PCB Material Selection for High-speed Digital Designs

Add a subtitle

Outline

- Printed Circuit Boards (PCBs) for High-speed Digital (HSD) applications
- PCB factors that limit High-speed Digital performance
- PCB material selection process and Isola product solutions
- Summary

What is High-speed Digital?

- **Digital signaling requiring use of high frequency design to preserve signal integrity**
  - Traces electrically behave as transmission lines
  - Crosstalk, attenuation, impedance mismatch are important
- **Common rule of thumb for threshold associated with trace electrical length**

$$t_d > t_r/4$$

  - t_d = line delay=delay/unit length*line length
  - tr = 20% - 80% signal rise time

isola

High-speed Digital Definitions

- In the 1990s digital signaling in electronics in 100 Mhz range was “high speed”
- Chips and computer performance have increased dramatically driving data rates to the Gbps range
- Today, data rates frequently exceed 10 Gbps and 25 Gbps signaling will be used to meet new demands

HSD Applications

- **Products utilizing high-performance PCBs include:**
  - Servers
  - Routers
  - Storage Area Networks
  - Power Amplifiers
  - Transceiver Modules
  - High Speed Data Channels (PCIe4 for example)
- **Greatest demands on PCBs are those with highest data rates & longest channel lengths**

Ever Increasing “Need for Speed”

- **Rapid growth of server, network & internet traffic drives need for higher data rates**
- **Key contributors of data demand include:**
  - Internet Consumer Applications
  - Cloud Computing and Storage
  - Virtual Servers
  - Advances in Scientific and Financial Computing

Hardware Response to Demand

- **Availability of 100G servers**
- **Migration to 100G Gen 2 using 4x25 Gbps channels**
- **IEEE-standard 802.3bj expected to be finalized mid-2014 defining 4-lane PHY for operation over PCB backplane**

Juniper T1600, Brocade MLX, Cisco CRS-3

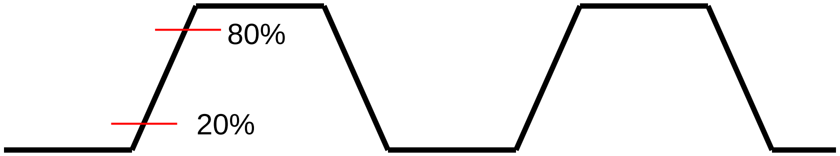
RF/Microwave vs HSD PCBs

- RF/microwave PCBs traditionally have only a few layers, in some cases just one or two
- PCBs for HSD applications often have 20 or more layers with hundreds of traces
- Materials suitable for RF/microwave may not be suitable for HSD due to processing considerations necessary for 20+ layer products

isola

RF/Microwave and HSD Similarities

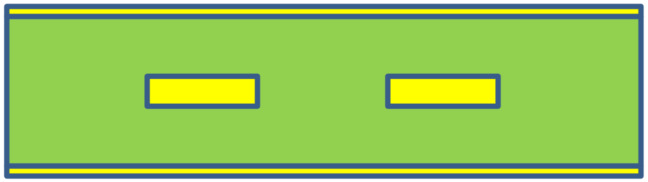
- For RF/microwave PCBs sinusoidal signals travel through PCB material and experience loss and distortion due to Dk & Df, skin effects
- For HSD PCBs trapezoidal-shaped digital waveforms travel through material experience attenuation, pulse broadening, timing errors

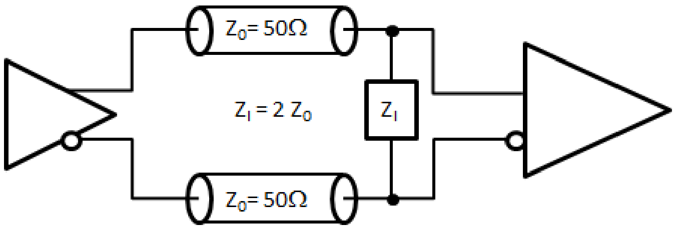


- Frequency of concern for material properties can be the same in both cases

Differential Signaling and HSD PCBs

- **Differential signaling uses a differential pair of transmission lines**
- **The transmission lines have equal and opposite polarity signals traveling on them and are tightly timed to one-another**





- **Differential signaling has several advantages**
  - Insensitive to ground connection quality between two ends of signal path
  - Data link maintains functionality with substantial attenuation in the channel
  - Supports very high data rates versus single-ended signal paths

Serial Differential Signaling is the Signaling Protocol for Modern HSD Designs

PCB Factors that limit HSD Performance

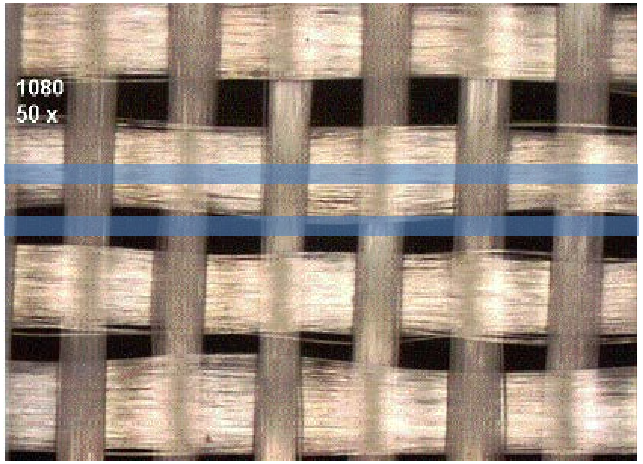
PCB Laminate Material Considerations

- **PCB laminates considered here consist of one or more plies of resin-impregnated glass cloth sandwiched between two copper foils**
- **The High-speed Digital performance of the laminate & resulting PCB depends on the quality of**
  - The resin
  - The copper foil
  - The weave of the glass

PCB Dielectric Constant

- **As a fiber/resin composite, PCB materials are inhomogeneous anisotropic dielectrics**

(1)
(2)

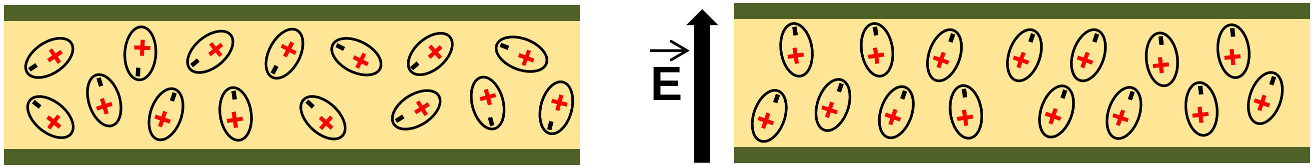


$\varepsilon_{resin} < \varepsilon_{fiber}$

- **Propagation speed within material can vary depending on location of traces**
  - Trace over fiber bundle (1) and trace over resin-rich area (2) see different effective dielectric constants
  - Differing propagation speeds creates timing skew & potential signaling errors

PCB Material Dielectric Loss

- Dielectric materials have polarized molecules that move when subjected to the electric field of a digital signal



