

PTH Core-to-Core Interconnect Using Sintered Conductive Pastes

Michael Matthews, Ken Holcomb, Jim Haley, and Catherine Shearer
Ormet Circuits, Inc.

10080 Willow Creek Road, San Diego, CA, 92131
P: 858.831.0010, F: 858.831.1437, web: ormetcircuits.com

Abstract

The market for high-layer-count printed circuit boards (PCB) containing blind and buried vias was once relatively small, and focused on specialized applications in the military and high end computing. The demand for these types of PCBs today is being driven by an increasing number of commercial applications in the telecommunications and semiconductor test market segments. These applications typically require high-aspect-ratio plated-through-holes (PTHs) and blind and buried vias in order to meet the applications interconnect density requirements. Blind and buried vias and high aspect ratio PTHs continue to present manufacturing challenges and frequently are the limiting features to achieving high fabrication yield. Multiple lamination cycles, the indeterminate yield of sub-cores, backdrilling and extended plating times add to the overall poor yield of high layer count PCBs. An attractive solution to the high-aspect-ratio PTH dilemma is to break the high-layer-count PCB into a number of subassemblies that can be 100% inspected and then be interconnected. Conductive paste-based interconnects are an attractive interconnect technology option, but passively loaded copper or silver filled pastes do not demonstrate the requisite performance and reliability. Sintering conductive pastes, which metallurgically bond directly to the copper pads of the PCB during a standard lamination cycle, do meet the requirements of these high-layer-count PCBs and at substantially lower cost than conventional manufacturing methods.

Key words:

Sintering conductive paste, telecommunications, connection of subassemblies, high-layer-count PCBs, high throughput and yield, PCB for semiconductor test, low cost and reliable

Introduction

The production of high-layer-count PCBs for the telecommunications, semiconductor test and high end computing industries requires an advanced set of fabrication techniques due to the complexity of their design. Frequently, manufacture of these PCBs requires the use of blind and buried vias, high-aspect-ratio plated through holes and backdrilling to meet their design specifications. The technical roadmap for industries utilizing high layer count PCBs indicates that future PCB designs will be increasingly complex and there is a drive toward higher interconnect density.

The reliability demands on high layer count boards are also increasing due to the increasing power dissipation requirements and higher operating frequencies of telecom, high end computing and military applications. This increase in thermal exposure

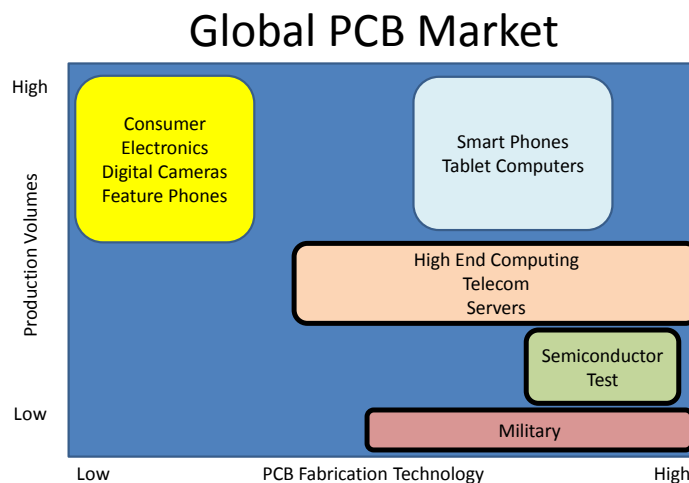


Figure 1. Global PCB Market (High Layer Count Applications in bold)

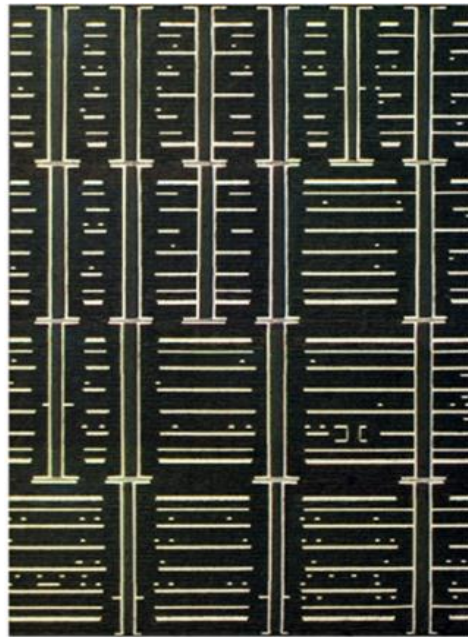


Figure 2: Interconnected subassembly cores. Source: Endicott Interconnect

over the lifespan of the PCB exacerbates the need for robust electrical interconnect structures throughout the board.

With the widespread adoption of 0.4mm pitch components, and the desire to route multiple circuit traces between copper pads, fabricators are being pushed to use smaller drill sizes and plate ever higher aspect ratio holes. The fine featured PTHs in these boards increase the cost of the final board as they are slow to fabricate, as well as expensive and difficult to plate consistently.

One solution to the difficulty of plating the high-aspect-ratio PTH in these PCBs is to break the PCB into subassemblies or 'cores' with manageable-sized PTHs. By breaking the PCB into cores of reasonable thickness, the PTHs of these cores can be plated in high quality, yield and throughput. These cores may then be interconnected using paste interconnects to form a monolithic PCB. Figure 2 illustrates a high-layer-count PCB manufactured by Endicott Interconnect Technologies formed from interconnected cores.

One of the important considerations when employing a subassembly core-to-core manufacturing strategy is determining an effective material for interconnecting the cores. Electrical performance, reliability, manufacturability and cost are all important considerations.

Interposer sheets consisting of copper-paste-filled vias drilled or laser ablated into prepreg are one option for interconnection of the cores during a conventional lamination cycle. The problem with this solution is that non-sintering copper-filled epoxy pastes used in these vias do not offer adequate electrical or reliability performance.

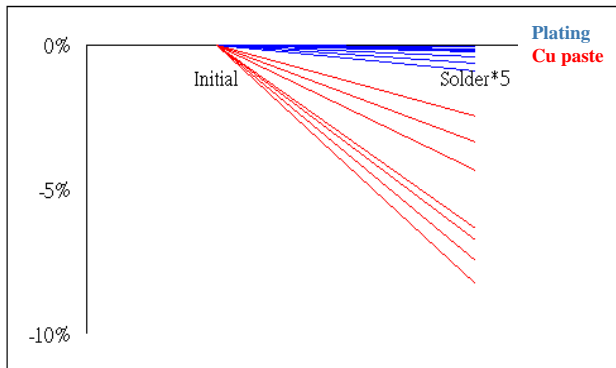
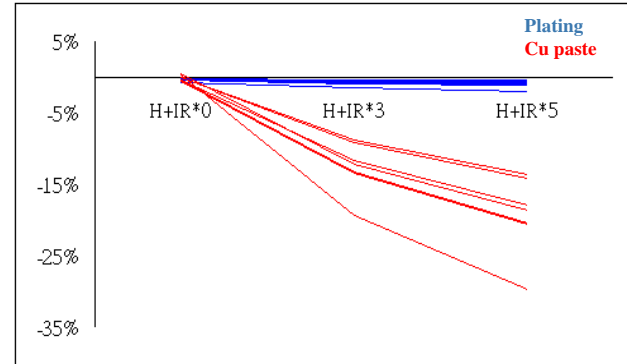
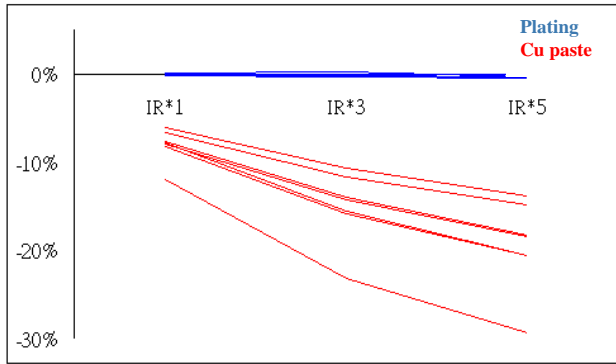
Sintering conductive pastes are an alternative technology solution that enables high performance and reliability in a core-to-core interconnection strategy for the manufacture of high-layer-count PCBs.

Sintering Conductive Paste vs. Conventional Conductive Paste

Conventional conductive epoxy materials are thermosetting or thermoplastic resins loaded with metal powders that provide electrical conductivity through percolation based conductive pathways. They can be either isotropic or anisotropic depending on the level of the conductive particle loading and the method of application. The main deficiencies of these passively loaded polymers occur with temperature and humidity fluctuations. These fluctuations cause loss of electrical continuity due to the oxidation of the contact pads and particles as well as the expansion of the polymer. In addition, no metallurgical connection is formed with the pads when using these conductive pastes. These issues are illustrated in the reliability data charts for a copper-paste filled via process shown in Figure 3. These conventional conductive paste materials cannot maintain an electrical shift below 10% when subjected to common reliability test requirements.

