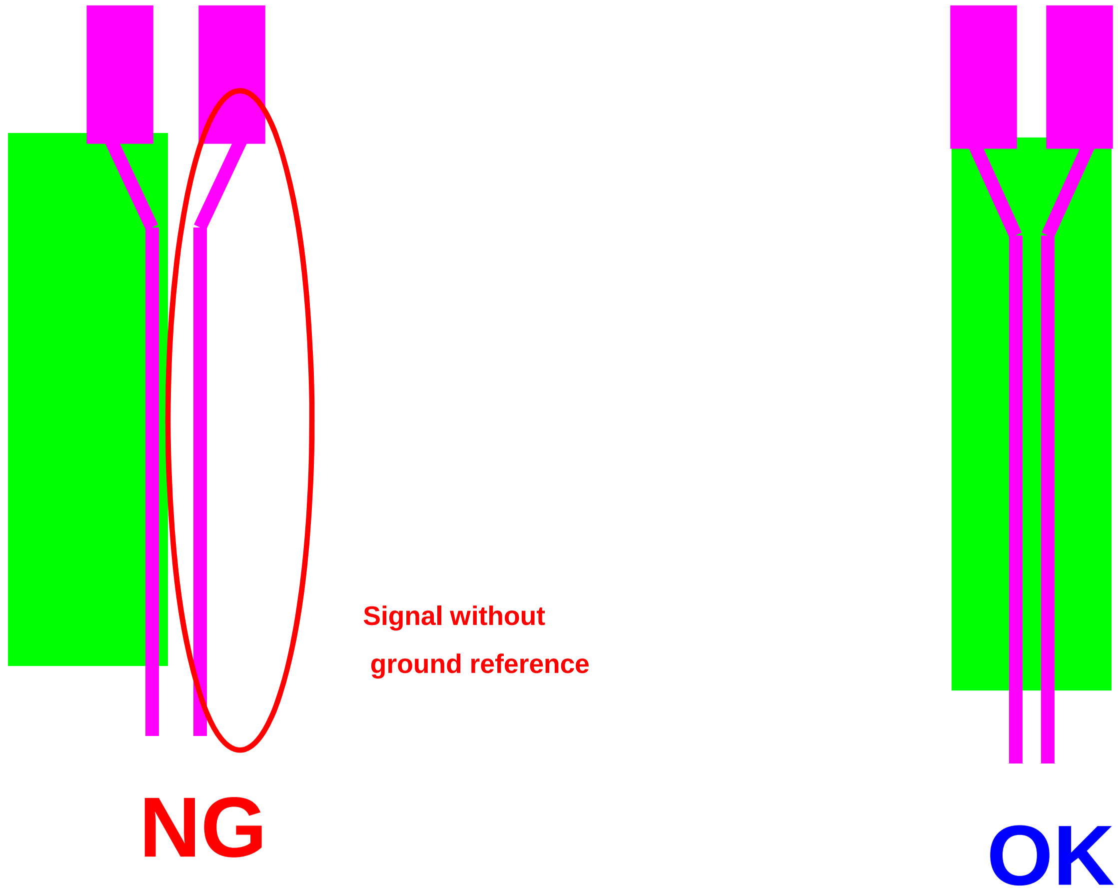




Key message:

. For ground plane, the signal passes through mono ground plane but mixed grounds