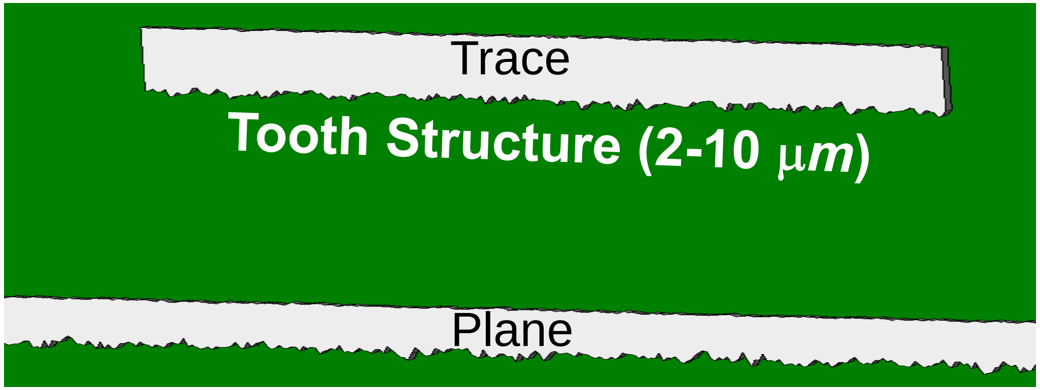
- This motion produces heat loss
- Loss results in signal attenuation that increases in direct proportion to signal frequency

PCB Material Conduction Loss

- The copper contributes to overall loss through the metal's resistive losses
- At high signal frequencies, the current in PCB copper is concentrated within a small depth near its surface (skin effect)
- Reduction in effective cross-sectional area increases the effective resistance

Conductor Surface Roughness

- Conductors on PCBs do not have perfectly smooth surfaces
- Rough copper improves peel strength of laminate
- Maximum peak-peak tooth size varies ~ 2-10 microns
- Surface roughness increases bulk copper resistance 10-50%
- Electrical impact of conductor roughness increases with increasing data rates

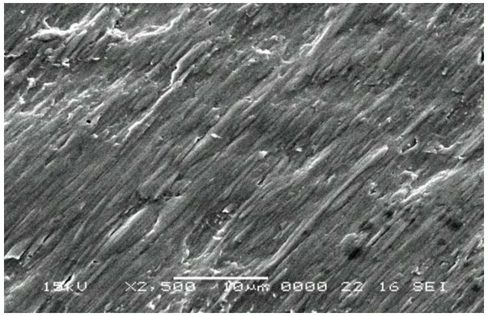


Frequency	Skin Depth (Copper)
50 Hz	9.3 mm
10 MHz	21 μm
100 MHz	6.6 μm
1 GHz	2.1 μm
10 GHz	0.66 μm

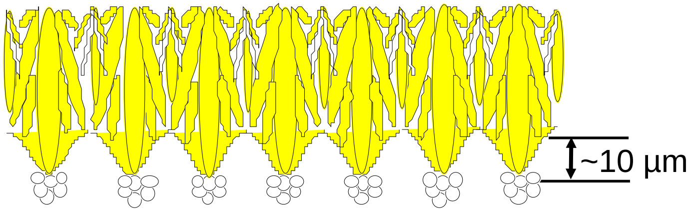
isola

Conductor Surface Roughness

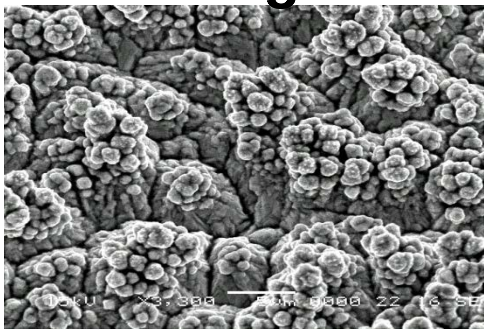
Resist side



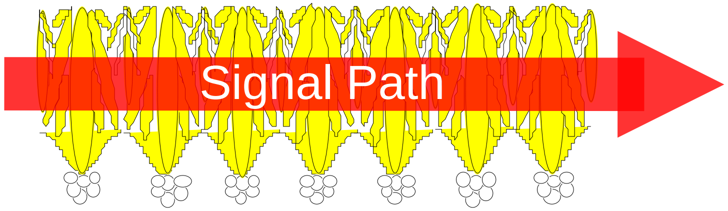
Standard foil



Bonding side

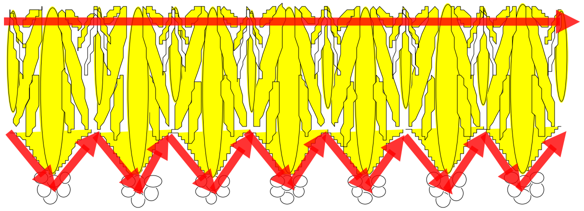


Frequency	Skin Depth
10 MHz	21 µm



The current is able to tunnel below the surface profile and through the bulk of the conductor

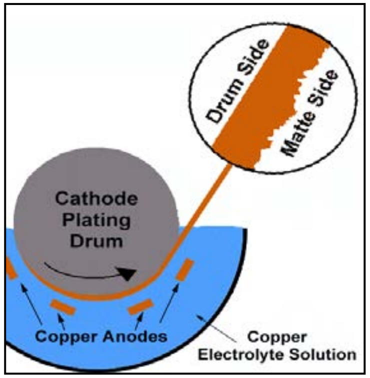
Frequency	Skin Depth
100 MHz	6.6 µm



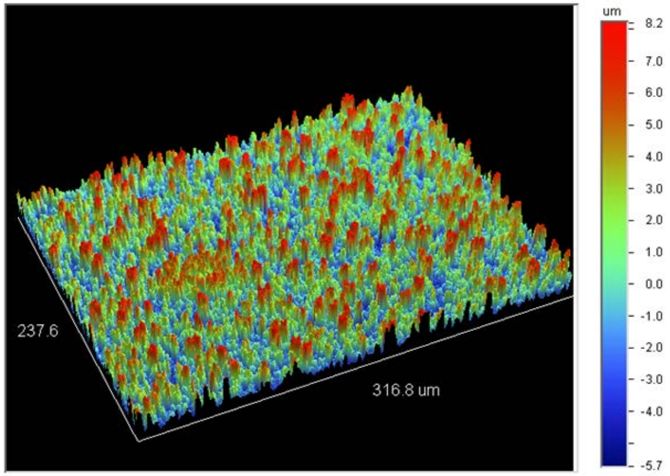
The current is forced to follow every peak and trough of the surface profile increasing path length and resistance

