ALTIUMLIVE 2019:
ANNUAL PCB DESIGN SUMMIT

San Diego & Frankfurt,
October 9-11 & October 21-23

Michael R. Creeden CID+ MIT
Insulectro - Tech. Dir. Design Education
Next Generation Materials
Mike Creeden, MIT CID+

- Insulectro – Technical Director Design Education
- Eptac: IPC-CID/CID+ MIT (Master IPC Trainer)
- Primary Contributor of the IPC-CID+ Curriculum
- IPC Designers Council – Executive Board Member
- Founder of San Diego PCB, Design, LLC
- PCB Designer 43 Years: “I Love PCB Design and I Love Altium”
Today’s Designer must meet 3 Perspectives for Success

Design For:
- **DFSolvability**
- **DFPerformance**
- **DFManufacturability**

1. **Layout Solvability**
   - A skill set to solve the place and route of all parts and connections often with complex High Density Interconnect (HDI)
   - ... and master this on your CAD Tool

2. **Making Revision One Work**
   - DFX all considerations producing high yield and lower cost

3. **Performance**
   - Signal Integrity/EMC
   - Power Delivery
   - Thermal

THE RESULT
- Maximum placement and routing density, optimum electrical performance and efficient, defect-free manufacturing
The Road to 5G

- **Ubiquitous Connectivity**
- **End-user Data Rates & Connected Devices**
  - 2Kbps (1981)
  - 64Kbps (1992)
  - 384 Kbps (2001)
  - 100 Mbps (2009)
  - > 10 Gbps (2020)

- **Lower Latency**
  - 2009: 2hrs
  - 2020: 6min
  - 2035: 3.6s

- **Better Battery Life**
  - 2009: 10x-100x
  - 2020: 5x
  - 2035: 10x

- **Connected Devices**
  - 2009: 10x-100x
  - 2020: 5x
  - 2035: 10x

**End-user Data Rates**

- **Basic voice**
  - Analog protocols
- **Voice services**
  - Digital standards
- **First mobile**
  - Broadband IP protocols
- **True mobile**
  - Broadband Unified standard LTE
- **Tactile internet**
  - Service aware devices
  - Fiber-like speeds

**Ubiquitous Connectivity**

-$12T$ Global Economic Activity by 2035

Dow DuPont CONFIDENTIAL - Do not share without permission
The 5G ERA is here…

5G is the fifth generation of wireless technology and will be a game changer!

We will know it as one of the fastest, most robust technologies the world has ever seen. Enabling quicker downloads, outstanding network reliability and have a significant impact on how we live, work and play.

The connectivity benefits of 5G will make businesses more efficient and give consumers access to more information faster than ever before.

All Industry segments will rely on 5G:
• Super-connected EV and autonomous cars,
• Smart communities
• Industrial IoT
• MIL-AERO
• Medical & Pharma transportation
• AI - Artificial Intelligence
US Radio Spectrum Frequency Allocations
Microwaves are a form of electromagnetic radiation with wavelengths ranging from about one meter to one millimeter; with frequencies between 300 MHz (1 m) and 300 GHz (1 mm) They follow line of sight and are limited with effective ranges of 1 kilometer to 40 miles.

**mmWave** frequencies between 30 GHz (1 cm) and 300 GHz (1 mm)

A **wavelength** is the **spatial period** of a periodic wave—the distance over which the wave's shape repeats.

<table>
<thead>
<tr>
<th>International Telecom Union (ITU) designation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Frequency Name</strong></td>
</tr>
<tr>
<td>Ultra High Freq.</td>
</tr>
<tr>
<td>Super High Freq.</td>
</tr>
<tr>
<td>Extremely High Freq.</td>
</tr>
<tr>
<td>Far Infrared Freq.</td>
</tr>
<tr>
<td>Tremendously High Freq.</td>
</tr>
</tbody>
</table>
The **mmWave band** was considered unsuitable for mobile communications, mainly due to high loss and propagation issues. These concerns are being solved with the development of **phased arrays** and **beam-steering or beam-forming antennas**.

A phased-array antenna is composed of multiple RF elements. Each element is connected to a phase shifter, which forms the beam that steers the antenna via constructive or destructive interference.
RF signals while generated on our boards are taken off by antenna or connector and used between two boards over the air.

High Speed Digital (HSD) signals communicate between devices on a board and should stay on the board.

HSD/RF frequency of operation and data bandwidths: (HSD/RF signals on the Board) and (RF signals over the air) are handled with different parameters and methods.
5G - RF technology is designed to work with (HSD) High-Speed-Digital technologies such as PCIe 5.x and 400G Ethernet to bring us into the ever increasing **HIGH-SPEED** market.