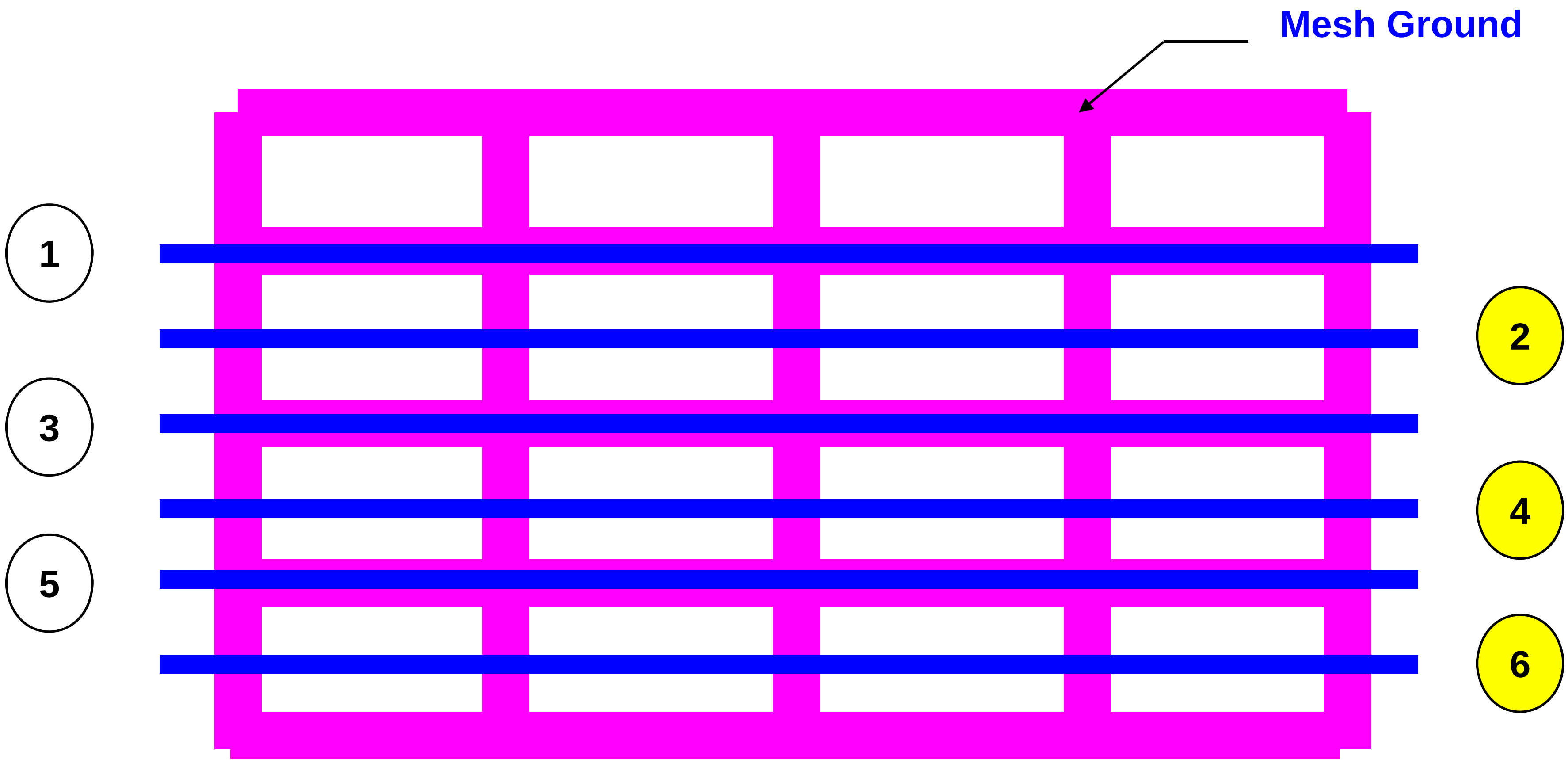
Impedance Design Suggestion(IV)-Missing Ground Plane



Key message:

. The signal has to be located over the ground plane.Otherwise, it will cause impedance unstable.

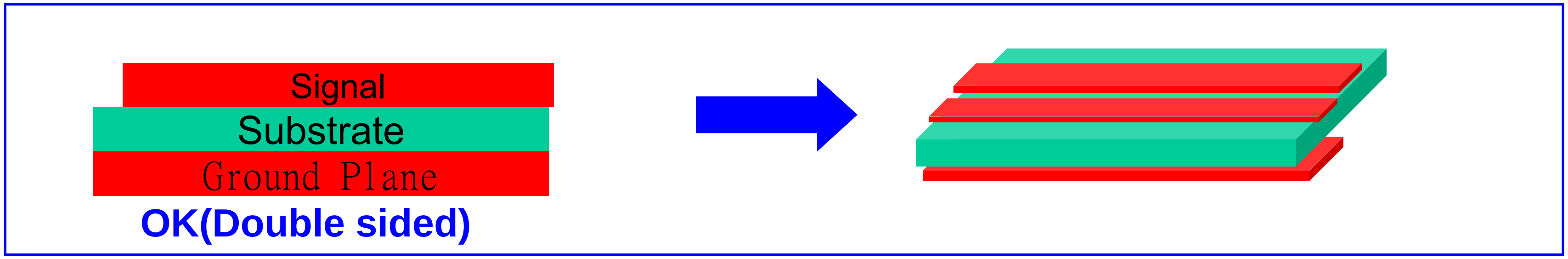
Impedance Design Suggestion (VI)-Improper Mesh Distribution

