Ryan Satrom, Jason Mroczkowski
Multitest | ECT Interface Products

High Frequency Solutions for Wafer Level Package Test

June 9, 2010
San Diego, CA USA
Drive Toward Final Test at Wafer Level

• **Wafer Level Packaging Offers Low Cost, High Frequency solution**
  – Eliminates second (package) test
  – Increased package pitch allows Pogo® pin architectures
  – Improved electrical performance compared to traditional packaging

• **Test Interface**
  – Often performance limiter for tests at high frequency

• **High-Frequency Testing at Wafer Level Can Be Challenging**
  – Successful solutions require signal integrity optimization
  – Optimization must include full test interface
  – Test hardware can be very costly

Pogo® is a registered trademark of Everett Charles Technologies
New High Frequency WLP Solution

• **High-frequency Pogo Pin Solution Addresses Challenge**
  – PCB design guidelines for optimized signal integrity
  – Correct Pogo Pin for each specific application
  – Cost-effective

• **Turnkey Solution Specific to High-Speed WLP Test Market**
  – Design
  – Simulation
  – Fabrication
  – Assembly
Agenda

• Simulation-Measurement Correlation
  – Both PCB and contactor
  – Ensures maximum simulation accuracy

• Frequency-Based PCB Design Guidelines
  – Provides rules based on frequency
  – Avoids unnecessary overdesign and additional costs

• Hardware Selector Tool
  – Ensures performance specifications are met
  – Results based on application-specific variables

• Example Application
Simulation- Measurement Correlation

• Simulation Proven Through Extensive Characterization
  – Correlates simulation to measurement
  – Test hardware fabricated using Multitest manufacturing processes
  – Includes PCBs, contactors, and PCB/contactor interactions
  – Ensures simulation model accuracy
PCB
• Microstrip traces
• 1”, 4”, 6”
• Nelco 4000-13/Sl

Contactor
• MER040
• 0.4mm pitch
• G-S-G Configuration

Correlation

MEASUREMENT SIMULATION
System Correlation

- Stripline Trace to BTM080 Contactor
  - Includes via transition from PCB to contactor
- Good Correlation out to 20GHz

**MEASUREMENT**  **SIMULATION**
Agenda

• Simulation-Measurement Correlation
  – Both PCB and contactor
  – Ensures maximum simulation accuracy

• Frequency-Based PCB Design Guidelines
  – Provides rules based on frequency
  – Avoids unnecessary overdesign and additional costs

• Hardware Selector Tool
  – Ensures performance specifications are met
  – Results based on application-specific variables

• Example Application
PCB Design Guidelines

- **Design Rule Considerations**
  - Typically derived from ideal environment
  - Often do not relate to actual performance
  - Rules valid below 500 MHz do not apply above 1 GHz

- **Design Rules Improved Through Simulation**
  - Matched to application frequency
  - Ensures performance meets application requirements
  - Minimizes cost

- **Examples of Optimized Rules:**
  - Connector footprints, stub drilling, ground via optimization (shown)
  - Relay/component footprints, trace spacing, component locations, etc.
PCB Design Guidelines – Connector Footprint

- **Datasheet Specification:**
  - Bandwidth = 18.0 GHz

- **Simulation:**
  - Top microstrip -1dB: 1.2 GHz (Standard); 2.9 GHz (Optimized)
  - Bottom microstrip -1dB: 1.9 GHz (Standard); 6.2 GHz (Optimized)

- **Optimized Footprints Now Used as Design Standard**

![Graphs showing insertion loss vs. frequency for top and bottom microstrips](image-url)
PCB Design Guidelines – Stub Drilling

• **Stubs**
  – Create undesired noise in signal path
  – Creates signal reflections that limit bandwidth

• **Stub Drilling**
  – Removes undesired stubs from signal paths
  – Becomes increasingly important as frequencies increase
PCB Design Guidelines – Stub Drilling

- **Stub Drilling - Rule of Thumb:**
  - Stub length must be less than $\frac{1}{4}$-wavelength of max frequency

- **Simulation Results:**
  - Stub drill required well below $\frac{1}{4}$-wavelength frequency
  - Required in GHz range
  - Much less dependent of signal layer than $\frac{1}{4}$-wave formula suggests

---

**Backdrill Recommendations**

<table>
<thead>
<tr>
<th>LYR</th>
<th>Stub Len (mil)</th>
<th>$\frac{1}{4}$-λ Equation</th>
<th>Bandwidth (-1dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lyr03</td>
<td>234</td>
<td>6.5 GHz</td>
<td>1.9 GHz</td>
</tr>
<tr>
<td>Lyr11</td>
<td>167</td>
<td>9.1 GHz</td>
<td>2.3 GHz</td>
</tr>
<tr>
<td>Lyr22</td>
<td>83</td>
<td>18.2 GHz</td>
<td>2.4 GHz</td>
</tr>
<tr>
<td>Lyr30</td>
<td>17</td>
<td>89.1 GHz</td>
<td>2.3 GHz</td>
</tr>
</tbody>
</table>
PCB Design Guidelines – Ground Via