Copper Foil Definitions

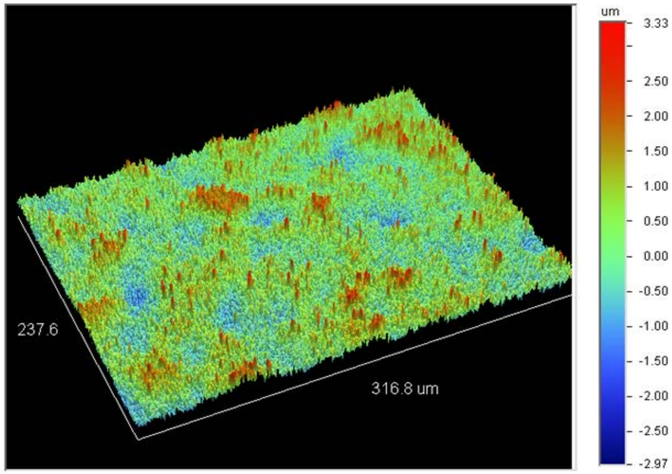
- **DSTF® (Drum Side Treated Foil):** Adhesion treatment is applied to shiny/drum side
- **RTF (Reverse Treated Foil):** Same as DSTF®
- **LP:** Low Profile Foil with Tooth 5.1-9.9 Microns
- **VLP:** Very Low Profile Foil with Tooth < 5 Microns
- **e-VLP/H-VLP:** Very Low Profile Foils
- **STD HTE (Standard Shiny Copper):** Adhesion treatment is applied to matte side



RTF: Rq=2.6 um, RF=1.85

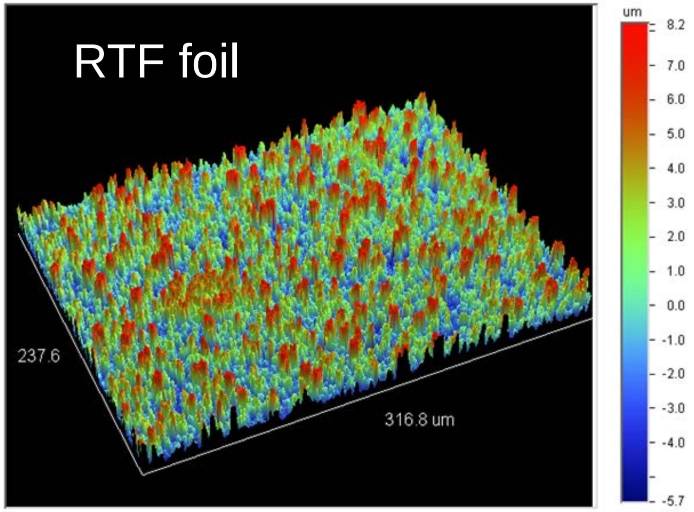


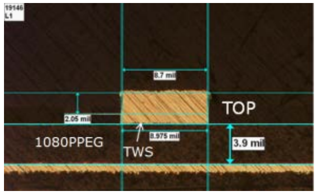
VLP: Rq=0.68 um, RF=1.3

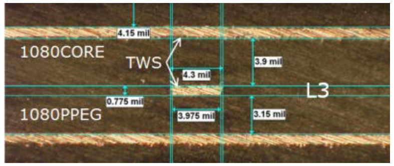


Surface Spikes Cause Increase in Capacitance

- **Multiple spikes are about 10 um from top to bottom**
- **Electric field is singular on the spikes (similar to strip edges)**
- **Consistent for 2 line types**
  - About 5% increase for MSL with one RTF surface
  - >10% increase for strip line with two RTF surfaces
- **Consistent increase in group delay and decrease in characteristic impedance over very wide frequency band**





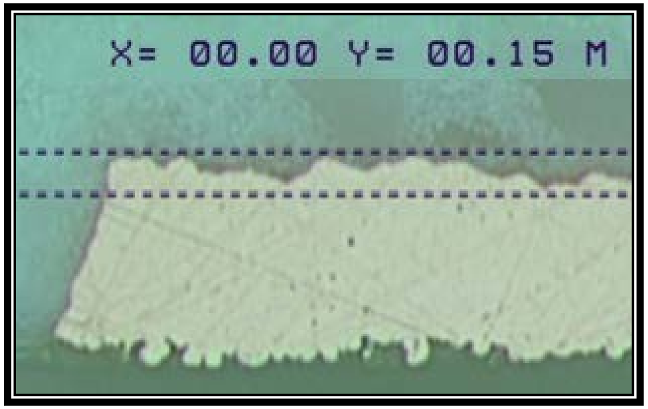


The effect was first noticed in *Deutsch, A. Huber, G.V. Kopcsay, B. J. Rubin, R. Hemedinger, D. Carey, W. Becker, T Winkel, B. Chamberlin, "Accuracy of Dielectric Constant Measurement Using the Full-Sheet-Resonance Technique IPC-T650 2.5.5.6 " p. 311-314, ., IEEE Symposium on Electrical Performance of Electronic Packaging, 2002*