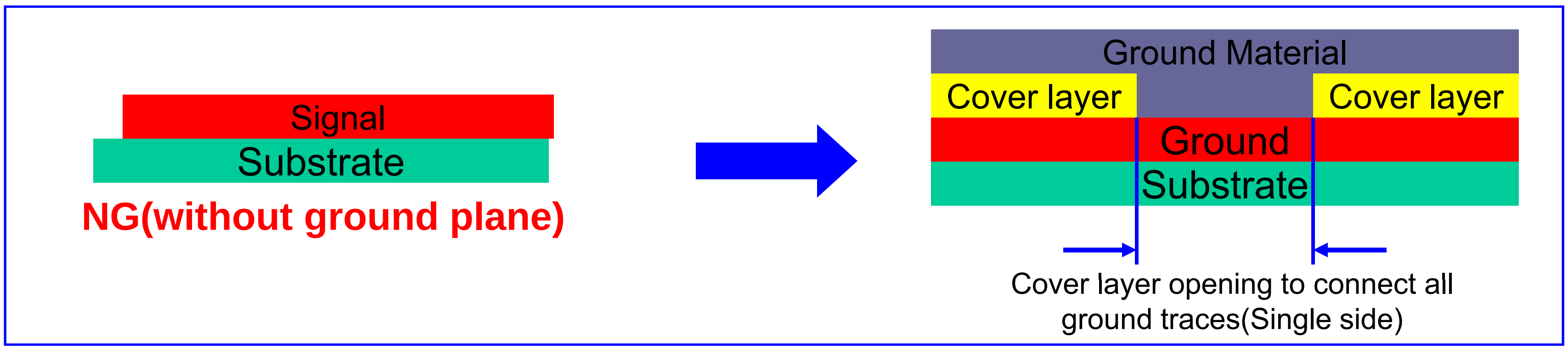


Key message:

. There are 6 transmission lines with same width (single end) travelling across the mesh ground. But impedance of group 1 (1,3,5) will differ from that of group 2(2,4,6). Because group 2 will behave like solid ground.