- **Ground Via Proximity – Rule of Thumb:**
  - No standard design rules for ground via location

- **Simulation Results:**
  - Ground via location have significant impact on performance
  - Design rules created based on frequency
PCB Design Guidelines – Summary

- Connector Manufacturer Specifications are not Accurate
- Quarter-Wavelength Stub Drill Approximation Insufficient
- Design Rules Require 3D Simulation
  - Accounts for all necessary variables:
    - PCB Thickness
    - Signal Layer
    - Ground Via Proximity
    - Stub Length
Agenda

• **Simulation- Measurement Correlation**
  – Both PCB and contactor
  – Ensures maximum simulation accuracy

• **Frequency-Based PCB Design Guidelines**
  – Provides rules based on frequency
  – Avoids unnecessary overdesign and additional costs

• **Hardware Selector Tool**
  – Ensures performance specifications are met
  – Results based on application-specific variables

• **Example Application**
PCB Material & Contactor Selection

- **Factors impacting High Frequency Hardware Selection**
  - PCB material
  - Trace topology
  - Trace length
  - Device pitch
  - Signaling type – single-ended or differential
  - Ground-signal configuration – G-S, G-S-G, G-S-S-G

- **Multitest Hardware Selection Tool**
  - Accounts for all relevant variables
PCB Selection Parameters

• **Trace Topology**
  – Stripline (Internal), Microstrip (External)

• **Trace Length**
  – 2”, 4”, 8”, 12”, 16”
Contactor Selection Parameters

• Pitch
  – 0.3mm, 0.4mm, 0.5mm
  – As pitch increases, impedance increases

• Ground Configuration
  – Signal Type
    • Single-ended (G-S, G-S-G) – one signal trace
    • Differential (G-S-S-G) – two signal traces
  – Number of adjacent grounds
    • G-S, G-S-G
    • As number of grounds increases, impedance decreases
Agenda

• Simulation-Measurement Correlation
  – Both PCB and contactor
  – Ensures maximum simulation accuracy

• Frequency-Based PCB Design Guidelines
  – Provides rules based on frequency
  – Avoids unnecessary overdesign and additional costs

• Hardware Selector Tool
  – Ensures performance specifications are met
  – Results based on application-specific variables

• Example Application
Application Example

• Example
  – 48 QFN, 0.5 mm pitch, 7x7 mm
  – Application: Gigabit Ethernet Controller
  – 2.5 Gbit/s Differential PCI-express
    • Requires 3.75 GHz bandwidth (3\textsuperscript{rd} harmonic)

• Goal – Meet Application Frequency Requirements
  – Determine connector type
  – Determine if stub drill is required
  – Select optimal PCB material
  – Select optimal Pogo pin for contactor
Application Example – Connector Type

- Requires 3.75 GHz bandwidth
- Connector choice
  - Surface mount, right-angle, top-launch SMA connector
  - Allows for backdrill
  - Minimizes loss without use of high-cost connector

<table>
<thead>
<tr>
<th>Connector Type</th>
<th>Loss @ 3.75GHz</th>
<th>Bandwidth (-1dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>High-Cost Surface SMA</td>
<td>0.1dB</td>
<td>15.0 GHz</td>
</tr>
<tr>
<td>Med-Cost Surface SMA</td>
<td>0.1dB</td>
<td>11.1 GHz</td>
</tr>
<tr>
<td>Thru-Hole SMA</td>
<td>1.2dB</td>
<td>2.9 GHz</td>
</tr>
</tbody>
</table>
Application Example – Stub Drill

- Stub Drill Not Necessary at DUT or at Connector
- Performance Stays Within Loss Budget (-3dB) Without Stub Drill
- Recommendation
  - Use high-aspect ratio drilling at DUT
  - Eliminates the need for sequential lamination
Application Example – PCB Material

- Majority of Loss Due to PCB Trace
- Recommendation – Nelco 4000-13
  - Good high-speed material
  - Meets performance requirements without expense of exotic material

<table>
<thead>
<tr>
<th>PCB Material</th>
<th>Loss @ 3.75GHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>FR4</td>
<td>2.8dB</td>
</tr>
<tr>
<td>Nelco 4000-13</td>
<td>2.1dB</td>
</tr>
<tr>
<td>Rogers 4003</td>
<td>1.6dB</td>
</tr>
</tbody>
</table>
Application Example – PCB & Contactor

- **Application Specifications**
  - 0.5mm
  - GSSG (Differential)
  - Stripline
  - 8” PCB Trace
- **Nelco 4000-13**
- **MER050 is best solution**
  - System Bandwidth (-3dB) is greater than 3.75GHz

![MT CONFIDENTIAL Performance Matrix](image)
System Performance

- SMA Connector → 8” Stripline → MER050 Contactor
- System Bandwidth (-3dB): 3.9 GHz
Conclusion

• A new WLP high frequency solution is available

• Multitest Solution Uses:
  – Simulation and Characterization
  – PCB design recommendations based on high frequency 3D electromagnetic simulation
  – Hardware selection tool to fit the needs of each customer specific application