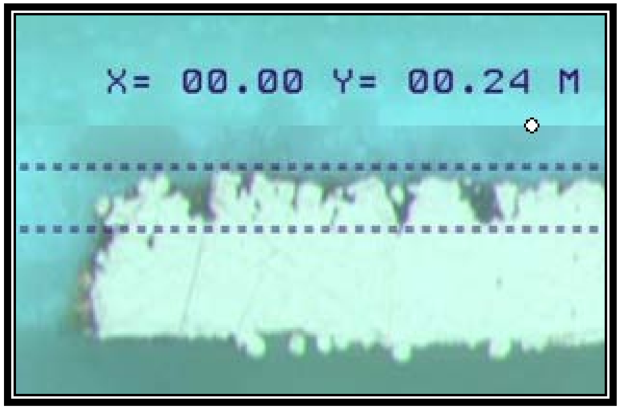
isola

Adhesion Promoter Effects

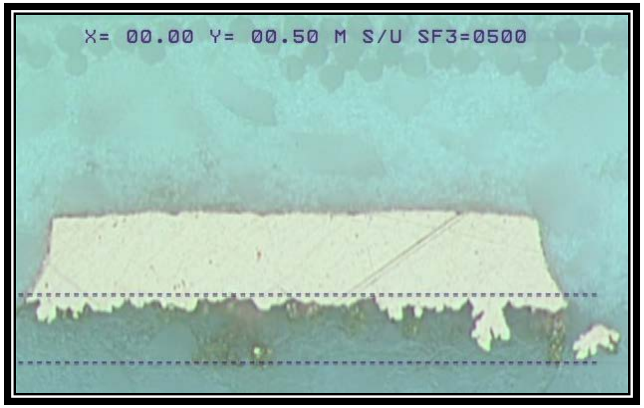
RTF Foil



RTF Foil + Adhesion Promoter



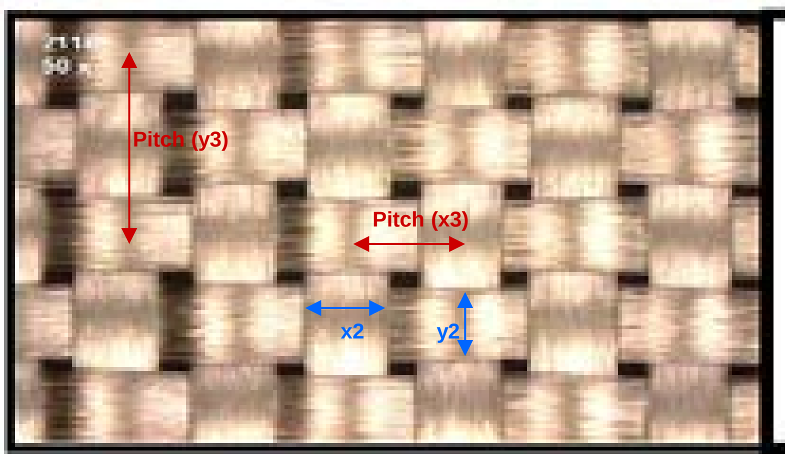
Shiny Foil

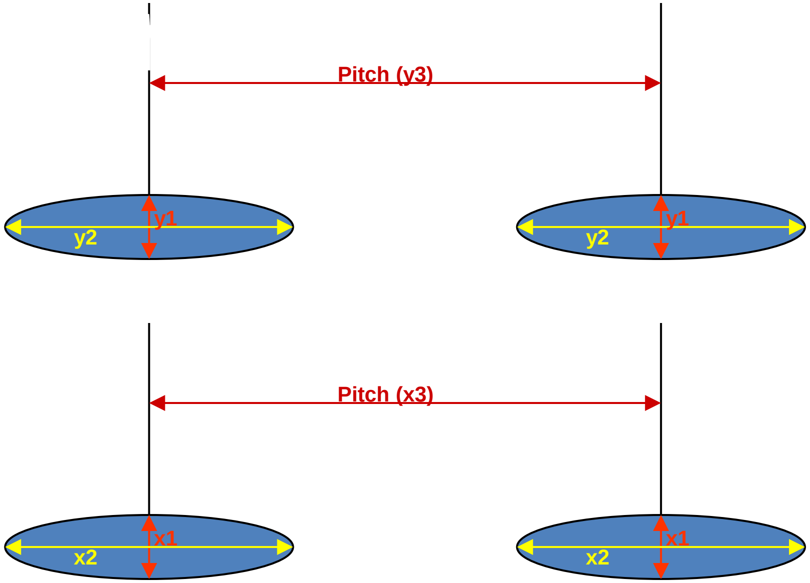


Shiny Foil + Adhesion Promoter

Courtesy of Viasystems

Glass Fabric Definitions





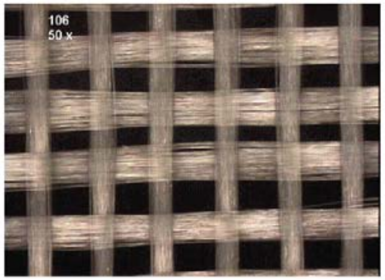
Glass type	X1	X2	X3	Y1	Y2	Y3
106	1.0	4.8	18.5	0.60	10.2	20.6
1067	0.82	8.85	14.3	0.78	12.4	13.7
1080	1.6	8.2	17	1.1	12.1	22.4
1086	1.44	10.8	16.6	1.0	14.7	17.1
2113 / 2313	2.4	10.5	17	1.0	15.3	18.2
3313	1.9	13.1	16.2	1.5	11	16.3
3070	1.7	12.7	14.8	1.7	12.6	14.2
2116	2.2	14.1	17.2	2.0	14.5	17.3
1652	2.4	15.3	17.5	2.9	15.9	18.8

All Measurrments in mls

isola

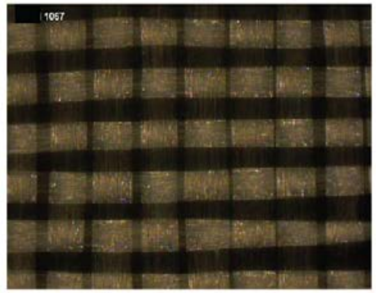
Fiber Weave Effect

- **PCBs materials are inhomogeneous dielectrics**



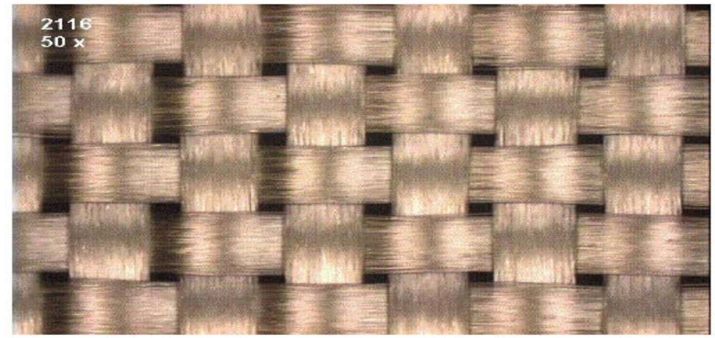
106

106
Warp and Fill count: 56 x 56 (ends/in)
Thickness: 0.0015"/0.038mm



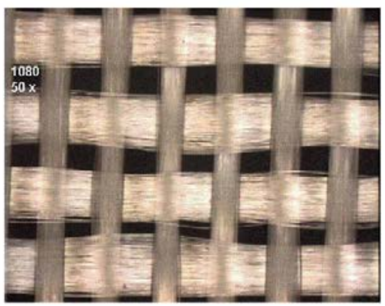
1067

1067
Warp & Fill Count: 70 x 70 (ends/in)
Thickness: 0.0013"/0.032mm



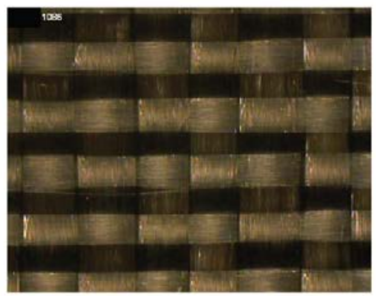
2116

2116
Warp & Fill Count: 60 x 58 (ends/in)
Thickness: 0.0038"/0.097mm



1080

1080
Warp & Fill Count: 60 x 47 (ends/in)
Thickness: 0.0025"0.064mm



1086

1086
Warp & Fill Count: 60 x 60 (ends/in)
Thickness: 0.002"/0.050mm

$$\varepsilon r_{composite} = \varepsilon r_{resin} \times Percentage\ Resin + \varepsilon r_{glass} \times (1 - Percentage\ Resin)$$