Impedance Design Suggestion(V)-Adding Extra Ground Plane





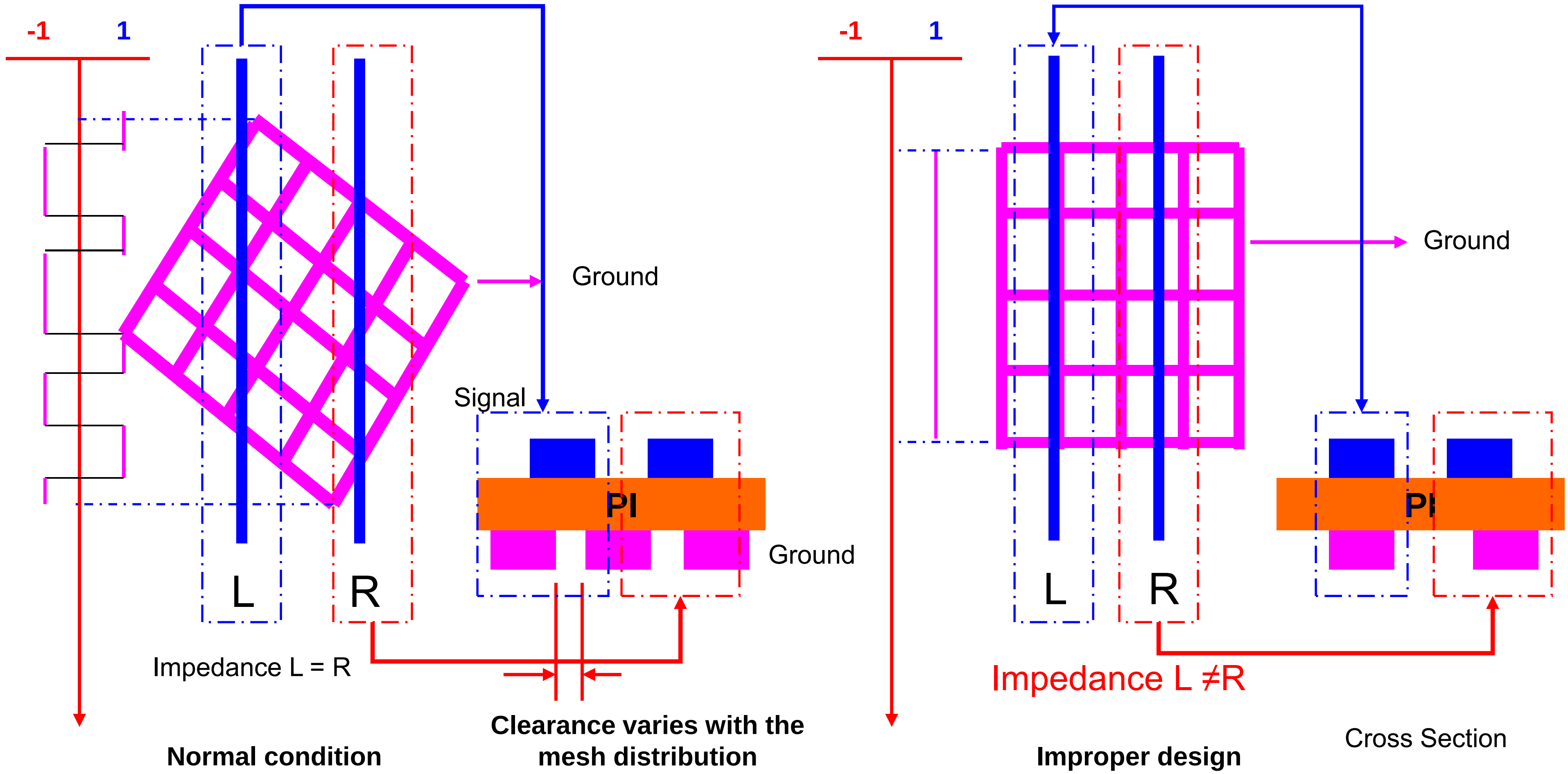
Key messages:

. Impedance varies with different types of ground due to capacitance variations.

. To achieve impedance stability, it is imperative to keep the ground pattern/angle uniform across various regions of the flex.