Plating:4-2-4 structure

Cu paste= HDI process with non-sintering copper paste



Resistance Shift after Reliability Test

	Plating	Cu Paste
IR	0% - 0.4%	6% -30%
Humidity and IR	0.2% ~ 2%	6% - 30%
288 solder	0.2% ~ 1%	2.5% - 8%
Requirement is <10% shift after reliability testing		

Figure 3: Core plus microvia process, Cu plated microvias vs. Cu-paste-filled microvias

The reliability and performance limitations of conventional conductive pastes are surmounted by sintering conductive paste materials. Sintering conductive pastes combine small particles of solder materials with small particles of solderable metals in a fluxing polymer binder. During lamination of the subassembly cores into a single PCB the solder particles melt, wet the solderable particles and form an interconnected metallic mesh. The metallic mesh formed has a melting point higher than the original solder alloys and is therefore stable through subsequent thermal excursions such as lead-free assembly operations. Because the solder particles also wet to the copper caps on the PTHs, the caps become permanently, metallurgically bonded to the metal mesh of the sintered paste interconnect. Therefore, during lamination, the sintering conductive paste delivers a continuous and robust thermal and electrical conduction through sintered metal joints. Figure 4 illustrates the sintering paste concept and a cross section of the sintering conductive paste after a lamination cycle. The dark areas in the sintered conductive paste are polymer phase that is entwined with the metallic mesh and reinforces it. The copper pad can be seen at the bottom of the right hand image in Figure 4 and it is evident that the solder particles in the unsintered paste have wetted both the metal particles in the paste as well as the copper pad to form a continuous metal mesh. In some ways, these sintering conductive paste materials are analogous to cermets, except that unlike the ceramic composites, processing of sintered conductive pastes is done at temperatures compatible with standard printed circuit board materials and common lamination cycle conditions. Once sintered, as seen in the cross section view, there are a variety of metallic phases in the metallic mesh, however, all of the phases present in the sintered material are stable in thermal exposures well above the initial process temperature and can thus withstand assembly or other subsequent thermal processing. The metal matrix does not remelt during lead-free solder reflow.

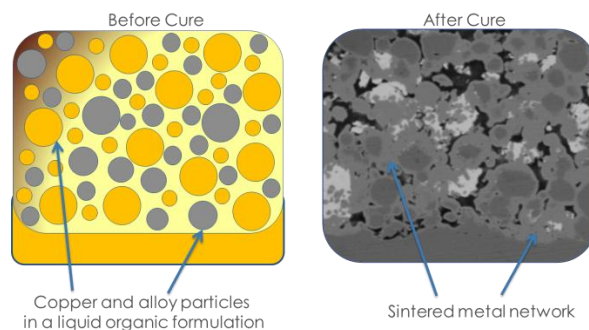


Figure 4: The sintering paste reaction

Table 1: Compilation of Reliability Analysis of PCBs with Sintered Paste Interconnects

Test	Test Conditions	Result
Thermal shock	-55°C – 125°C, 500 cycles	Pass, Max resistance change 7.5%
Temp Cycling	0-100°C, 1000 cycles	Pass
Humidity + bias testing	85°C, 85RH%, 50V bias, hold 240 hrs	Pass
Humidity + Thermal Aging	85°C, 85RH%, hold 1000hrs	Pass, Max resistance change 4.0%
High Temperature Storage	150°C, hold 1000h	Pass, Max resistance change 7.0%
Solder reflow test	Reflow @ 260°C, 5cycles	No delamination
Electrical test	No shorts or opens	Pass

These sintering conductive pastes overcome several of the disadvantages of copper paste materials. Of particular importance in vertical interconnection, they provide a low and stable electrical resistivity on the order of 50 $\mu\Omega$ -cm. This stable electrical performance in both normal use and under environmental and thermal cycling is the result of the alloyed metallurgical web. Furthermore, the sintered conductive paste has a high thermal conductivity of 25 W/mK,ⁱ an order of magnitude better than most conventional conductive adhesives. More critically, because the metallurgical joint is contiguous from pad-to-pad, the high bulk thermal conductivity is not squandered on poor interfacial transfer between the pad and the deposit.

The reliability of sintered paste interconnects in high-layer-count PCBs has been independently verified by third party sources. A compilation of the reliability data collected can be found in Table 1. As can be seen in the table, the sintered paste vias perform very well in standard industry test conditions for PCBs. Particularly notable is the performance gains over conventional conductive pastes in resistance to humidity and solder reflow conditions.

S-parameter measurements were made on a net that contained only PTH versus one that contained four sintered interconnects. Figure 5 demonstrates that the use of sintered interconnects in place of PTH does not significantly degrade the signal performance. Below 10GHz the difference between a copper barrel structure and a sintered via structure is negligible and above 10 GHz, there is only a slight degradation due to the additional via length. Overall the performance is similar to a solid copper barrel.ⁱⁱ

Cost

The value proposition for the use of blind and buried vias in the fabrication of high-layer-count PCBs is well understood. Blind and buried vias can increase the interconnect density of complex PCBs by reducing the number of outer layer plated through holes. Final yield is a primary cost driver for these complex PCBs, and the conventional drill and plate fabrication methods suffer from increased yield loss as layers counts, PTH aspect ratios and interconnect density requirements increase. Alternatively, utilizing sintered paste interconnects can improve yield by reducing the aspect ratio of the PTHs in the PCB sub-assemblies, and minimizing if not eliminating the need for outer layer drilling. Additionally, the sub-assembled PCBs can be fully tested before lamination, thus ensuring only good cores are used during final lamination. Ormet and Savansys, a PCB cost modeling organization, developed a cost model comparing the fabrication of a 50-layer PCB utilizing blind and buried vias vs. sub-assemblies interconnected by sintered conductive paste.

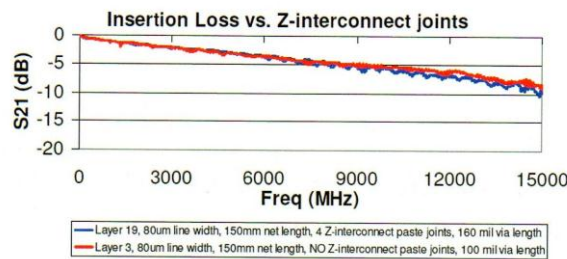


Figure 5: Insertion loss of nets with different numbers of sintered interconnects
Source: Endicott Interconnects

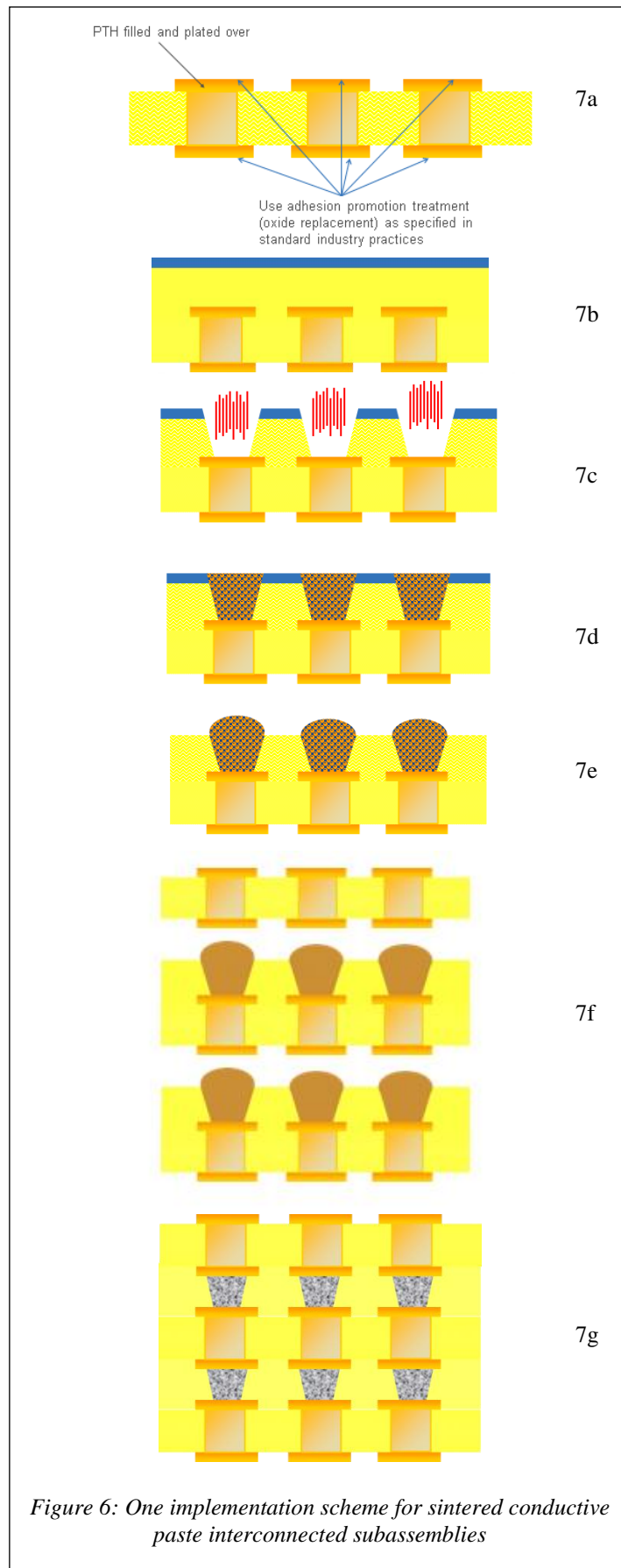


Figure 6: One implementation scheme for sintered conductive paste interconnected subassemblies

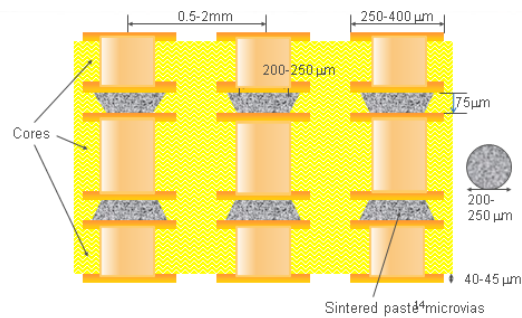


Figure 7: Typical dimensions of a high-layer-count PCB interconnected with sintered paste vias

The output of the model is summarized in Figure 6. The results of the model indicate that by using a sintered paste interconnect strategy, there can be substantial costs savings versus conventional processes. The savings is driven by eliminating the need for low yielding drill processes to connect the outer PCB layer, and that sintered paste interconnects enable the full electrical testing of each of the cores prior to lamination.