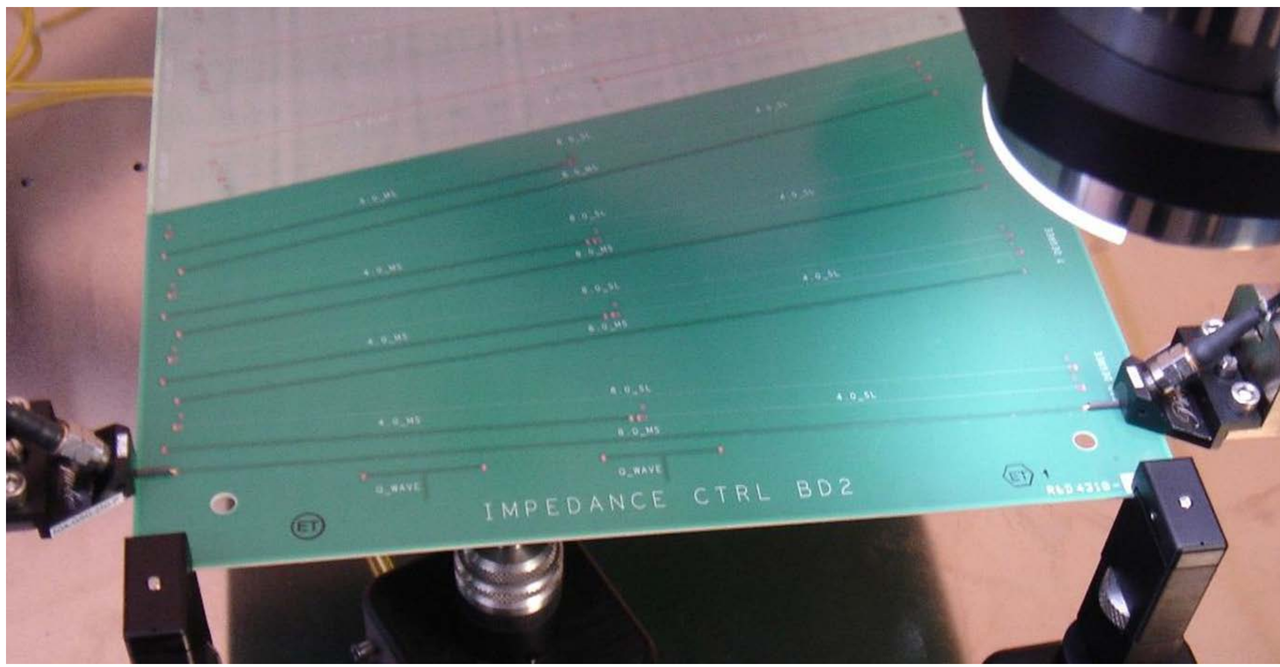
Inhomogeneous properties of PCB laminates lead to ***Fiber Weave Effects***

Test board to Measure Fiber Weave Effects

- **8 layer stack-up with 2 microstrip layers (top and bottom) and 2 strip-line layers (L3, and L6)**
- **Microstrip Top:** RTF copper foil, 1080 prepreg, no solder mask
- **Strip L3:** RTF copper foil, laminate 1080 core and prepreg
- **Strip L6:** VLP copper foil, laminate 2116 core and prepreg
- **Microstrip Bottom:** VLP copper foil, laminate 2116 prepreg

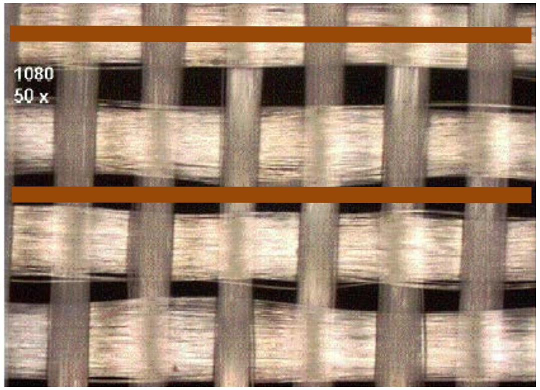
StackUp: LU=[mil], NL=8, T=37.525[mil]
1| Signal: "TOP", T=2.75, Ins="Air", Cond="CondTop"
2| Medium: T=3.9, Ins="1080PPEG", DIE_002
3| Plane: "LAYER_2", Cond="COPPER", T=0.55, Ins="1080CORE"
4| Medium: T=3.9, Ins="1080CORE", DIE_004
5| Signal: "LAYER_3", T=0.775, Ins="1080PPEG", Cond="COPPER"
6| Medium: T=3.15, Ins="1080PPEG", DIE_006
7| Plane: "LAYER_4", Cond="COPPER", T=0.55, Ins="1080CORE"
8| Medium: T=2.5, Ins="1080CORE", DIE_008
9| Plane: "LAYER_5", Cond="COPPER", T=0.55, Ins="1080CORE"
10| Medium: T=5.05, Ins="2116PPEG", DIE_010
11| Signal: "LAYER_6", T=0.775, Ins="2116PPEG", Cond="COPPER"
12| Medium: T=4.9, Ins="2116CORE", DIE_012
13| Plane: "LAYER_7", Cond="COPPER", T=0.55, Ins="2116CORE"
14| Medium: T=5.3, Ins="2116PPEG", DIE_014
15| Signal: "BOTTOM", T=2.325, Ins="Air", Cond="COPPER"

Test structures: 4-inch and 8-inch line segment with transitions to probe pads

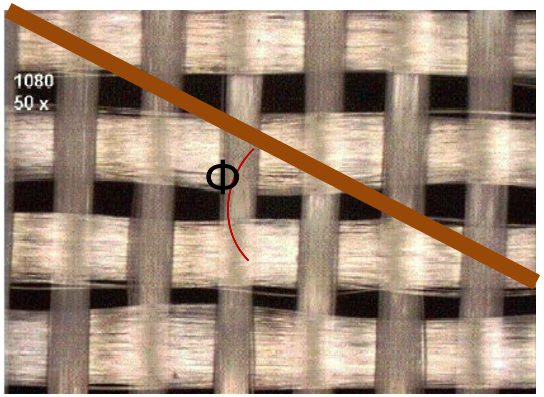


Two Main Fiber Weave Effects

1. **Effects due to location of trace with respect to fiber weave bundles**

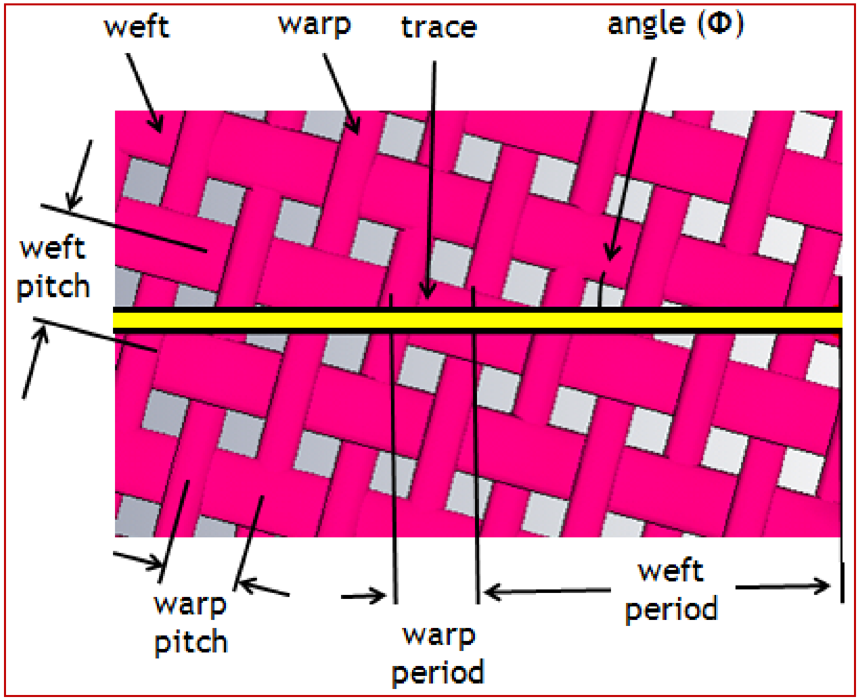


2. **Effects due to periodic loading of trace by fiber weave bundles – resonance**