<table>
<thead>
<tr>
<th>Raw Bit Rate/Lane</th>
<th>Link BW</th>
<th>BW/Lane</th>
<th>Total x16 Bi-Directional Bandwidth</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCIe 1.x</td>
<td>2.5GT/s</td>
<td>2Gb/s</td>
<td>250 MB/s</td>
</tr>
<tr>
<td>PCIe 2.x</td>
<td>5.0GT/s</td>
<td>4Gb/s</td>
<td>500 MB/s</td>
</tr>
<tr>
<td>PCIe 3.x</td>
<td>8.0GT/s</td>
<td>8Gb/s</td>
<td>~1GB/s</td>
</tr>
<tr>
<td>PCIe 4.x</td>
<td>16.0GT/s</td>
<td>16Gb/s</td>
<td>~2GB/s</td>
</tr>
<tr>
<td>PCIe 5.x</td>
<td>32.0GT/s</td>
<td>32 Gb/s</td>
<td>~4GB/s</td>
</tr>
</tbody>
</table>

HSD 32Gbit/s per lane, PCIe 5.0 technology delivers a 64GBytes/s over 16 lanes in each direction for a total link bandwidth of ~128GB/s. More than enough to meet the needs of 400G Ethernet.

**Isola’s Tachyon®100G** is rated for: 100Ghz, Dk 3.02 Df, .0017, Spread Weave, Tg 200°, Td 360°, processes well with **Astra®** High Freq RF epoxy (improving long term reliability)

- **GHz:** is a operational switching frequency
- **Gbit/s per lane:** is an amount of data (one or multiple lanes, bi-directional, thus doubled)
Electrical and Physical Performance

- Low Dielectric Constant (Dk)
  - Relative Permittivity
  - Low Latency – Low relative permittivity decreases latency due to substrate

- Low Dissipation Factor (Df)
  - Loss Tangent - Low loss tangent improves insertion loss and enhances efficiency

- Spread Weave – Ensure consistent impedance for a diff-pair

- Low Profile CU & good adhesion

- High Tg - Resin movement for multi-layer & lamination

- High Td - Decomposition for high temperature mfg. & usage

- Multiple Copper & Laminate Thickness - for performance and mfg.

- Cost and availability from a fabricator that knows how to process it
PCB & Flex Copper Types

(ED) Electro Deposited Foil
For Rigid PCB

(ED) Electro Deposited Foil
For Rigid PCB

(RA) Rolled Annealed Foil
For Flex

Plated
At PCB Fabricator

Electrodeposited copper

Wrought copper

Electroplated copper
Copper Foil Surface Profiles (Rz)

- When the roughness of the copper surface becomes close to the wavelength of the signal, the loss increases.
- This begins to be a big factor at frequencies of 10 GHz and higher.
- The benefits of a low loss laminate can be undone with rough copper.

Skin Effect

Cu Trace Cross Sections
(dark areas show highest field density)

Low frequency

High frequency

Standard Foil

STD HTE

DSTF/RTF

DSTF/RTF

HVLP/ e-VLP

Standard

VI P2
Copper Foil Surface Profiles (Rz)

Acronyms: Copper Surface Profiles
• ED – Standard Shiny Copper
• HP – High Performance Foil with extra tooth for high peels*
• HTE– High Tensile of Elongation, Standard Shiny Copper
• DSTF ® – Drum Side Treated Foil*
• RTF – Reverse Treated Foil
• VLP – Very Low Profile
• e-VLP – Extra(?) Very Low Profile*
• H-VLP – H (Hyper) Very Low Profile*
• VLP-2 – Isola’s designation for very low profile copper*
• VLP-1 – For Ultra Low loss applications*
• RA – Rolled annealed

* Not IPC Designations
Copper surface roughness is a balance between:
- Lower Loss
- Peal Strength.

Copper foil surfaces from Standard to VLP2

Typically, HVLP or VLP2 coppers come at a cost premium

Rougher = better adhesion
Smotherer = lower loss

Yes, these are at the same magnification.
Isola’s Product Ladder

Rigid Laminates:

All types of circuit technologies:

**DF-Solvability:**
- HDI, uTraces, uVias

**DF-Performance:**
- SI, EMI, PDN, Thermal

**DF-Manufacturability:**
- CTE, Plating, CU surface, Process, Reliability

---

**“Materials Matter” for Advanced Routing**

<table>
<thead>
<tr>
<th>Product</th>
<th>Tg by TMA</th>
<th>Td</th>
<th>Dk</th>
<th>DF</th>
<th>VLP Foil</th>
<th>PMM Sensitive applications</th>
<th>IPC Slash Sheets, Comments and Recommended Bit Rate/Frequency range</th>
<th>Number of laminates cycles</th>
<th>Compatible with for Hybrid Builds</th>
<th>Replaces These Products</th>
</tr>
</thead>
<tbody>
<tr>
<td>185HR</td>
<td>180</td>
<td>340</td>
<td>4.01</td>
<td>0.02</td>
<td>N/A</td>
<td>N</td>
<td>Low cost Lead Free solder compatible FR4 PCB. 2 to 3 GHz</td>
<td>3 to 4</td>
<td>Tera MT40, Tachyon 100G, Astra MT77</td>
<td>Panasonic R-1755V and R-1855V</td>
</tr>
<tr>
<td>370HR</td>
<td>180</td>
<td>340</td>
<td>4.04</td>
<td>0.021</td>
<td>N/A</td>
<td>N</td>
<td>Legacy High rel and lead free compatible FR4. 2 to 3 GHz max</td>
<td>3 to 4</td>
<td>Tera MT40, Tachyon 100G, Astra MT77</td>
<td>Panasonic R-1755V and R-1855V</td>
</tr>
<tr>
<td>FR408HR</td>
<td>190</td>
<td>360</td>
<td>3.68</td>
<td>0.0922</td>
<td>Available</td>
<td>N</td>
<td>Multifunctional low resin up to 12 GHz</td>
<td>3 to 4</td>
<td>Tera MT40, Tachyon 100G, Astra MT77</td>
<td>Nexcel N1000-13 and 13EP</td>
</tr>
<tr>
<td>i-Speed</td>
<td>180</td>
<td>360</td>
<td>3.64</td>
<td>0.006</td>
<td>Standard</td>
<td>N</td>
<td>Best Signal performance at this cost. Up to 20 GHz</td>
<td>4 to 5</td>
<td>TERA HR, 370HR, 408HR, i-Speed, Tachyon 100G, Astra MT77</td>
<td>Panasonic Megtron 4 R-5725 and R-5620 EM-888 and 888K</td>
</tr>
<tr>
<td>i-Tera MT40</td>
<td>200</td>
<td>360</td>
<td>3.45</td>
<td>0.0031</td>
<td>Available</td>
<td>N</td>
<td>Very good signal and thermal performance. Up to 60 GHz</td>
<td>10</td>
<td>168HR, 370HR, 408HR, i-Speed, Tachyon 100G, Astra MT77</td>
<td>Panasonic R-5775K and R-5670K Megtron 6, 6G, 6N</td>
</tr>
<tr>
<td>i-Tera MT40 (RF/MW)</td>
<td>200</td>
<td>360</td>
<td>3.39/3.45/3.66/3.76</td>
<td>0.0029/0.0035</td>
<td>Available</td>
<td>Yes, with VLP-2 foil</td>
<td>Same as i-Tera MT40, but Dk tuned for RF applications. Up to 77 GHz</td>
<td>10</td>
<td>Speed, i-Tera MT40, Tachyon 100G, Astra MT77</td>
<td>Rogers RO-4350B, RO-4003C, RO-4534, RO-3050, RO-4835</td>
</tr>
<tr>
<td>TerraGreen*</td>
<td>200</td>
<td>390</td>
<td>3.44</td>
<td>0.0039</td>
<td>Available</td>
<td>N</td>
<td>IPC-401/17 Halogen Free version of i-Tera MT40</td>
<td>Up to 60 GHz</td>
<td>6</td>
<td>IS-300MD</td>
</tr>
<tr>
<td>TerraGreen* (RF/MW)</td>
<td>200</td>
<td>390</td>
<td>3.45</td>
<td>0.0032</td>
<td>Available</td>
<td>Yes, with VLP-2 foil</td>
<td>IPC-401/17 Halogen Free for RF</td>
<td>Up to 77 GHz</td>
<td>6</td>
<td>IS-300MD</td>
</tr>
<tr>
<td>IS300MD</td>
<td>190</td>
<td>390</td>
<td>3.06</td>
<td>0.0033</td>
<td>Available</td>
<td>N</td>
<td>Low loss halogen free for thin build-up on mobile devices. Up to 60 GHz</td>
<td>6</td>
<td>TerraGreen</td>
<td>EM-890 and 890K</td>
</tr>
<tr>
<td>IS680</td>
<td>200</td>
<td>360</td>
<td>2.80/3.45</td>
<td>0.0025/0.0032</td>
<td>Available</td>
<td>N</td>
<td>IPC-401/17 Low cost PTFE alternative for double sided RF</td>
<td>N/A</td>
<td>N/A double sided only</td>
<td>Rogers RO-3003 (double sided only)</td>
</tr>
<tr>
<td>IS680 AG</td>
<td>200</td>
<td>360</td>
<td>3.00/3.38/3.45/3.48</td>
<td>0.0020/0.0029</td>
<td>Standard</td>
<td>Yes</td>
<td>IPC-401/17 Low cost double sided material for PWB sensitive RF applications. Up to 77 GHz</td>
<td>N/A</td>
<td>N/A double sided only</td>
<td>Rogers RO-3003 (double sided only)</td>
</tr>
<tr>
<td>Tachyon 100G</td>
<td>200</td>
<td>360</td>
<td>3.02</td>
<td>0.0021</td>
<td>Standard</td>
<td>N</td>
<td>IPC-401/17 Ultra low loss and low Dk for high speed applications. Up to 100 GHz</td>
<td>10</td>
<td>168HR, 370HR, 408HR, i-Speed, i-Tera MT40, Tachyon 100G, Astra MT77</td>
<td>Panasonic Meg 7, 7N R-5785, R-5680</td>
</tr>
<tr>
<td>Astra MT77</td>
<td>200</td>
<td>360</td>
<td>3</td>
<td>0.0017</td>
<td>Standard</td>
<td>Yes</td>
<td>Low Dk alternative for RF multilayer applications. Up to 100 GHz</td>
<td>10</td>
<td>168HR, 370HR, 408HR, i-Speed, i-Tera MT40, Tachyon 100G</td>
<td>Rogers RO-3003 clad and bondply</td>
</tr>
<tr>
<td>Material</td>
<td>Supplier</td>
<td>Dk</td>
<td>Df</td>
<td>Tg °C</td>
<td>Td °C</td>
<td>CTE Z</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-----------</td>
<td>----------</td>
<td>-----</td>
<td>-----</td>
<td>-------</td>
<td>-------</td>
<td>-------</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>370HR</td>
<td>Isola</td>
<td>4.00</td>
<td>0.0210</td>
<td>180</td>
<td>340</td>
<td>2.8%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>185HR</td>
<td>Isola</td>
<td>4.00</td>
<td>0.0200</td>
<td>180</td>
<td>340</td>
<td>2.7%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4000-13</td>
<td>Nelco</td>
<td>3.60</td>
<td>0.0090</td>
<td>210</td>
<td>Not listed</td>
<td>3.5%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FR408HR</td>
<td>Isola</td>
<td>3.68</td>
<td>0.0092</td>
<td>190</td>
<td>360</td>
<td>2.8%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Polyimide</td>
<td>Isola</td>
<td>3.76</td>
<td>0.0170</td>
<td>260</td>
<td>396</td>
<td>1.5%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>I-Speed</td>
<td>Isola</td>
<td>3.64</td>
<td>0.0060</td>
<td>180</td>
<td>360</td>
<td>2.7%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RO4000</td>
<td>Rogers</td>
<td>3.55</td>
<td>0.0027</td>
<td>280</td>
<td>390</td>
<td>2.0%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Megtron 6</td>
<td>Panasonic</td>
<td>3.60</td>
<td>0.0031</td>
<td>210</td>
<td>410</td>
<td>2.8%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>I-Tera MT40</td>
<td>Isola</td>
<td>3.45</td>
<td>0.0031</td>
<td>200</td>
<td>360</td>
<td>2.8%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RO3003</td>
<td>Rogers</td>
<td>3.00</td>
<td>0.0010</td>
<td>Not listed</td>
<td>500</td>
<td>1.0%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Astra MT77</td>
<td>Isola</td>
<td>3.00</td>
<td>0.0017</td>
<td>200</td>
<td>360</td>
<td>2.8%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tachyon 100G</td>
<td>Isola</td>
<td>3.02</td>
<td>0.0021</td>
<td>200</td>
<td>360</td>
<td>2.5%</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Astra® MT77 vs. PTFE