Implementation of Sintered Paste Interconnections

There are a variety of ways in which sintering paste interconnections can be implemented in core-to-core integration for high-layer-count PCBs. The specific method of manufacture will depend on the equipment, process flow and general practices of any one manufacturing facility. Flexibility of process flow is one of the advantages of paste interconnect technology. Because the sintering conductive paste interconnects during standard pre-preg lamination conditions, the number of subassemblies, the method of application of the sintering paste, the manufacturing flow and the specific configuration are all at the discretion of the manufacturer.

A generalized concept of how the sintered interconnections might be installed is depicted in Figure 6 (a-g). Subassembly cores are manufactured using standard PTH manufacturing methods. The number of subassemblies created for each PCB is at the discretion of the manufacturer. The PTHs are filled and cap plated to form a land for the sintered paste interconnects (a). Generally an adhesion promotion treatment is applied to the outer surfaces of the cores to ensure good bonding with the prepreg during lamination of the cores, as is standard industry practice for laminated PCB. Prepreg and a release sheet, generally PET film, are tack laminated to one side of the mating pairs of cores (b). The prepreg may consist of one or more layers depending on the thickness of the copper on the surface. Via holes are laser ablated through the PET film and prepreg using the copper cap plate as a stop for the laser (c). Alternatively the vias could be performed by punching, or mechanically drilling prior to attachment of the prepreg to the core. Sintering conductive paste is then deposited into the vias using the PET film as a mask (d). The PET film is removed (e). The subassemblies bearing the interconnect layers are aligned (f) and laminated under standard lamination conditions. When the board is removed from the lamination press, all of the electrical interconnections have been formed and the PCB is complete (g). Typical dimensions for high-layer-count PCBs manufactured using core-to-core interconnection with sintering pastes are depicted in Figure 7.

Conclusion

The market for high-layer-count PCBs using blind and buried vias is highly specialized on addressing the needs of high end computing, military, telecom and semiconductor test applications. These applications are requiring higher aspect ratio PTHs to meet the interconnect density requirements for 0.4mm pitch components. The cost of manufacturing these complex PCBs is driven by low yielding outer layer drilling processes and the use of blind and buried vias. Using conventional drill and plate process steps to build these PCBs is becoming unsupportable as interconnect density requirements continue to increase. Creating the necessary PCBs from several subassemblies that can be independently fabricated and joined using sintering pastes is an attractive alternative manufacturing strategy that leverages existing manufacturing flow. It is important to carefully consider the interconnect paste technology to ensure suitable reliability performance of these boards. Conventional copper pastes do not provide the requisite performance or reliability, but sintering conductive pastes are a viable option. Both the technical performance and the cost benefit of this approach have been favorably evaluated by independent sources. Core-to-core interconnection with sintering conductive pastes provides a vetted solution for the cost-effective manufacture of next generation high layer count PCBs.

References

- i R. E. Taylor, H. Groot, and J. Ferrier, TPRL Report 1584, "Thermophysical Properties of Conductive Ink."
- ii Das, Rabindra, Egitto, Frank, Lauffer, John, Antesberger, Tim and Markovich, Voya, "Z-Axis Interconnections for Next Generation Packaging," **Advancing Microelectronics**, Vol. 38 No. 6, pp 12-19



PTH Core-to-Core Interconnect Using Sintered Conductive Pastes

Jim Haley, Ken Holcomb,
Michael Matthews, and Catherine Shearer
Ormet Circuits, Inc.

10080 Willow Creek Road, San Diego, CA, 92131

p: 858.831.0010, f:858.831.1437, web: ormetcircuits.com



High Performance Electronics Applications Require PCBs with High Aspect Ratio PTH

- Applications
 - High end computing
 - Semiconductor Test
 - Military
- PCB Design Trends
 - Increasing layer count
 - Increasing board thickness
 - Increased interconnect density
 - Signal integrity at high frequencies

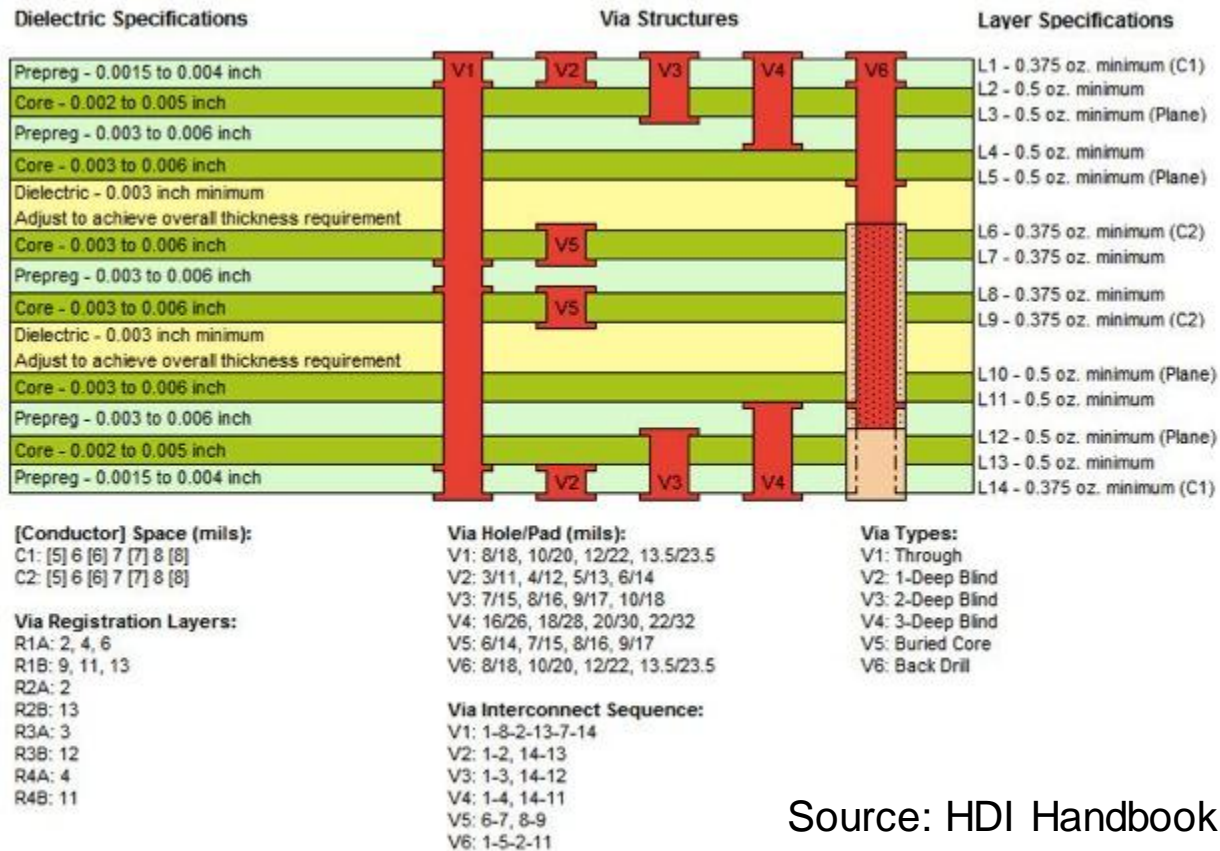


Traditional Methods for Producing High Performance PCBs Result in Low Yield

- Outer-layer hole drilling yield
 - High aspect ratio drilling
 - Back drilling
 - Barrel plating
- Blind and buried vias
 - Multiple laminations



Blind and Buried Vias and Back-Drilling Technologies are Used to Increase Interconnect Density



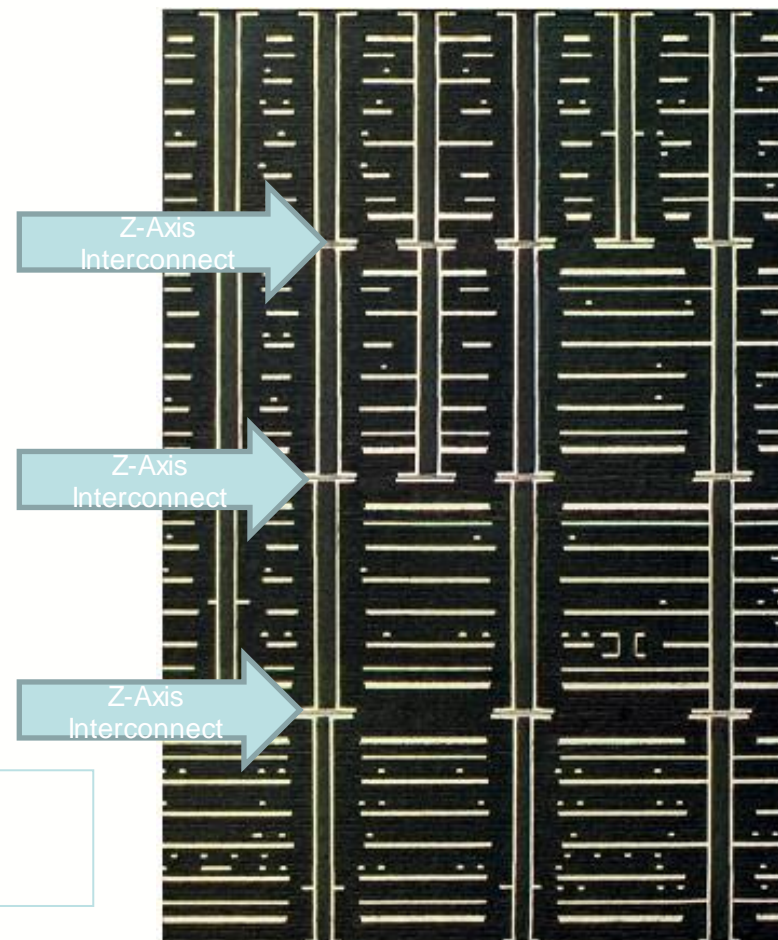
Source: HDI Handbook, Happy Holden



Creating High-Aspect-Ratio PTH Boards from Interconnected Subassemblies Using Sintering Pastes is an Attractive Solution