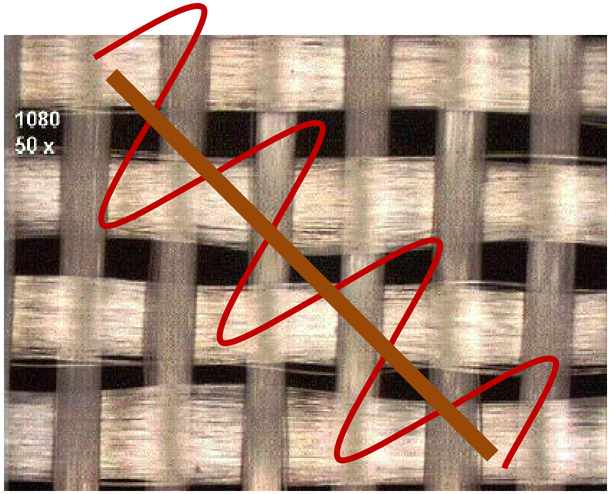


Analytical Formulation of FWE

Arbitrary Angle: Φ = 0° > 45°



- Challenge is to find/define the spatial period
- Separate the Weft and Warp loading
- Warp loading is in pitch scale → high frequencies → neglected
- Weft loading occurs in a larger scale → lower frequencies → key role

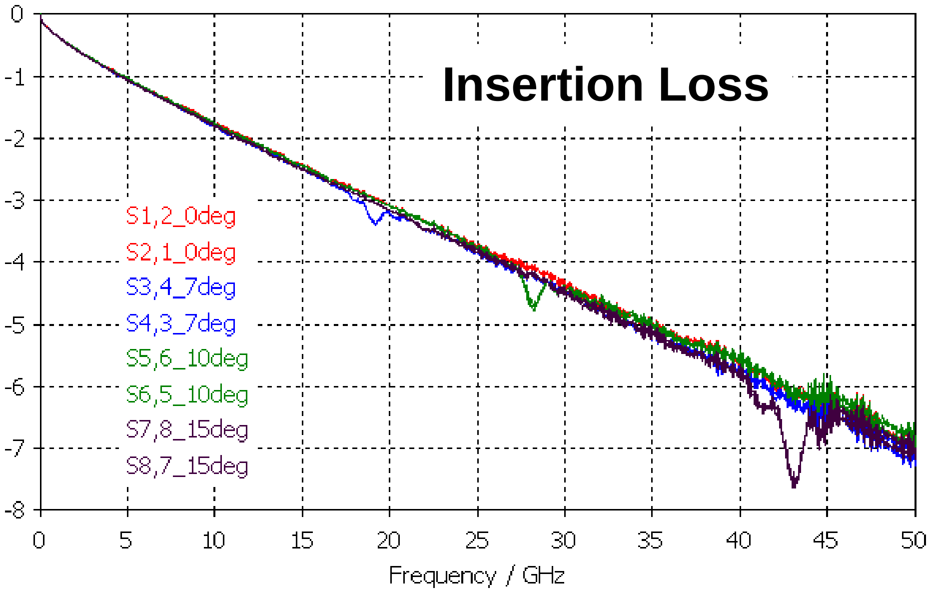


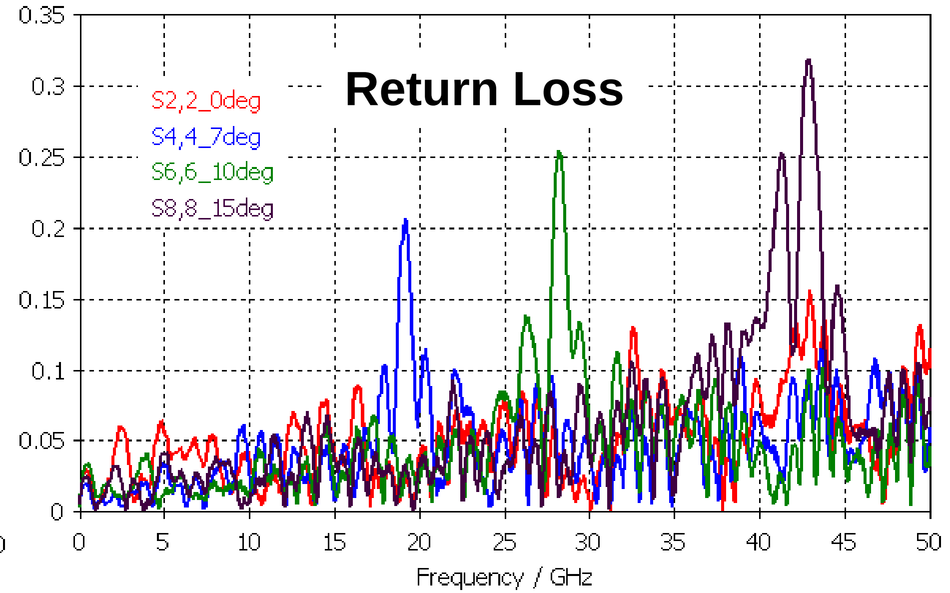
Weft spatial period is obtained from trigonometric expressions:

$$weft\ period = \sqrt{pitch^2\left(\frac{1}{[\tan(\Phi)]^2}+1\right)} \Rightarrow f_{res} = \frac{c}{2\times\sqrt{\varepsilon_{eff}\, pitch^2\left(\frac{1}{[\tan(\Phi)]^2}+1\right)}}$$

Fiber Weave Effect: Angle (1080)

- **Insertion and Return Loss, 4-inch traces**





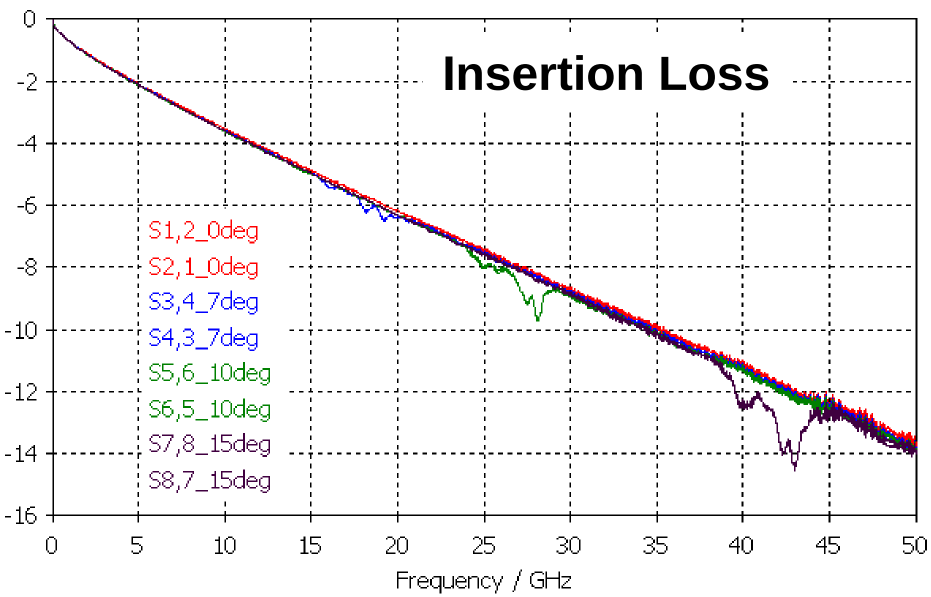
- **Excellent agreement with analytical equation**
- **Strength of resonance increases with angle**