Impedance Design Suggestion(VI)-Uniform Ground Distribution

1: With ground *-1:Without ground*

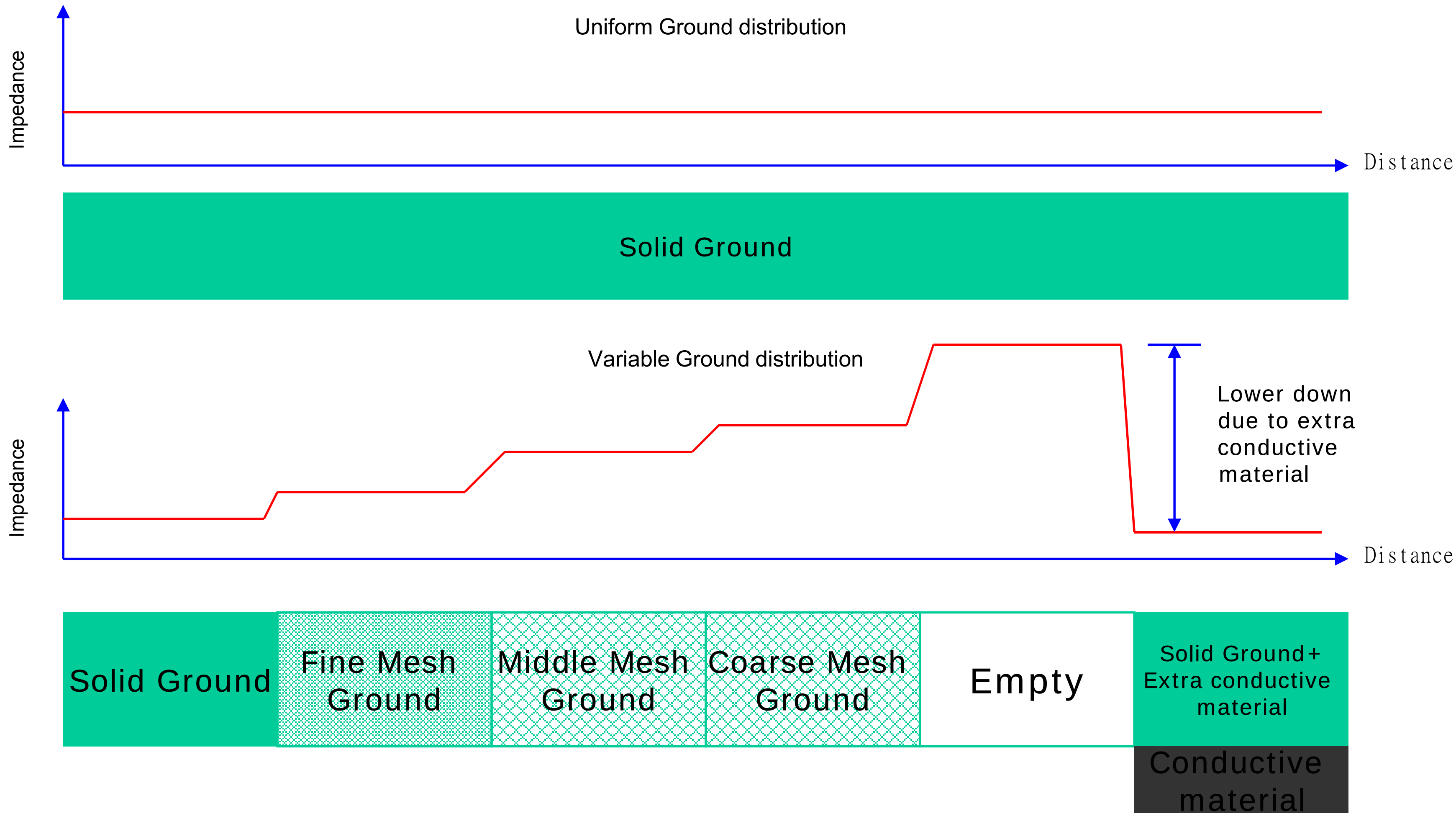


Key message:

.Under normal condition, the ground copper distribution varies in the trace axial direction.

.Under the improper design, the ground copper distribution stays constant along axial direction. It will behave like solid copper.

Impedance Variation(Ground Mesh)