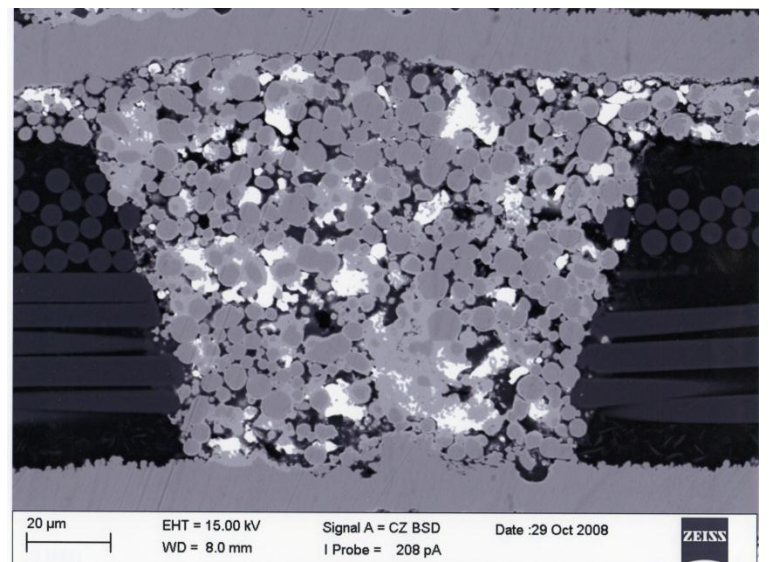
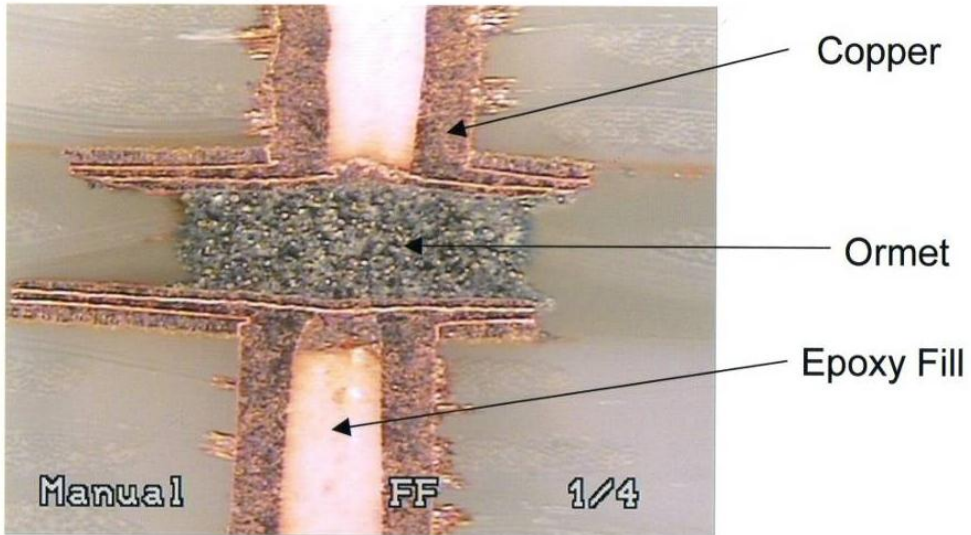
- **Interconnect core subassemblies using sintering conductive paste filled vias**
- **Thinner subassemblies**
 - Small holes with high-yield aspect ratios
 - Higher current density can be used for PTH = higher throughput
 - Higher yield

Sintering pastes enable low cost, high performance PCB structures.





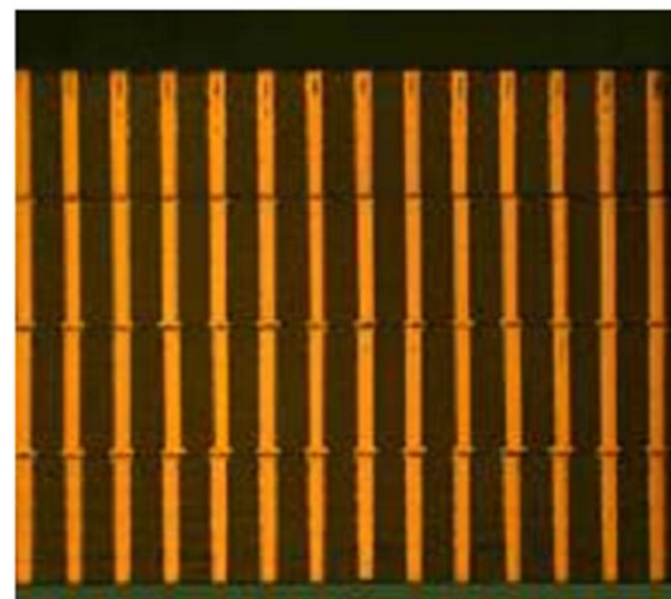
Sintering Paste Can Be Used to Electrically Interconnect Multilayer Cores During Lamination





Features of PCBs with Sintered Paste Core-to-Core Interconnects

- Break up high aspect ratio through holes
- Cores can be processed in parallel
- RoHS robust (reliable through lead-free reflow)
- Continuous metal joint – alloys to pads
- Substantially reduces plating time for high throughput
- High yields

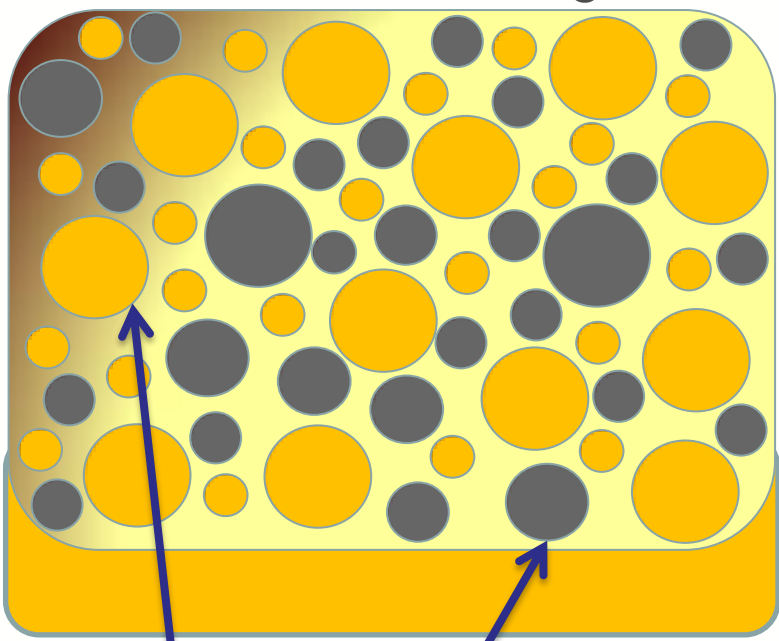


Source: DDI



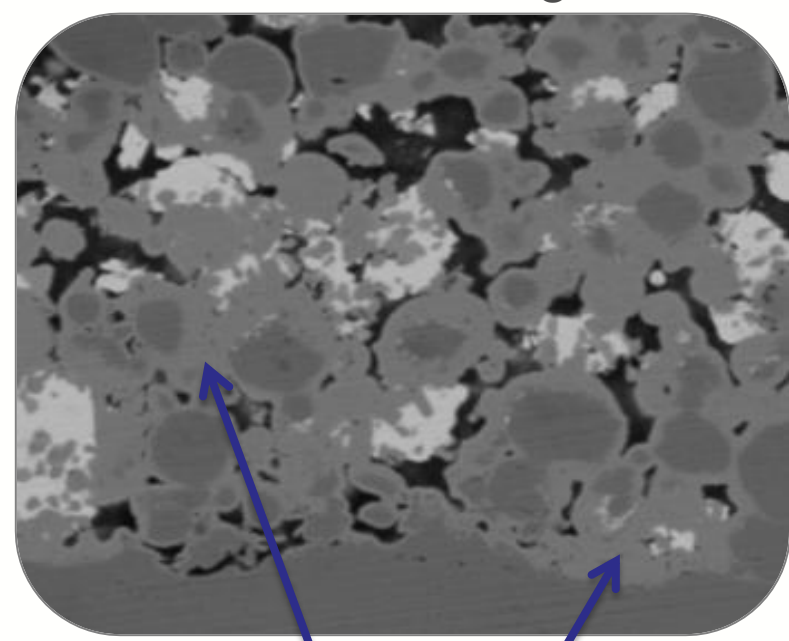
Sintering Pastes Metallurgically Bond to Copper Catch Pads