- Thermoset system to replace PTFE laminates.
- Dk of 3.0 up to 100 GHz across all thicknesses.
- Smooth copper foil (VLP2) while maintaining adhesion.
- Stable Dk and Df over lifetime and over different temperatures.
- Well suited for PCB hybrid constructions.
- Can be foil laminated.
- Mostly Resin with a very thin spread glass weave
HSD Epoxy Material Benefits

Technically Appropriate for Frequency of Operation

Spread Glass Options

<table>
<thead>
<tr>
<th>Glass Style</th>
<th>370HR®</th>
<th>408HR®</th>
<th>I-Speed®</th>
<th>I-Tera®MT40</th>
<th>Tachyon®100G</th>
<th>Astra®MT77</th>
</tr>
</thead>
<tbody>
<tr>
<td>1035</td>
<td></td>
<td></td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>1067</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>1078</td>
<td></td>
<td></td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>1086</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td></td>
</tr>
<tr>
<td>3313</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
<td>Y</td>
</tr>
</tbody>
</table>
Fiberweave Routing vs. Spread Weave Material

- Std FR4 (i.e. 1080): Fabric Weave and Resin have different Dk’s, one trace routed over glass and the other trace over resin, results in a mismatched impedance
- Spread Weave: Provides a consistent Dk and will ensure a matched characteristic impedance for both signals in the diff-pair

Isola’s spread weave product families:
- 370HR®
- I-Tera® MT40
- I-Speed®
- Astra® MT77
- Tachyon® 100G
Flex for 5G and mmWave

- **Why Flex?**
  - Flex is “still” primarily driven by packaging needs
  - Separation of digital/analog/antenna components
  - Elimination of cables and connectors (rigid-flex)

- **Where Do Flex Materials Play?**
  - Feedlines, Beamformers, Antennas (1%-5% Bandwidth)
  - Hybrid board layers

- **Benefit to Flex Materials?**
  - Glass-Free (Skew Elimination, Phase Flat Transmission Lines)
  - Low relative permittivity and loss tangent vs glass epoxy

- **(FCCL) Flex Copper Clad Laminate with excellent performance:**
  - Some polyimide substrates such as Pyralux® AP have electrical properties comparable to (LCP) Liquid-Crystal Polymers
  - Polyimide and LCP test vehicles show similar performance in a high moisture environment
  - **Pyralux® TK** Polyimide and Teflon® based FCCL’s can outperform both LCP and all polyimide substrates.
### Performance Properties of Flex Dielectric Films