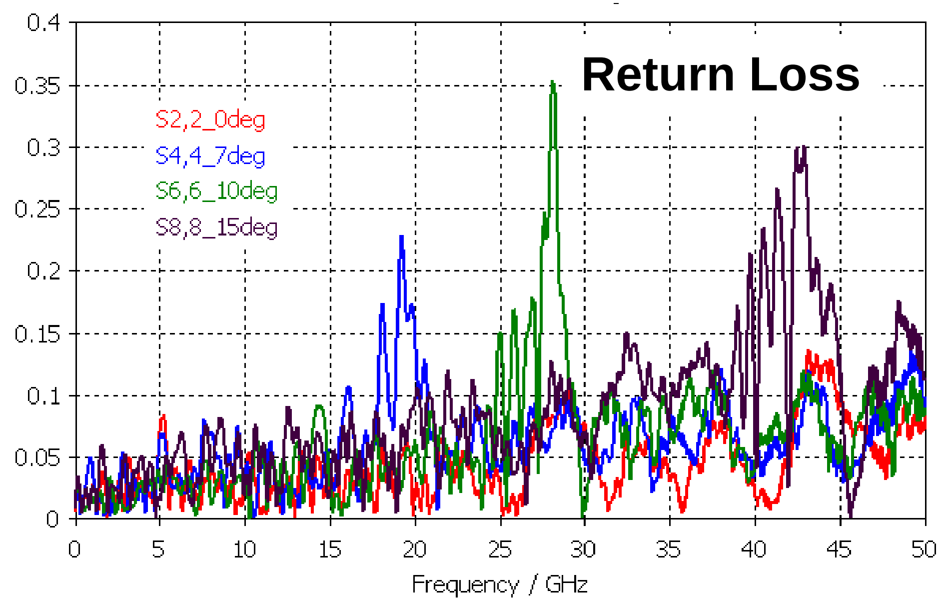
Angle [Deg]	Equation 2 (GHz)	Measured (GHz)
7	19.6	19.17
10	26.76	28.2
15	41.39	42.86

isola

Fiber Weave Effect: Angle (1080)

- Insertion and Return Loss, 8-inch traces

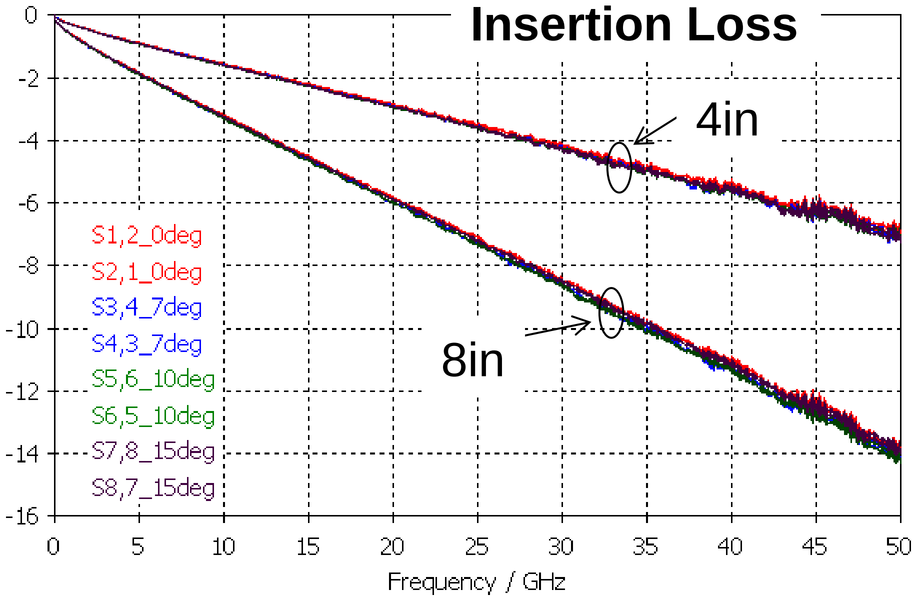


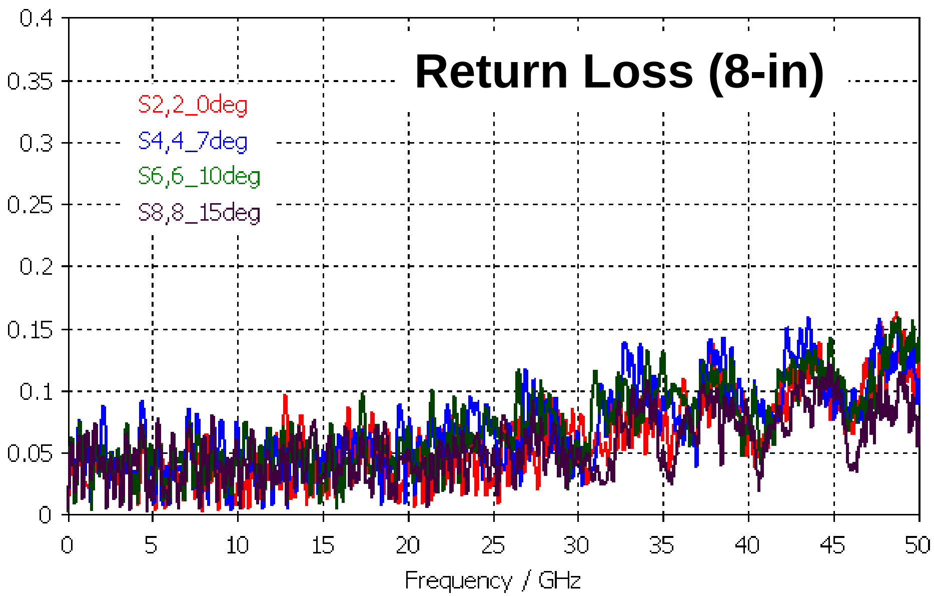


- Excellent agreement with analytical equation
- Resonances “spread” over frequency due to fiber weave imperfections

Fiber Weave Effect: Angle (2116)