Before Sintering



Copper and alloy particles
in a liquid organic formulation

After Sintering



Sintered metal network

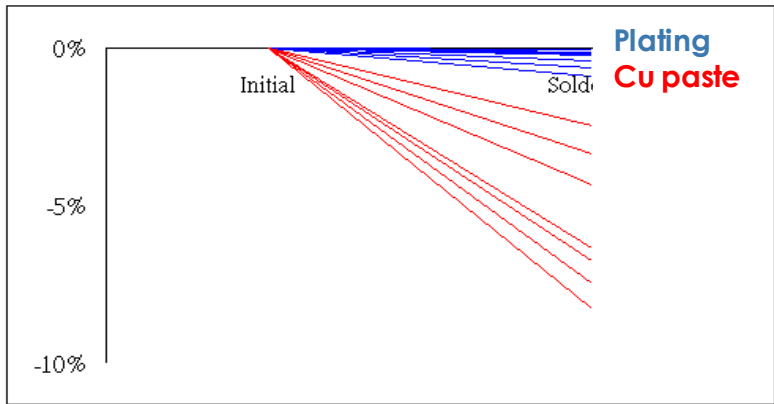
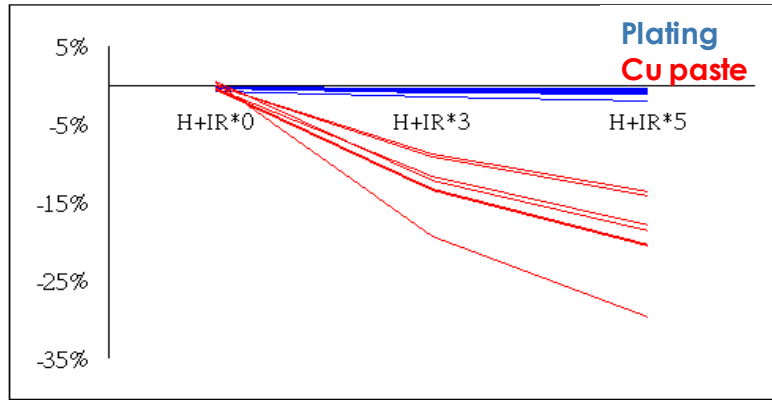
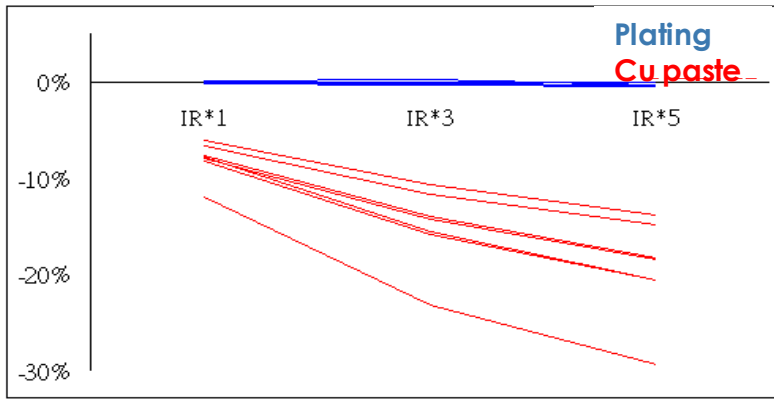
After lamination, sintered pastes will not remelt during solder reflow



Non-Sintering Copper Pastes Do Not Meet PCB Reliability Requirements

Plating: 4-2-4 structure

Cu paste = HDI process with non-sintering copper paste



Resistance Shift after Reliability Test

	Plating	Cu Paste
IR	0% - 0.4%	6% - 30%
Humidity + IR	0.2% - 2%	6% - 30%
288 Solder	0.2% - 1%	2.5% - 8%

Requirement is <10% shift after reliability test

Plating is an expensive alternative and non-sintering paste materials do not meet PCB reliability requirements.



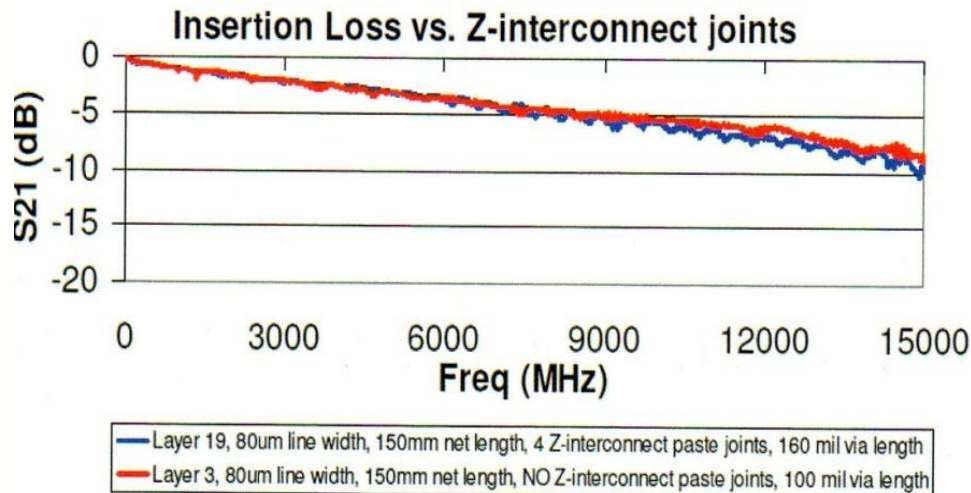
3rd Party Interconnect Reliability Data Shows Sintering Pastes Meet Requirements

Test	Test Conditions	Result
Thermal shock	-55°C—125°C, 500 cycles	Pass, Max resistance change 7.5%
Temp Cycling	0-100°C, 1000 cycles	Pass
Humidity + bias testing	85°C, 85RH%, 50V bias, hold 240 hrs	Pass
Humidity + Thermal Aging	85°C, 85RH%, hold 1000hrs	Pass, Max resistance change 4.0%
High Temperature Storage	150°C, hold 1000h	Pass, Max resistance change 7.0%
Solder reflow test	Reflow @ 260°C, 5cycles	No delamination
Electrical test	No shorts or opens	Pass

Sintering paste interconnects are well established in HDI and high layer PCB applications.



Sintering Pastes Have Robust Electrical Results in High Layer Count Applications



Insertion loss of nets with and without sintered interconnects demonstrate a negligible difference over a wide frequency range

Source: Das, Rabindra, Egitto, Frank, Lauffer, John, Antesberger, Tim and Markovich, Voya, "Z-Axis Interconnections for Next Generation Packaging," Advancing Microelectronics, Vol. 38 No. 6, pp. 12-19



Sintered Paste Interconnects can Lower Costs of High Layer Count PCBs

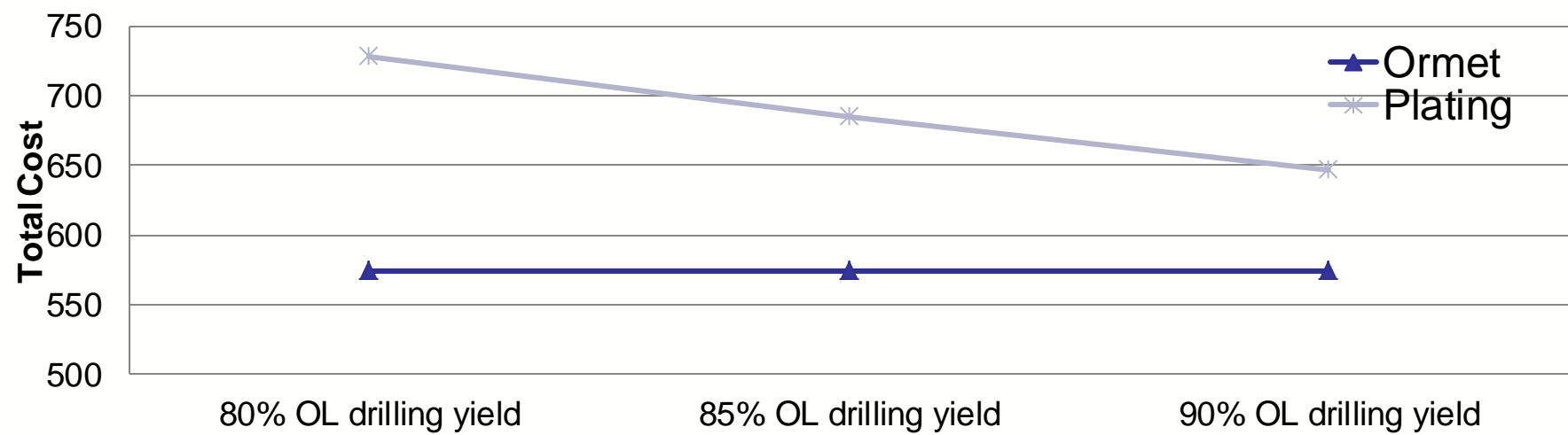
- Cost Savings
 - Elimination of outer-layer drilling
 - Higher yield from reduced aspect ratio plating
 - Higher yield from known good cores



Sintered Paste Vias Reduce the Cost of PCBs containing Blind / Buried Vias

Example: 50 layer PCB, design features in High End Server Boards
 Cost savings based on size of PTHs to be replaced by sintered paste.