<table>
<thead>
<tr>
<th>Property</th>
<th>Unit</th>
<th>Method</th>
<th>Kapton® HN</th>
<th>Pyralux® AP</th>
<th>Pyralux® HT Bondfilm</th>
<th>Pyralux® TK</th>
<th>LCP Liquid-Crystal Polymers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thicknesses</td>
<td>mil</td>
<td>–</td>
<td>1 - 5</td>
<td>1 - 6</td>
<td>1-4</td>
<td>2 – 4</td>
<td>1, 2, 4</td>
</tr>
<tr>
<td>Dk @ 10 GHz</td>
<td>–</td>
<td>Method 2.5.5.5</td>
<td>3.4</td>
<td>3.2</td>
<td>3.0</td>
<td>2.5</td>
<td>3.0 - 3.1</td>
</tr>
<tr>
<td>Df @ 10 GHz</td>
<td>–</td>
<td>Method 2.5.5.5</td>
<td>0.010</td>
<td>0.002 - 0.003</td>
<td>0.003</td>
<td>0.002</td>
<td>0.001 - 0.002</td>
</tr>
<tr>
<td>% Moisture uptake</td>
<td>%</td>
<td>Method 2.6.2</td>
<td>2.8</td>
<td>0.8</td>
<td>0.8</td>
<td>0.6</td>
<td>0.04</td>
</tr>
<tr>
<td>CTE (x-y axis)</td>
<td>ppm/°C</td>
<td>50 to 250 °C</td>
<td>20</td>
<td>25</td>
<td>25</td>
<td>27</td>
<td>18</td>
</tr>
<tr>
<td>CTE (z axis)</td>
<td>ppm/°C</td>
<td>50 to 250 °C</td>
<td>115</td>
<td>90</td>
<td>90</td>
<td>102</td>
<td>120</td>
</tr>
<tr>
<td>Peel strength</td>
<td>N/mm IPC-TM650</td>
<td>N/A</td>
<td>2.0 (ED)</td>
<td>1.6 (RA)</td>
<td>N/A</td>
<td>1.2 (RA)</td>
<td>1.0 (ED) 0.4 (RA)</td>
</tr>
<tr>
<td>Tg</td>
<td>°C</td>
<td>DMA</td>
<td>360 - 410</td>
<td>220</td>
<td>220</td>
<td>270</td>
<td>–</td>
</tr>
<tr>
<td>Tm</td>
<td>°C</td>
<td>DSC</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>300</td>
<td>280 - 315</td>
</tr>
<tr>
<td>Flammability</td>
<td>–</td>
<td>UL94</td>
<td>V-0</td>
<td>V-0</td>
<td>V-0</td>
<td>V-0</td>
<td>V-0</td>
</tr>
</tbody>
</table>

The properties of Pyralux® AP and HT v. LCP are similar while Pyralux® TK has a low dielectric constant due to the presence of Teflon®. All are superior to “traditional” FPC Kapton®.
“Materials Matter” Flex PCB Types

Type 1: Single Sided

Type 2: Double Sided

Type 3: Multi-Layer

Type 4: Rigid-Flex
### “Materials Matter” Flex Construction

<table>
<thead>
<tr>
<th>Term</th>
<th>Description</th>
<th>Products</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>Copper Clad - 2 Layer also called; FCCL – Flexible Copper Clad Laminate All-polyimide clad Adhesiveless clad</td>
<td>Copper foil laminated directly to the polyimide base material.</td>
<td>Pyralux® AP</td>
<td>![Pyralux® AP Image]</td>
</tr>
<tr>
<td>Copper = Layer 1</td>
<td></td>
<td>Pyralux® HT</td>
<td>![Pyralux® HT Image]</td>
</tr>
<tr>
<td>Polyimide = Layer 2</td>
<td></td>
<td>Pyralux® AC</td>
<td>![Pyralux® AC Image]</td>
</tr>
<tr>
<td>Copper Clad - 3 Layer also called; FCCL – Flexible Copper Clad Laminate</td>
<td>Copper foil laminated to one or both sides of a adhesive coated polyimide base material.</td>
<td>Pyralux® LF</td>
<td>![Pyralux® LF Image]</td>
</tr>
<tr>
<td>Copper = Layer 1</td>
<td></td>
<td>Pyralux® FR</td>
<td>![Pyralux® FR Image]</td>
</tr>
<tr>
<td>Adhesive = Layer 2</td>
<td></td>
<td>Pyralux® AC</td>
<td>![Pyralux® AC Image]</td>
</tr>
<tr>
<td>Polyimide = Layer 3</td>
<td></td>
<td>![Pyralux® AC Image]</td>
<td></td>
</tr>
<tr>
<td>Single-sided Clad</td>
<td>Copper foil on one side of a flexible dielectric base material.</td>
<td>Pyralux® LF</td>
<td>![Pyralux® LF Image]</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pyralux® FR</td>
<td>![Pyralux® FR Image]</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pyralux® AC</td>
<td>![Pyralux® AC Image]</td>
</tr>
<tr>
<td>Double-sided Clad</td>
<td>Copper foil on both sides of a flexible dielectric base material.</td>
<td>Pyralux® AP</td>
<td>![Pyralux® AP Image]</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pyralux® HT</td>
<td>![Pyralux® HT Image]</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pyralux® LF</td>
<td>![Pyralux® LF Image]</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pyralux® FR</td>
<td>![Pyralux® FR Image]</td>
</tr>
</tbody>
</table>
**Pyralux® AP**
All Polyimide construction. Industry standard double sided clad for building flex and rigid flex PCB's

**Pyralux® AC**
All Polyimide construction. Single sided roll clad for building flex and rigid flex PCB's

**Pyralux® HT Clad**
Adhesiveless Polyimide Double sided clad for High Temperature applications. (>225 Service Temperature)

**Pyralux® HT Bondfilm**
Adhesiveless Polyimide bonding material for High Temperature applications. (>225 Service Temperature)

**Pyralux® TK Clad**
A flexible copper clad Kapton/Teflon laminate and bonding film system formulated for high-speed digital and high-frequency flexible and stripline applications.