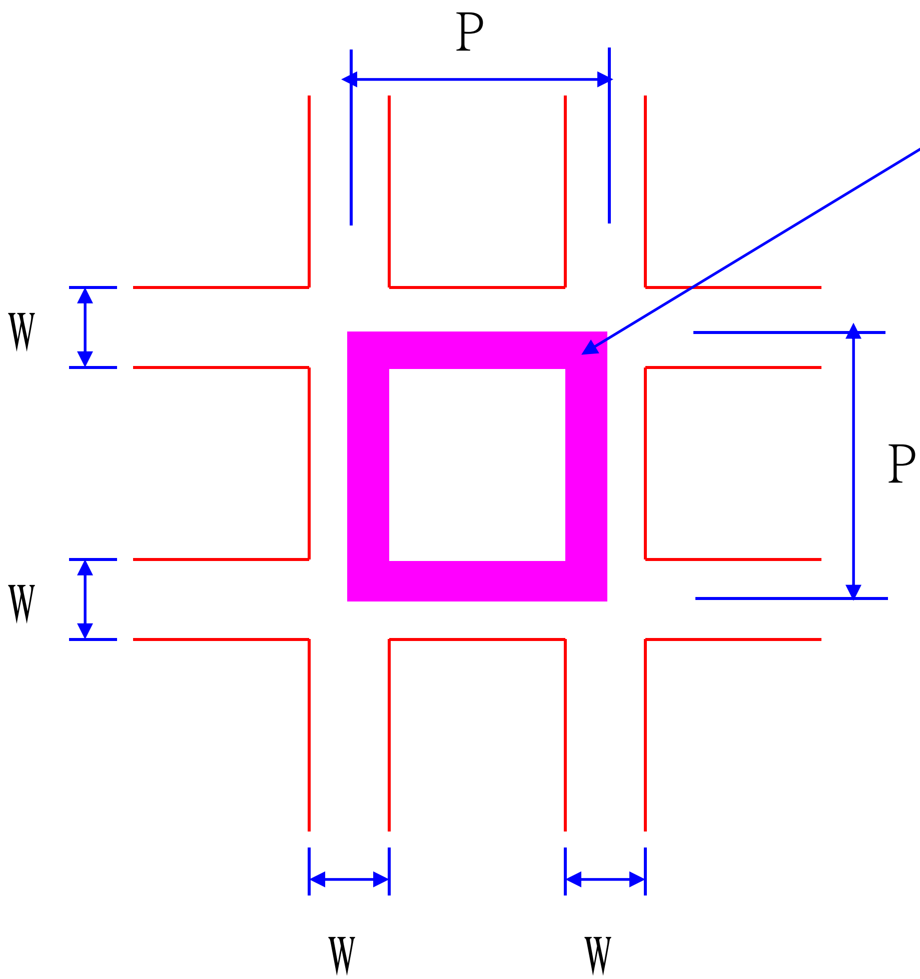


Key message:

. Impedance varies with different types of ground hatch density.

. For impedance stabilization, the ground pattern has to be uniform.Otherwise, it is very difficult to control well.

Ground Hatch Cover Rate Definition



P: Trace pitch

W:Trace width

Effective Ground Cover rate:

$$\text{Rate: } \frac{P*P-[(P-W)*(P-W)]}{P*P}$$

Solid ground:

Rate=1

Without ground:

Rate=0

Mesh ground:

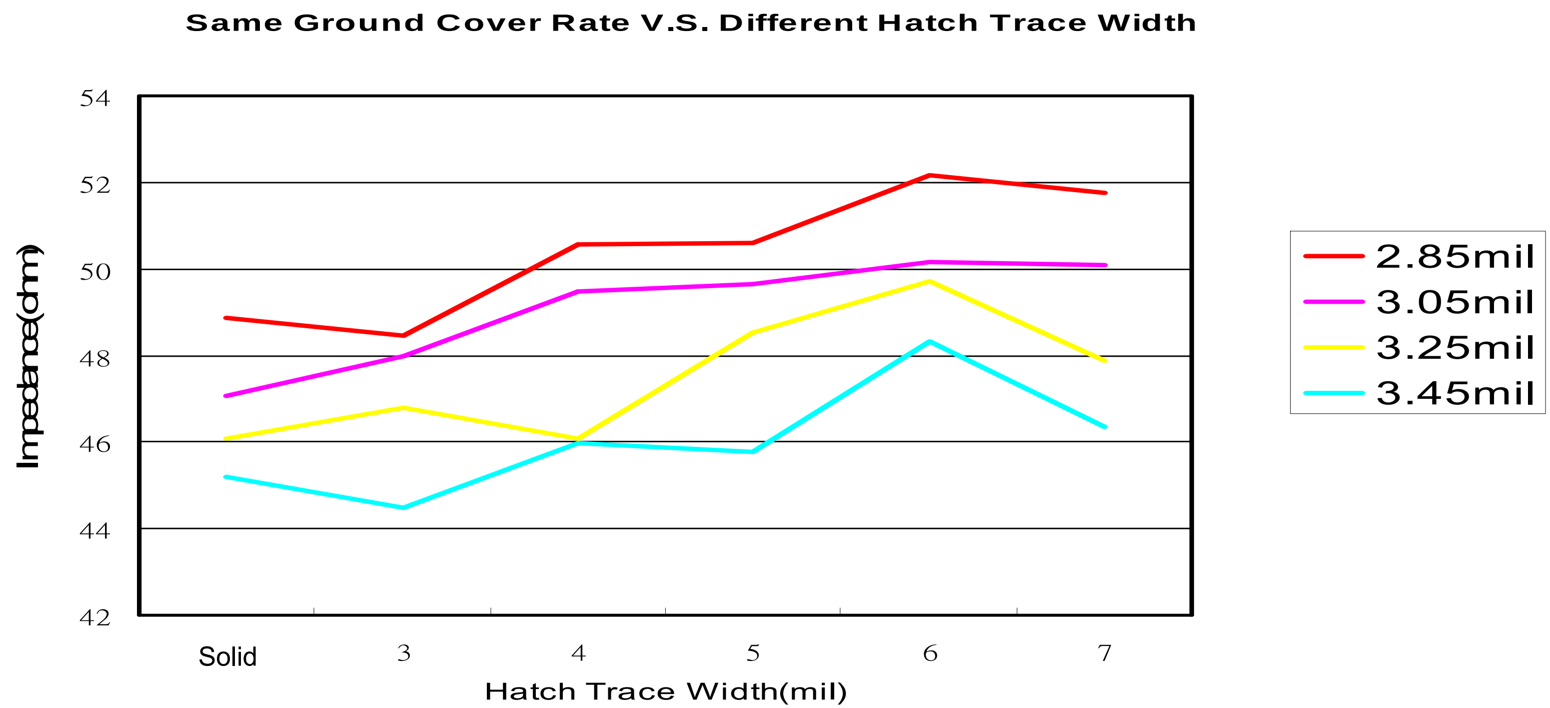
0<Rate<1

In general ,the rate is 50~60%.

If the rate is less than 50~60%,the shield performance does not work

well. But it doesn't decrease flexibility either.

Ground Hatch Cover Rate:(75%)

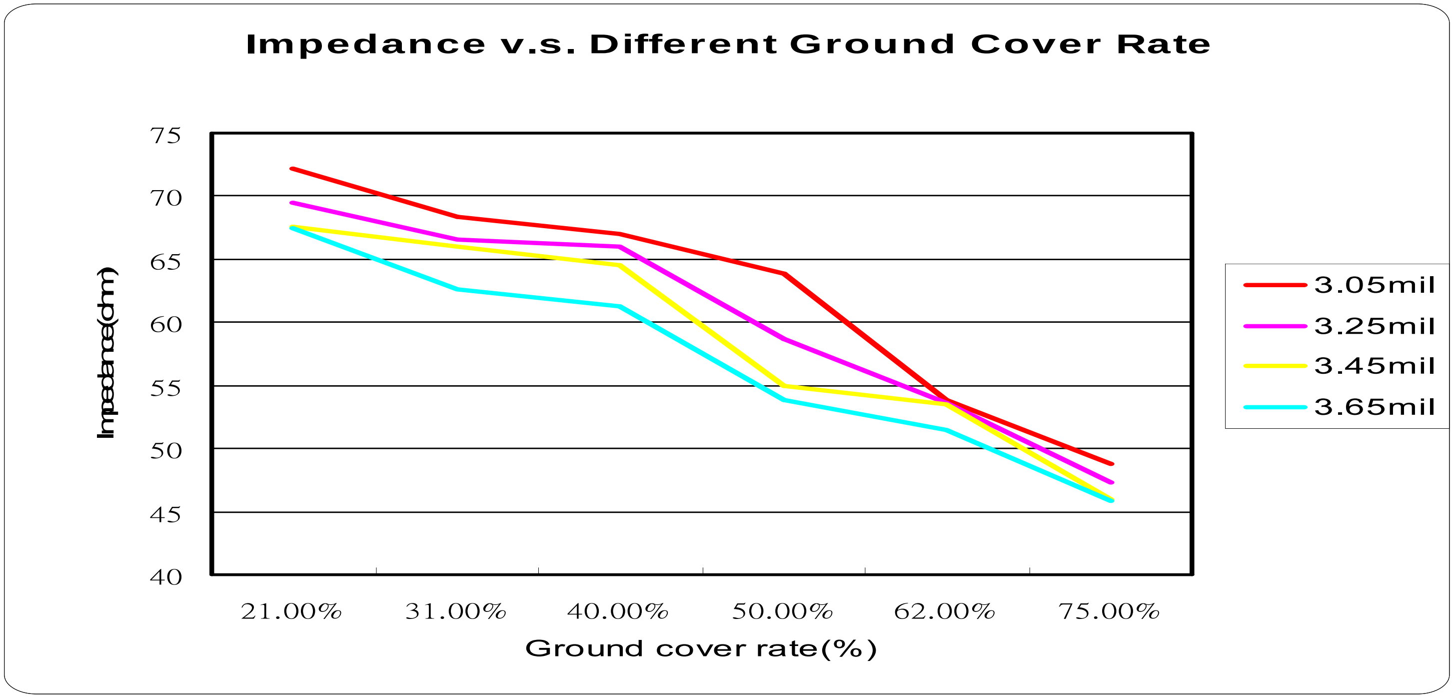


Key message:

.If the cover rate is fixed by 75%, the impedance still varies with the hatch trace width.For thicker trace,the pitch has to be adjusted to keep the same cover rate.