10 Mil PTH Diameter- Total Cost



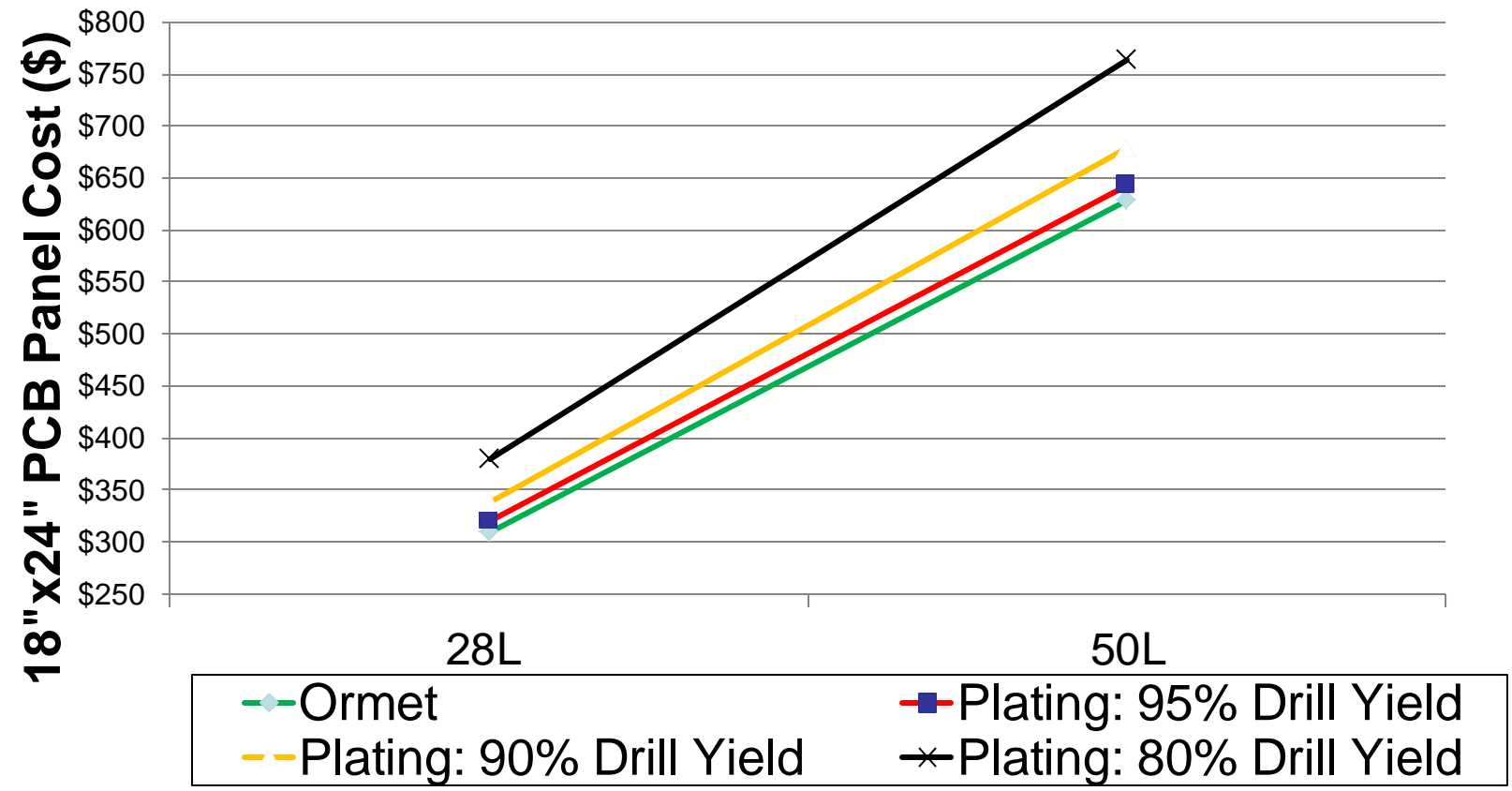
TOTALCOST	80% OL drilling yield	85% OL drilling yield	90% OL drilling yield	95% OL drilling yield
Sintered paste	574.14	574.14	574.14	574.14
Plating	727.25	684.47	646.45	612.42

Source: Savansys cost model



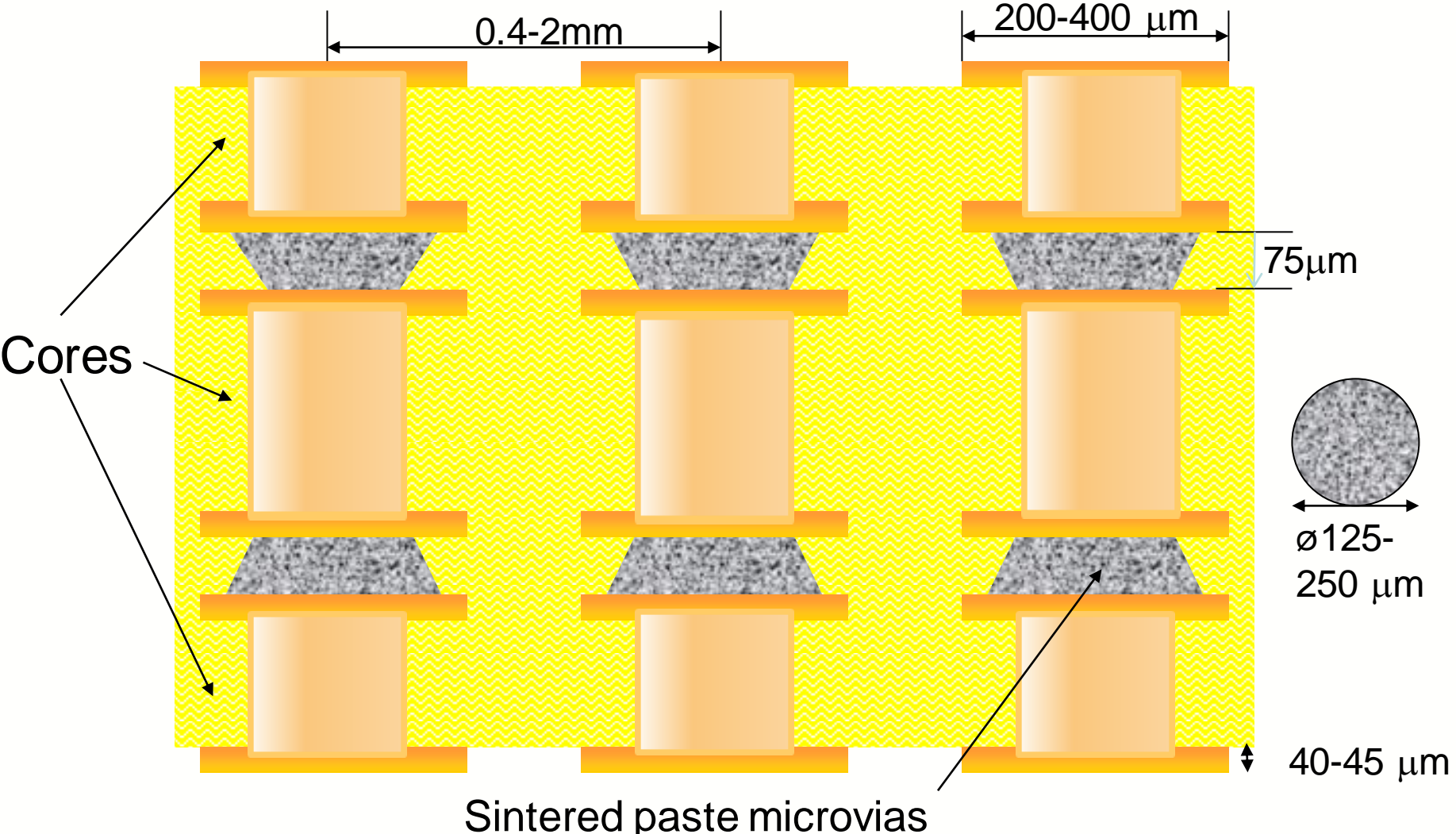
Sintered Paste Interconnects Can Reduce the Cost of PCBs Requiring Multiple Laminations

PCB Cost Comparison: 8 Mil Drill





Sintered Paste Vias Support a Wide Range of Interconnect Pitch



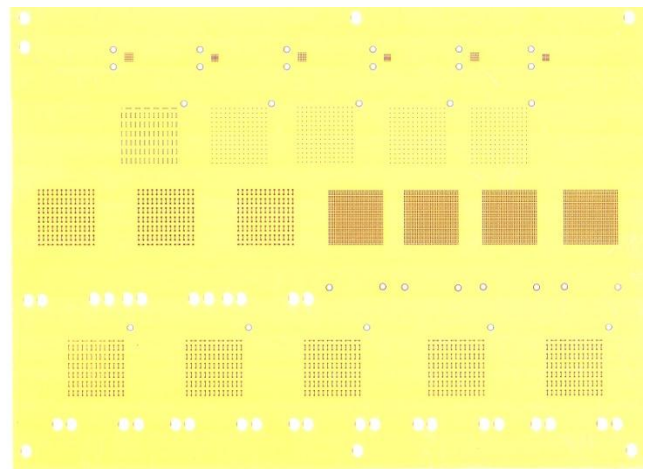


Implementing Paste Interconnects can be Accelerated by Using a Simple Test Platform

- **Standard daisy chain test board: EasyBoard**
 - Demonstration vehicle
 - Assembled at customer site on their equipment
 - Tested at customer site
- **Fabrication tutorial**
- **Application support services**
 - Design controls
 - Assistance with materials selection (e.g. prepregs)
 - Prototype fabrication
 - Reliability data



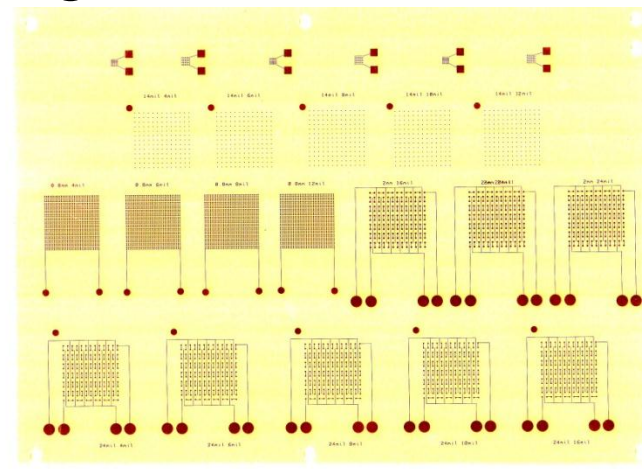
Using a Daisy Chain Test Vehicle to Optimize Interconnect Processes and Prepreg Selection