- **Insertion and Return Loss, 4- and 8-inch traces**

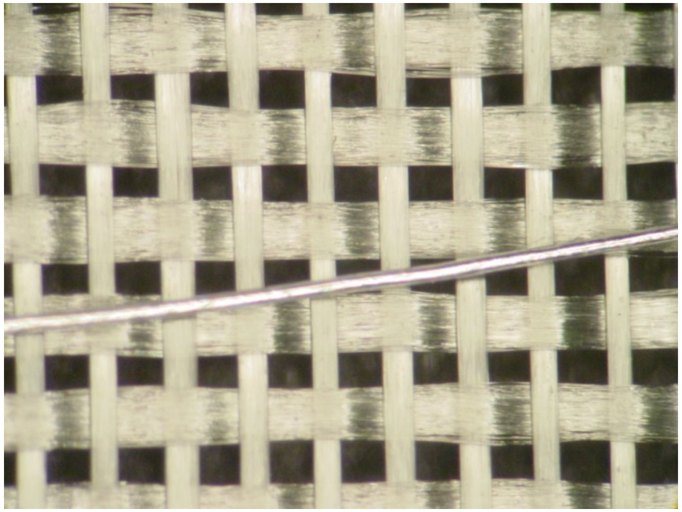




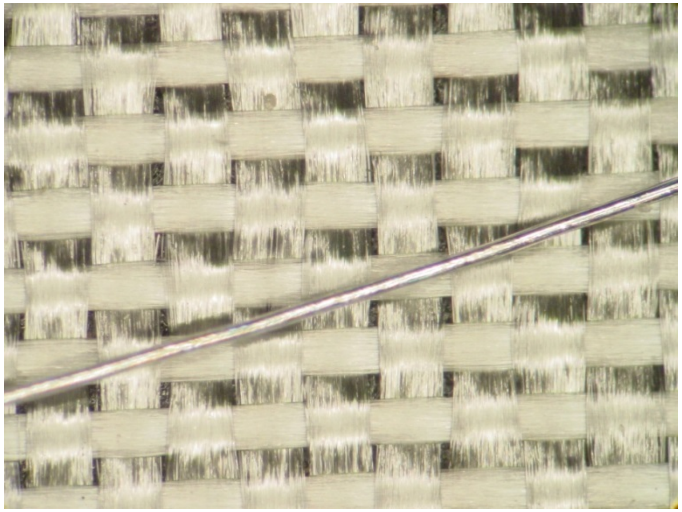
- **No resonances were detected from measurement**
- **This is due to the denser and more homogeneous material**

Ways to Mitigate Fiber Weave Effects

- **Use more uniform glass such as 2116 or 3113 shown**
  - These styles have more uniform distributions of glass across the entire surface greatly reducing impedance variations
- **Use as wide a trace as practical for layout**
- **Route traces at an angle with respect to fiber warp/fill**
- **Use zig-zag routing**
- **Select materials having resin and fiber dielectric constants as close as possible to one another**



1080



3113

PCB Material Selection Process and Isola Product Solutions

PCB Material Selection

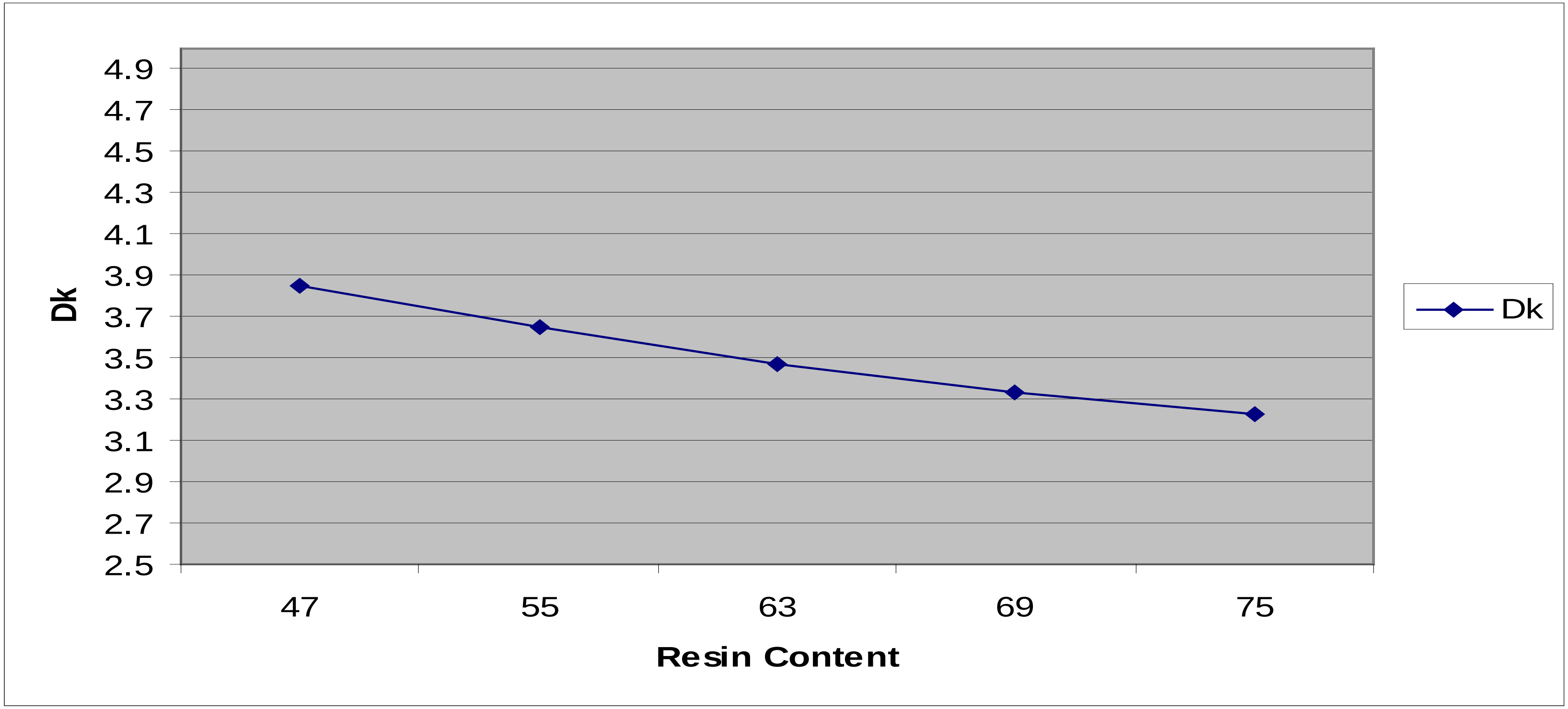
Selecting the Right Material

- Laminate material selection can not be condensed into a single page chart for easy selection
- High performance laminate material suppliers have a much better understanding of material performance
- Cost-to-performance evaluations must still be done by the system design team to ensure the lowest cost material that will do the job is selected

Key Laminate Performance Indicators

- Dielectric Constant (Dk)
  - Matching material performance numbers is important. A small difference in this value between materials can impact impedance, line widths, and thus losses significant
  - Consider construction options that allow you to find a drop in and match impedance
- Dissipation Factor (Df)
  - Values vary dramatically by resin content, resin type, frequency, and test method
  - It is important to thoroughly understand the methods used to derive the numbers
  - Compare Df to Df at equal test conditions and resin contents when looking at a suppliers data sheets
- Copper Type
  - Low profile copper provides better results than standard profile copper
  - RTF or DSTF type foils offer significant improvements in loss characteristics. VLP foils are being used as well for better impedance

FR408HR Dk vs Resin %



isola

Summary

- **Printed Circuit Boards (PCBs) for High-speed Digital (HSD) applications**
- **PCB factors that limit High-speed Digital performance**
- **PCB material selection process and Isola product solutions**