.Through the diagram, the impedance difference is minor.

Different Ground Hatch Cover Rate:(21%~75%)



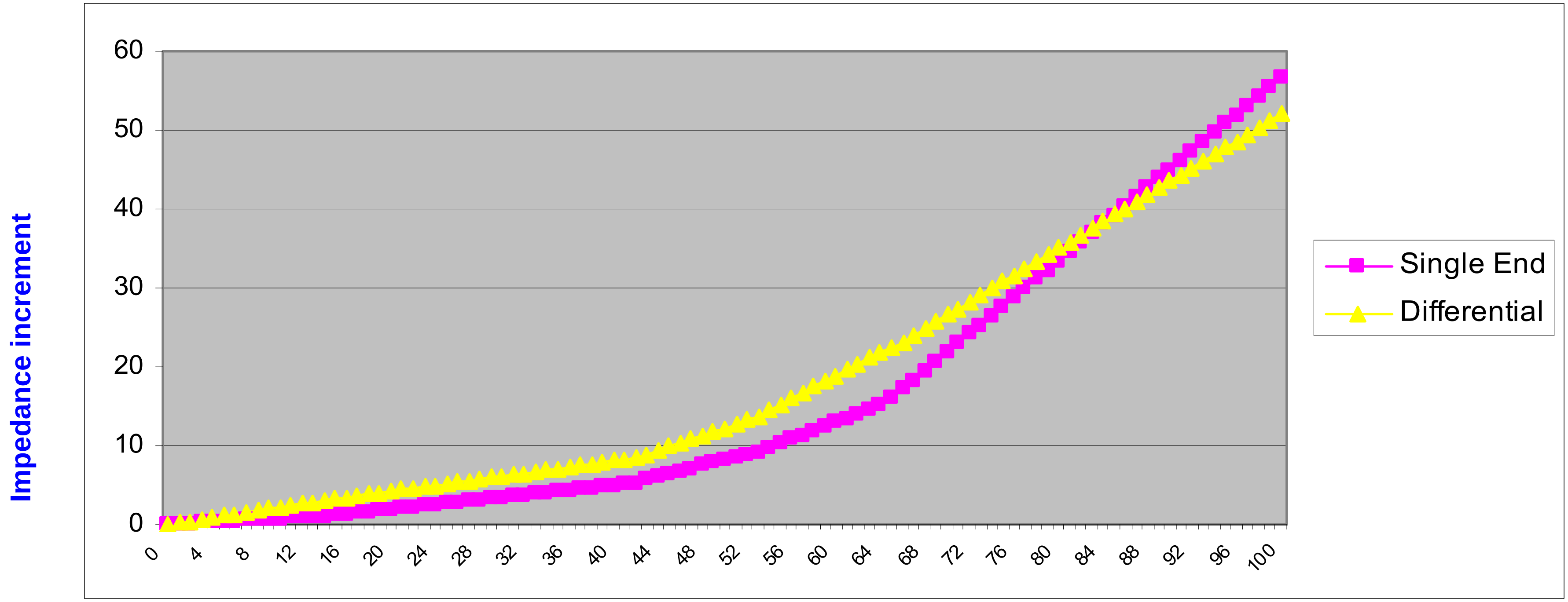
Different trace width

Rate	3.05	3.25	3.45	3.65
21.00%	72.23	69.43	67.55	67.49
31.00%	68.35	66.57	65.96	62.64
40.00%	67.055	65.99	64.52	61.25
50.00%	63.89	58.7	55.01	53.85
62.00%	53.79	53.57	53.56	51.495
75.00%	48.8	47.33	45.91	45.89

Key message:

.Through the diagram, the higher ground cover rate, the lower impedance.

Ground Copper Reduction v.s. Impedance Increment



Ground Copper reduction rate %

Key message:

.The more ground copper reduction, the more impedance increment.

Conclusions:

.There is no significant difference of impedance dropping between SF PC-1000 and SF PC-5000 even though different thickness

.Silver ink affects impedance variation more than shield film.(average 5 ohm dropping)

.The shield material on the signal side affects impedance variation than the one on the ground side.

. There is no significant difference between single side shielding and double sided shielding.

. Non-conductive material like silk screen does not cause much variation.

Thank You