**Pyralux® TK Bondply**
A flexible copper clad laminate and bonding film system formulated for high-speed digital and high-frequency flexible circuit applications.

**Pyralux® LF Clad**
Acrylic-based copper clad laminate, which has been the industry standard in high reliability applications for over 35 years.

**Pyralux® LFB**
Acrylic-based coverlay, bondply and sheet adhesive system. The industry standard in high reliability applications for over 35 years.

**Pyralux® LF Bondply, Coverlay and Sheet Adhesive**
A coverlay made with Kapton® brand black polyimide film for high performance applications where matte-black material enhances design aesthetics.

**Pyralux® FR Clad**
A copper-clad laminate composite of DuPont™ Kapton® polyimide film with copper foil, one or both sides, bonded together, flame-retardant, C-staged acrylic adhesive.

**Pyralux® FR Bondply, Coverlay and Sheet Adhesive**
Acrylic-based flame retardant coverlays, bondplys and sheet adhesives for products requiring UL rating.

**Pyralux® APR**
Copper-clad resistor laminate -polyimide composite bonded to copper and resistor foil

**Interra™ HK04J**
2 Layer Flex Routing with Return Path

When routing a signal over a flex, make the adjacent flex layer a **45° GND hatch pattern** and on that same layer make a **shadow GND copper trace**. This provides uninterrupted return path, EMI shielding and retains flexibility.
What is a Metallic Paste Z-Interconnect?

- Z-Interconnect allows the connection of two adjacent layers in a PWB or SCP/SIP
- It uses a conductive paste (proprietary formulations) and a joining layer
- Actual joining takes place during sub-composite or composite lamination
- Usable with a Sequential Build-up on a multilayer core

Advantages with Z-Interconnect

- Reduces the amount of lamination cycles
- Can solve Via Aspect Ratio and Annular Ring concerns for Class-3
- Elimination of back drilling
- Significant increase in routability (traces below vias)
- Potential to eliminate PTH (full "blind via" build)
- Significant gain in performance: reduces parasitic noise/loss
Z-Interconnect for PWB With Sub-Composites

4 Sub-composites, each consisting of:
- 4 – Buried Via Core structures
- 2 – Lamination Cycles (4 singles + 1 combined)
- 4 - 0.092” thick Cores with 0.008” drills
- Total board thickness =0.380”

1 (48:1 Aspect Ratio) or 4 (11:1 Aspect Ratio)

3 Z-Interconnect Layers
Stub reduction
No aspect ratio limitations

Improved routability
above / below terminated vias, no need for back drilling

72 Layer PCB
with Z-Interconnects

Courtesy i3 electronics
Ormet Paste Applications

Thick Boards – Aspect Ratio Reduction
• Overall thickness reduction
• Reduction of aspect ratio by splitting a board into separate builds and joining with Ormet paste which improve plating and drill quality.

High Speed Cap – Mixed Dielectric Builds
• No hole plating of high speed layers
• Separate fabrication of high speed layer results in smoother outer layer surface resulting in improved Rf performance.
Z-axis Interconnects in PCB’s

“Any Layer” HDI using Paste

Step 1

Z-Interconnects Paste applied prior to lamination to connect 2-layer cores in a single process step

5 cores

Step 2

1 Lamination Cycle
(5 cores + 4 Z-Interconnect)

Step 3
HDI Definition

High Density Interconnect, or HDI, circuit boards are printed circuit boards with a higher wiring density per unit area than traditional printed circuit boards (PCB). In general, HDI PCBs are defined as PCBs with one or all of the following: micro-traces, VIP (Via In Pad), microvias, blind vias, buried vias or other microvia technique, built-up laminations and high signal performance considerations.

Printed circuit board technology has been evolving with changing technology that calls for smaller and faster products. HDI products are more compact and have smaller vias, pad pitch, and lines and spaces. As a result, HDIs have denser wiring, which means lighter, compact, lower layer count PCBs. In essence, they can perform better and be more reliable.
Microvia IPC-2221 Definition

Microvias can be either a buried via or blind via.

A microvia is defined as a blind structure (as plated) with a max. aspect ratio of 1:1 terminating on or penetrating a target land, with a total length of no more than 0.25 mm [0.00984 in] measured from the structure’s capture land foil to the target land.

When utilizing a microvia structure, typically thinner dielectric materials will contribute to higher reliability. This is due to the physics of plating copper into the hole.

Note: X/Y = Microvia Aspect Ratio, with X=Y 1:1
Typical aspect ratio is about .8:1 (.004 dielectric with a .005 drill)
The microvia built in CAD software will result with a normal pad/hole/pad scenario.

Landing pad can be slightly larger if possible, but at least the same size.

Recommended pad diameter is drill + .006” More pad helps prevent breakout.

Correct uVia layer definition and usage ensures proper utilization that will be in line with manufacturing capabilities. Assign layer pairs.

Plan the routing requirements, signal performance in conjunction with the manufacturing capabilities, at the start of the layout.
Landless Vias

***Warning***
The best practice is that vias should always have an **annular ring to avoid drill breakout** and unless your fab claims different, use a sufficient annular ring!

Landless Vias: **From the past and needed for the future**...

Once done as a proprietary practice and not shared with industry, it has remained lost. Currently, the IPC-6012 standards committee is investigating this technology with multiple test cases, so stay tuned!