Layer 1 & 2

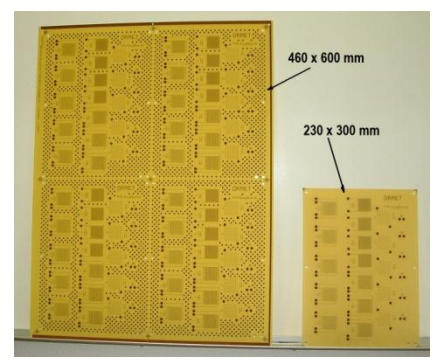


Paste-filled via



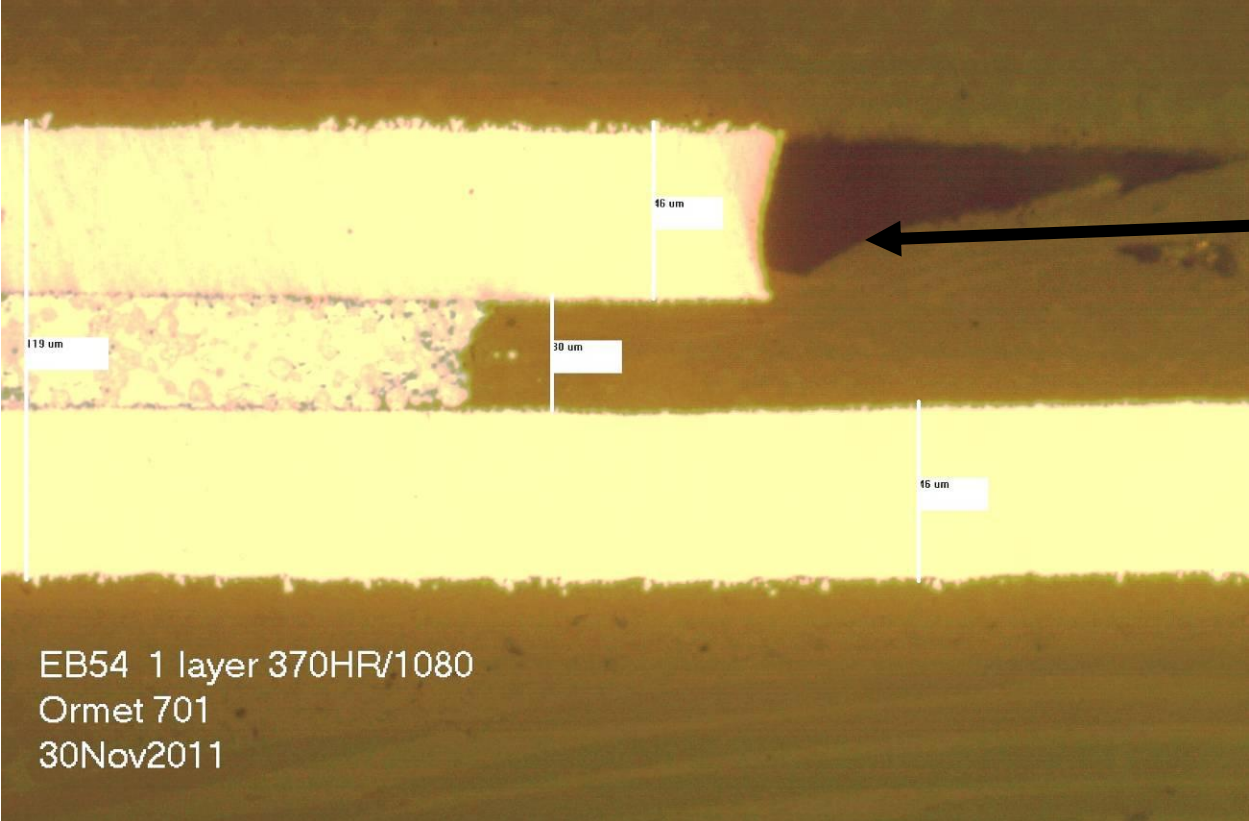
Layer 3 & 4

Standard 18" x 24" panel, 9" x 12" board →





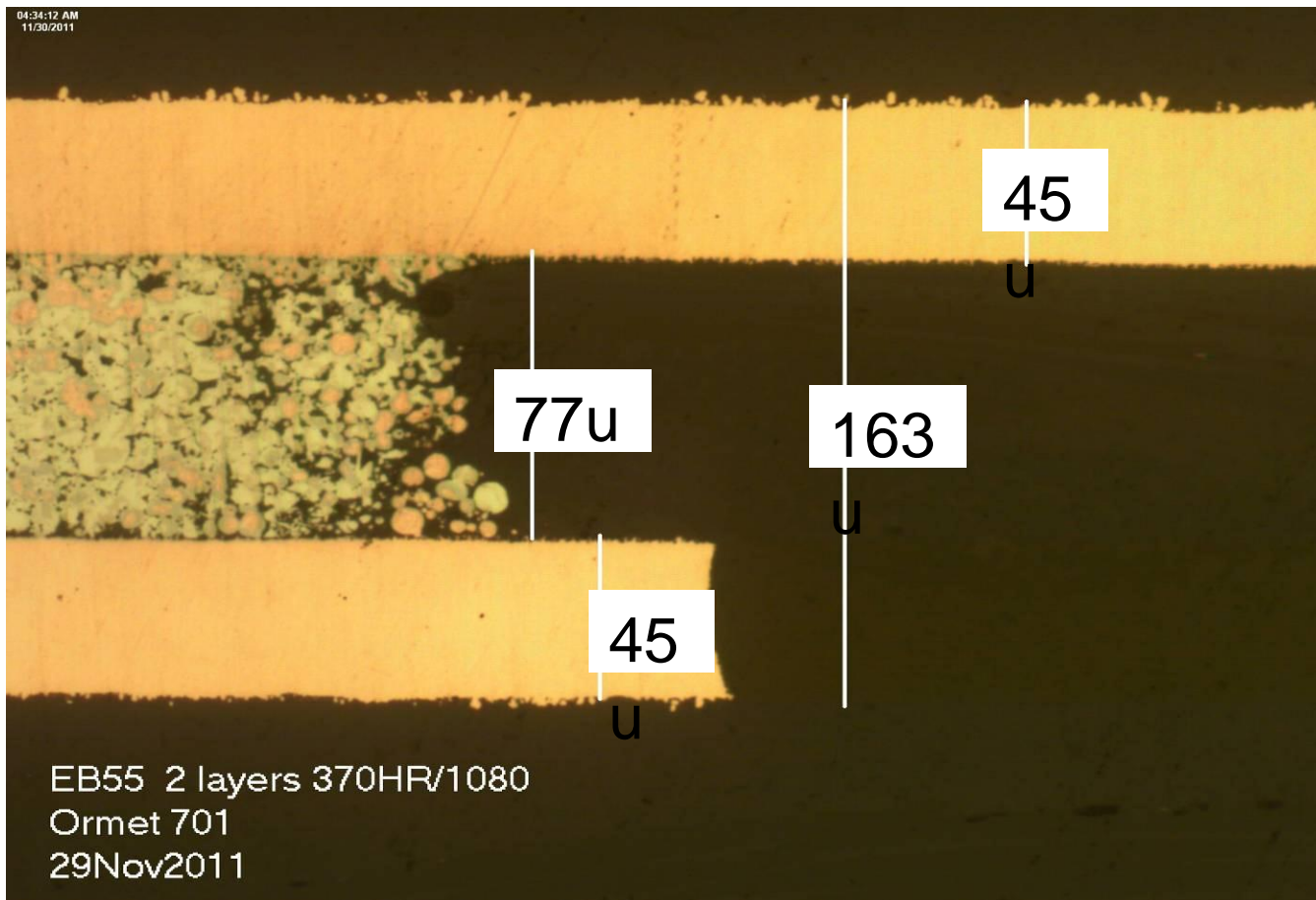
Prepreg Selection is Critical to Prevent Voids After Lamination



Lamination void

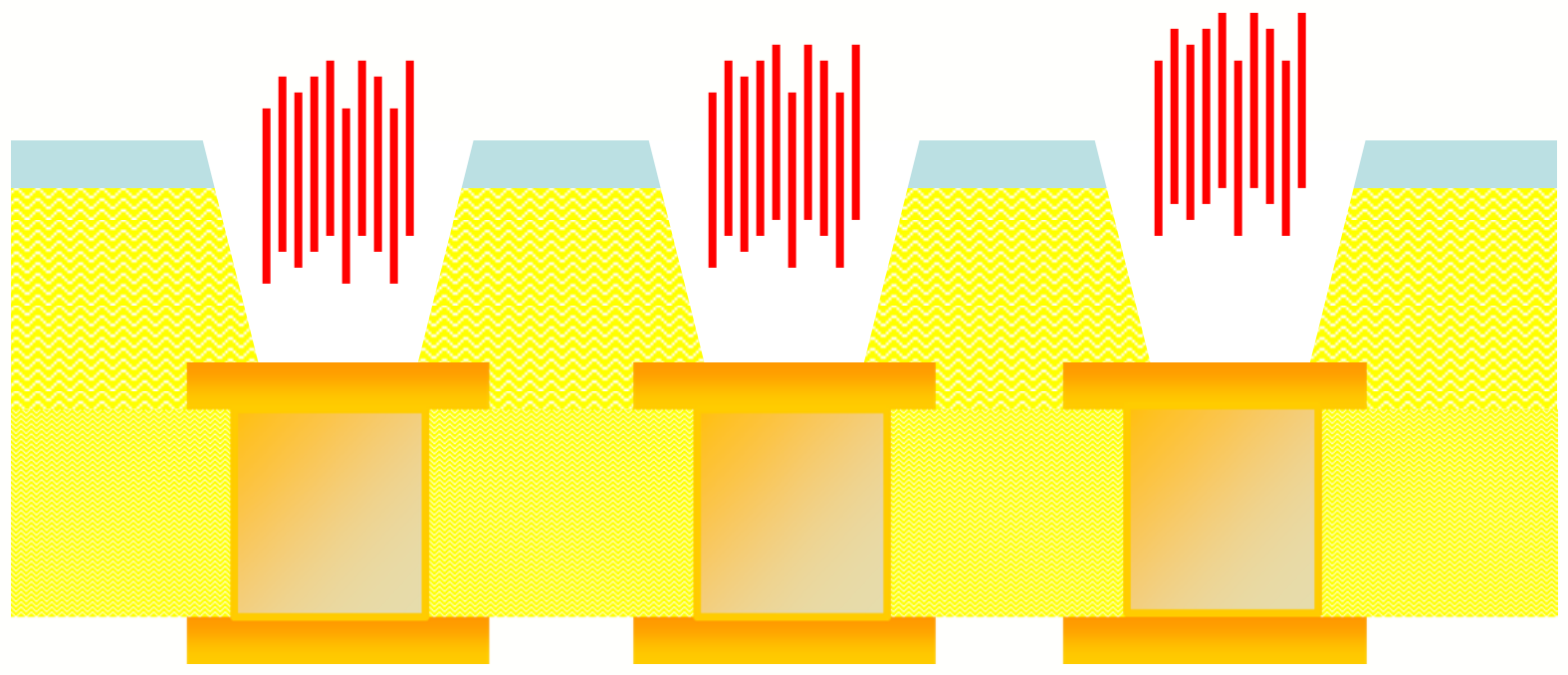


Correct Prepreg Selection Ensures Resin Fill Around Copper During Lamination





Laser Drilling of Prepregs is a Critical Process to Ensure Robust Paste Interconnection

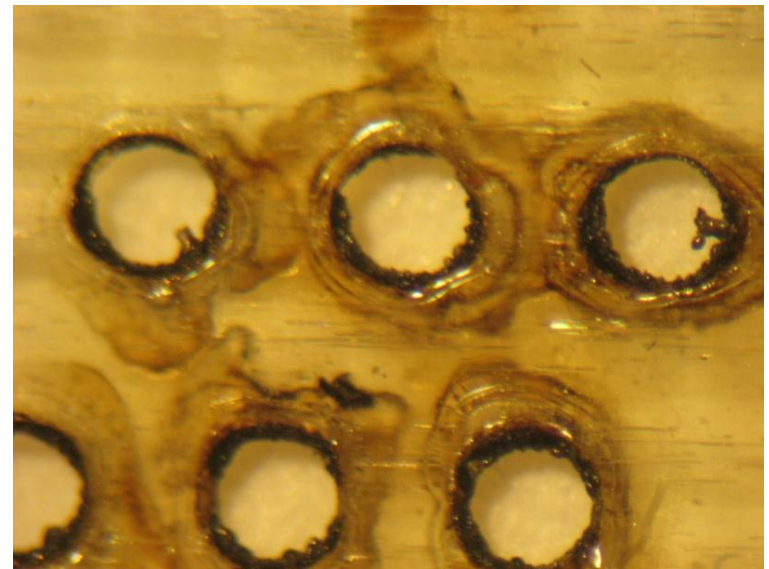
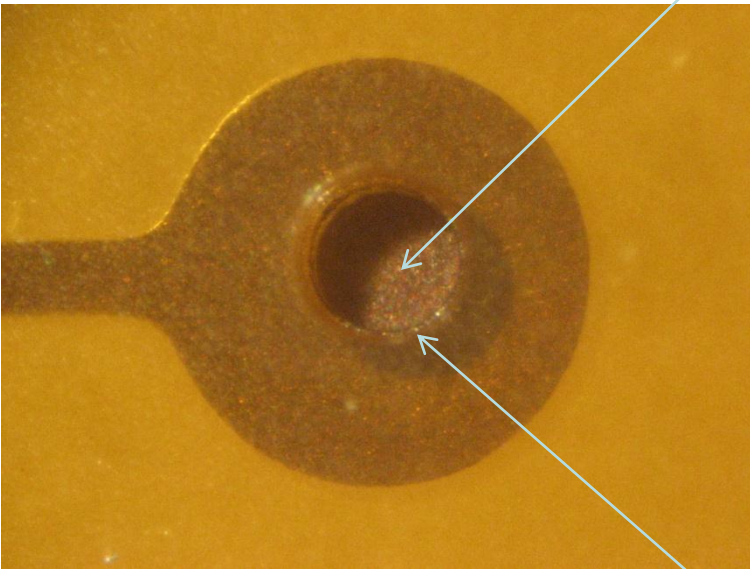


Laser ablation of prepreg requires very different parameters than forming vias in cured laminate



Debris-Free and Clean Copper at the Bottom of Vias is Key to Obtaining Robust Interconnects

Copper surface free of debris



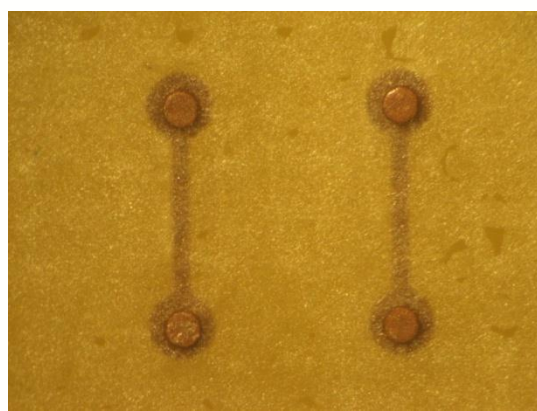
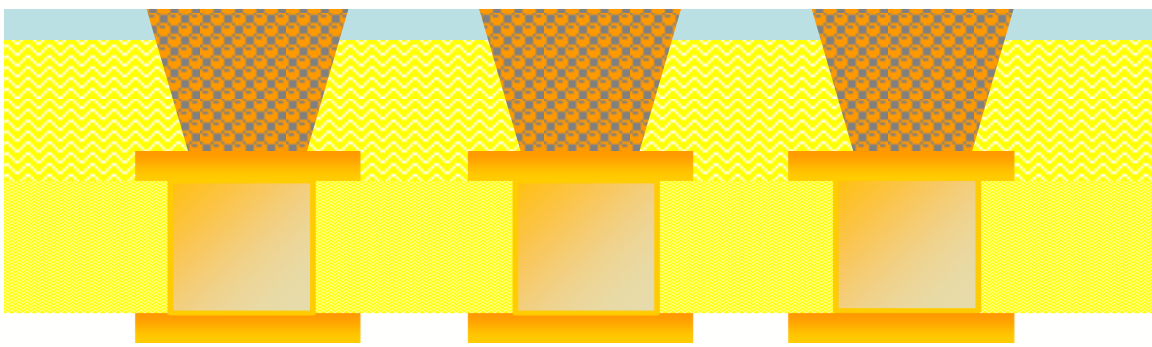
Good

Clean, planar hole edges.

Bad



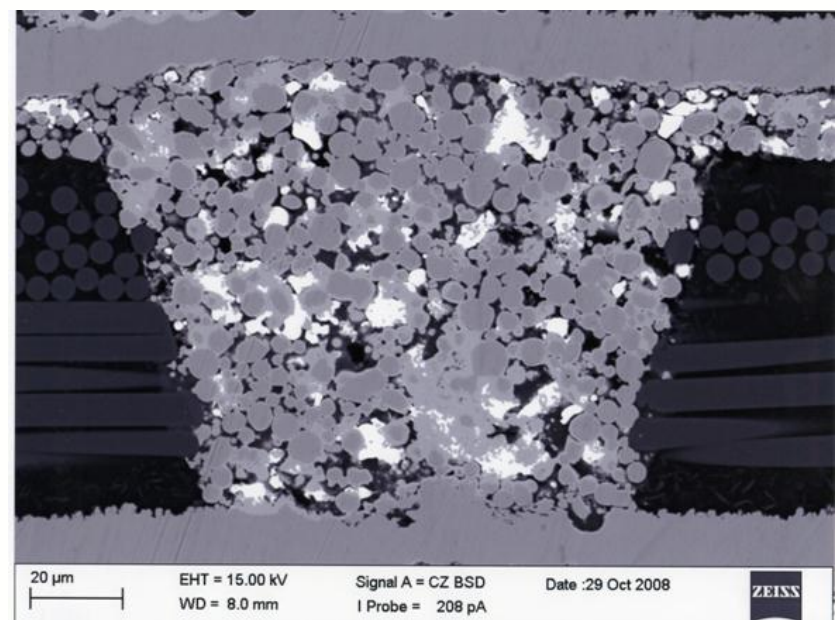
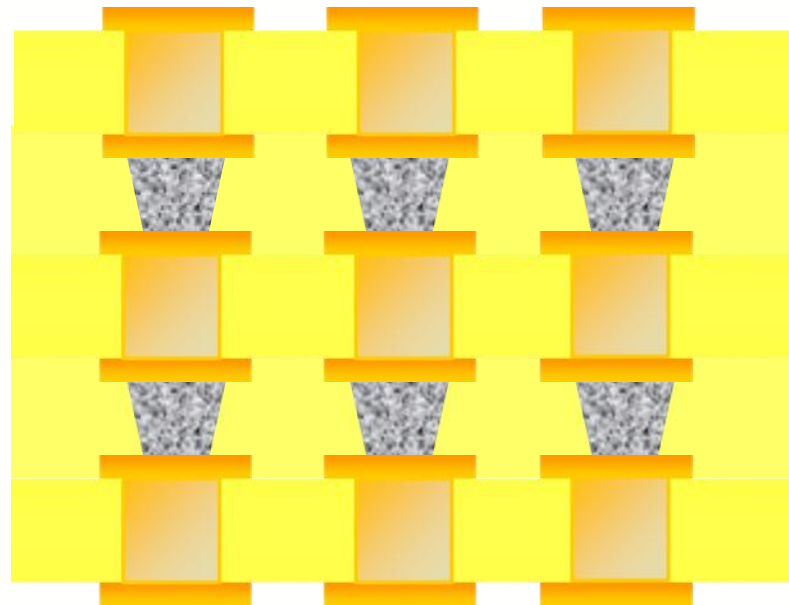
Filling Drilled Holes With Sintering Paste is Accomplished using Standard Printing Tools and Techniques



Industry standard stencil printing equipment is suitable for filling vias



Cores are Interconnected by Sintering Paste in Conventional Prepreg Lamination Process





Summary

- Electronic devices addressing high end computing, ATE, and military applications can require high layer count PCBs with blind and buried vias
- Increasing interconnect density requirements are pushing designs to utilize low yielding high-aspect-ratio PTHs
- High aspect ratio PTHs require complex drilling and plating processes, and frequently add cost to the PCB through low yield
- Breaking high-aspect-ratio through holes up by manufacturing and interconnecting subassembly 'cores' using paste interconnects is an alternative technology to plating high aspect ratio PTHs
- Sintering conductive pastes are a reliable interconnect solution for complex PCBs and enable lower costs, and higher yield compared to plating