Landless Vias: Once the annular ring from the foil-copper completely disappeared, the ductility of the plated-copper in the hole, makes this work.
IPC Types of HDI Stack-up (IPC-2226 HDI std.)

VIP, Via In Pad
(Plated surface, planar finish)

IPC Type I

IPC Type II

IPC Type III
(2-N-2, etc…)

IPC Type IV
(over core)

IPC Type V-VI
(ALV, any layer via)
Filled with sintered CU paste
What is done on the top side of the stack-up will be done to the bottom side to achieve a balanced stack up. You will get the potential for the same type microvias on the bottom, even if you don’t use them.

Remember additional microvia plating thickness for each layer!
Upcoming manufacturing process development will allow circuits to be designed with lines and spaces in the range of 1-2 mil (25-50uM) and below, with strict impedance control.

mSAP – Modified Semi-Additive Process

SAP – Semi-Additive Process

Note: The near **vertical trace** shapes with the mSAP Process vs. **Trapezoidal trace** shape with the Print & Etch Process shown below.

*Image cited from SAMSUNG ELECTRO-MECHANICS.*
HDI uTraces - Using mSAP and SAP

Trace width typically will closely match the dielectric thickness to achieve a 50 ohm characteristic impedance.

Drill diameter typically will match the dielectric thickness to achieve a 1=1 aspect ratio for plating purposes.

Thus, a thin dielectric thickness allows for thinner traces and smaller holes for microvias.

This in turn pushes the envelope concerning material and process.
Routing Density

Via Starvation

Internal layers are freed up for routing.

Plane layers no longer exhibit the “Swiss Cheese” effect.
Via fanout on a grid pattern aligned with the BGA pitch is typically beneficial for consistency and optimization.

However, microvias may require varied angled via fanouts from the BGA land to maximize routing channels on the escape layer.

Careful consideration should be observed to avoid solderability issues on BGA land sizes in assembly.
Pin-escape feasibility study:
Use a trial route to see how many traces will fit in an area and what via routing strategy will solve the pin-escape.

This may be required to determine the stack-up and manufacturing requirements. This can be saved as a re-use.

Plan the usage, see it from a cross section perspective.
Staggered or Stacked Microvias

**Staggered Microvias** don't require plugging and the copper caps.

**Stacked Microvias** will require plugging and the copper caps.

Note:
Both Staggered and Stacked microvias will require additional plating requirements and thus will add to increased board thickness.

With every additional microvia layer there exists many manufacturing steps (approx. 40) and can be cost considerations.
Stacked Vias require metal filling and capping as the glass is squeezed between layers.

They restrain the Z-Axis expansion because they both expand at a different CTE. Because the dielectric material is so thin that helps this concern.
The core-via in the center of the board should be plugged with non-conductive epoxy so it will expand in a similar fashion to the surrounding dielectric. Never stack micro-vias on top of the core-via for dis-similar Z-Axis CTE reasons.

If laser vias are stacked on the core-via, they would require cap plating. This practice has proven to cause reliability problems. *The error condition is shown in these lower left images.*
Benefits and Challenges of Using HDI

- Smaller - Miniaturization of products, pitch & features.
- More expensive as a general rule, due to more fabrication processes. Might be less expensive if layer reduction is achieved and the cost per connection improves.
- Performance - 1/10th parasitic of PTHs, fewer stubs, improved noise margins.
- Lighter - Lower substrate weight, thickness and volume.
- Reliable – Because of better aspect ratios vias, thinner dielectrics, and improved via metalization. This has been challenged of late and must be considered…
- Testability - Is significantly challenged due to limited access to all nets.

*Research all requirements at the start of the layout!*
Benefits and Challenges of Using HDI

- Placement Feasibility – SMT Parts will not fit with room for pin escapes to PTH vias, so use VIP via in pad.
- Standard PTH vias are too large to pin-escape a uBGA (Typically a .65mm or below pitch device)
- High Speed or RF performance – unwanted parasitics or excess inductance from standard PTH vias
- Requirement for thin PCBs in some market spaces.
- Back to back large active BGA’s on both sides of PCB.
- RF on Primary Side / Digital on Secondary side
- Use of OrMet Sintered Paste, any layer via
Increased Routing Density (HDI)

- Fine pitch BGA – <.65mm, .5mm (must be used)
- Sequential lamination - multiple stacking options
- Thin materials - down to 50um [0.002 inch] dielectric
- 3D Routing is more challenging from a layout perspective.
- Improved signal and power integrity performance

Does it take longer to complete an HDI design? Typically “Yes” = You’re routing 3D, Z-Axis
Frequencies and wavelengths are useful for calculations for dielectric constants and loss tangents. It is a measured cycle known as a frequency. This forms a switching binary language.

Rise and fall transitions contain many frequencies and their harmonics, within an operating frequency, as shown on the top image. Think of them as squared off.

However, the important feature of digital pulses is the slew rate (rise and fall time), as measured in ns (nano) or ps (pico) seconds as shown on the middle image.

Repeated cycles are measured as an eye-diagram. They should cross in a consistent point or you have a condition known as skew. Red eye diagram shows 155ps of inter-pair skew.
Signal Propagation and Return (energy moving forward & back) However, it is not really forward and back, rather, the energy field is immediate between trace and plane within the dielectric material

“Materials Matter”

It's not forward and back. Fields exist in the dielectric material.
In a circuit board, a high capacitance and low inductance environment is ideally created when a forward current is completely surrounded by its return current, totally containing both the electric and magnetic fields, as a coax cable might be.

An ideal setting would have a consistent balanced impedance between the forward path and the return path.

*Why this is important – you’re not just connecting a route, rather you are managing an EM field.

Energy fields exists between the trace and the plane (return path) within the dielectric material.
Perfect 12 Layer Stack-up and Route

Secret of BLACK MAGIC for Stack-ups

When defining any Stack-up:

• Sketch a resistor symbol from your signal layer to an adjacent uninterrupted GND plane

• Sketch a capacitor symbol from your voltage layer to an adjacent uninterrupted GND plane

| Layer Name | Type | Material | Thickness (mil) | Diameter (mil) | Orientation | Orientation
|------------|------|----------|----------------|---------------|-------------|-------------
| LAYER-1    |      |          |                |               |             |             
| LAYER-2    |      |          |                |               |             |             
| LAYER-3    |      |          |                |               |             |             
| LAYER-4    |      |          |                |               |             |             
| LAYER-5    |      |          |                |               |             |             
| LAYER-6    |      |          |                |               |             |             
| LAYER-7    |      |          |                |               |             |             
| LAYER-8    |      |          |                |               |             |             
| LAYER-9    |      |          |                |               |             |             
| LAYER-10   |      |          |                |               |             |             
| LAYER-11   |      |          |                |               |             |             
| LAYER-12   |      |          |                |               |             |             

12 Layer Stack-up Diagram
**Stack-ups that Support Functional EMC**

GND (0.0V) is the most important signal in the circuit!

Ground is where you plant tomatoes, but we commonly refer to this signal as GND (0.0V)

- **GND (0.0V)** is what is used to reference every signal for a return path. Never route signals over split GND plane!

- **GND (0.0V)** is what is used to reference every PWR net (Voltage rail). Never place PWR’s over split GND plane!

Get these last two bullets right and you have solved a significant amount of your Signal/PWR Integrity concerns!
Avoid this Stack-up Condition: Dual Asymmetrical Stack-up

- You conceptualized and setup your stack-up
- You secured a stack-up from the fabricator
- You make your fabrication drawing reflect this
- You require the fabricator to utilize an impedance coupon based on this stack-up
- You require the fabricator to TDR test the coupon
- You setup your rules and simulations to consider these as 2 - 50 ohm SE impedance lines

BUT…
Avoid this Stack-up Condition: Dual Asymmetrical Stack-up

- BUT...
- Because the board routing was so dense.
- Because alternating layer perpendicular routing strategy went out with through hole technology.
- Because most of the ICs on a dense board are BGAs.
- Because BGAs want to route “wagon-wheel” style away from the center of the chip.
- Because the routing channels where available.
- Because you did not want to add extra layers.
- So you routed like this.

You have changed the actual impedance and made a crosstalk nightmare, even though the coupon is good.
Solid GND Plane tightly coupled to a Split Voltage plane.

Transporting a Voltage/GND pair of planes with **Low Inductance** & **High Capacitance**

Located in the center core
Power Delivery – The Usage

Bring the voltage to the side of usage (BGA) as the blue color indicates, i.e. L3 & L8

This creates a tightly coupled pair of planes

High Capacitance & Low Inductance close to their respective usages

They need to have a balanced stack-up construction

(Not just what you use but how you use it)

Buried Capacitance material can provide a high performance solution for High density and High Speed

**DuPont’s In-Terra®**

Buried Capacitance material (better because no fillers)
Routing – RF style

The **RF circuit** is often lower in volume of routing but it is extremely more sensitive to every feature in its routing topology.

**Wider traces provide a less lossy transmission** and require increased dielectric thickness to maintain 50 ohm impedance. “Welling down” to a lower layer may be required. Reduce Signal Loss (Df): **wide traces, low loss material, low profile copper**.

Vias on transmission lines are problematic because they often leave **radiating stubs**.

**Coplanar waveguides** are often utilized to control EM signature with shield GND vias along the way.

**Isola’s ASTRA®MT77**

Is a great RF laminate with low loss performance points similar to PTFE material but it’s not dissimilar to other epoxy materials in the stack-up, will improve long term product reliability. (Mostly neat resin w/ a thin weave)
As we get ready to show some of the Advanced Routing features

- Software enhancements and industrial automation should increase productivity, accuracy & quality.
- You should always seek to improve your knowledge base and competence with technology trends and seek to be a Master of your software tool!

**Today’s Designer must meet 3 Perspectives for Success**

**Design For:**
- DFSolvability
- DFPerformance
- DFManufacturability

1. Layout Solvability
   A skill set to solve the place and route of all parts and connections often with complex High Density Interconnect (HDI)
   … and master this on your CAD Tool

2. Making Revision One Work
   Signal Integrity/EMC
   Power Delivery
   Thermal

3. DFX Manufacturability
   DFX all considerations producing high yield and lower cost
Thank you for your Attention!

Questions?

Stay tuned for
Software demonstration of
Advanced Routing Methods

mcreeden@insulectro.com