Larry W. Burgess
MicroPak Laboratories, Inc.
West Linn, Oregon, USA

Paul D. Madden
Praegitzer Design
Hillsboro, Oregon, USA

Abstract
This paper will describe in detail, one proven method for the designed fabrication of blind vias that permit circuit boards to be reduced in layers, allow more circuit functionality and significant cost reduction as measured by functionality per circuit. A test design will be shown that clearly demonstrates the advantages of multi-depth Via-in-Pad laser drilled blind vias over traditional leading edge through via technology. In addition, this paper will describe the process for introducing this technology into production at any traditional multilayer circuit board fabricator or OEM designer. A circuit design will be followed from Net List through setting up the CAD tool, through CAM tooling and through the Fabrication process.

Key Words: Blind Via, Microvia, Laser Drilling, Via-in-Pad, Build Up

Introduction

Today's high density circuits present Designers, Fabricators and Assemblers with unprecedented challenges. With electronic component interconnections per square inch at an all time high, conventional multilayer design and fabrication techniques are approaching their practical limits.

This paper will compare the effects of laser generated microvias on the design and manufacture of a dense surface mount test board versus the results attained using conventional mechanically drilled through hole vias.

The advantage of laser drilled blind vias are in its unique ability to drill multi-depth blind microvia interconnections as part of the multilayer process without using build up technologies.

Design

Today's circuit board designs with over 100 interconnections per square inch are forcing designers to reduce trace and space geometry's shrink via sizes and increase board layer count in order to complete all of the interconnections.

Surface mount devices, by their nature require all interconnections to be made on the surface layer of a board. Obviously, all connections cannot be made on these surface layers requiring large numbers of via holes to enable signals to access inner routing layers where the majority of interconnections are made.

The problem arises when a high percentage of available via locations are used before all interconnections have been made. This phenomenon, known as “via starvation”, is an indication that even smaller geometry's will be needed to successfully complete the routing task.

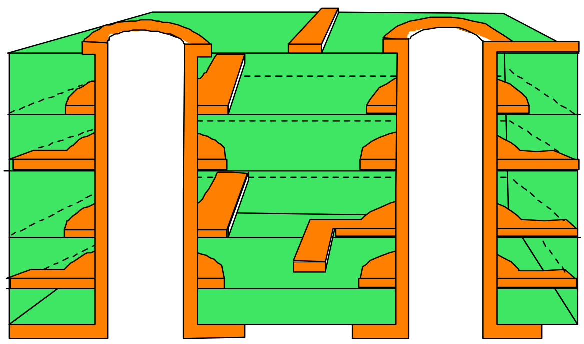

Fig. 1 Via Starvation

Reducing traces below 0.005” and drilling plated through holes smaller than 0.010” however, causes fabrication costs to skyrocket and dramatically reduces the number of board vendors capable of building such boards.

An alternate but emerging fabrication technology, laser drilled microvias, can offer greatly increased routing capability without requiring further decreases in feature or drill sizes.

In this test, a small, high density SMT board is designed using first, conventional mechanically drilled plated through vias, and second, using multi-depth laser generated blind vias.

The sample board used in this routing demonstration features two 121 pin ball grid arrays and six 48 pin SSOJ packages, 530 pins in all, placed on a routing area of just 2 square inches. Both routing operations were performed using the Spectra autorouter running on Pads Power-PCB CAD software.

In both cases, the conductor widths were set to .005” and spaces between conductors were also set to .005”. For the through hole via board, the via size was .025” pad size and .010” drill size, and for the laser via version, the via size was .010”pad and .006” drill.

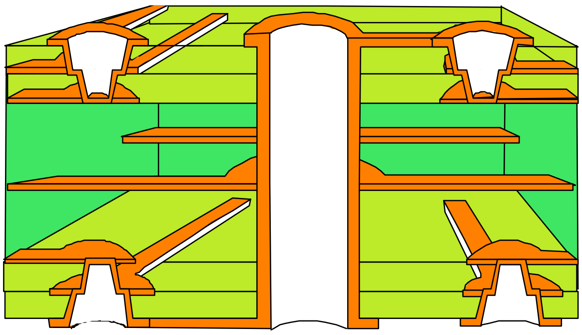

Fig. 2 Three Level Laser Drilled Blind Vias

In both cases the routing operations were scheduled for up to 100 routing passes followed by manufacturing optimization passes and a mitering pass to add 45 degree angles wherever possible. The autorouter automatically finishes whenever 100% of the interconnections have been completed. The results of these rout tests have not been manually altered or edited.

There are two major reasons for the big difference between the two sets of results. First and most obvious, the laser vias are much smaller. This alone accounts for some increased routing capability, but the fact that the laser vias may be placed within the pads of the SMT devices themselves that plays the greatest role in eliminating the via starvation problem.

From the design standpoint, one remaining point needs to be made, that is that the laser blind vias do not extend beyond the third layer, the through hole vias penetrate through all layers of a board. The advantage for the laser via in this context is with a circuit similar to this test circuit occupying the first three layers of a multilayer board; the corresponding layers on the opposite side of the board are non-perforated and available for mounting an additional circuit. The opposite side of the through via board, however, is already filled with via pads and vias and is largely unavailable for additional circuitry.

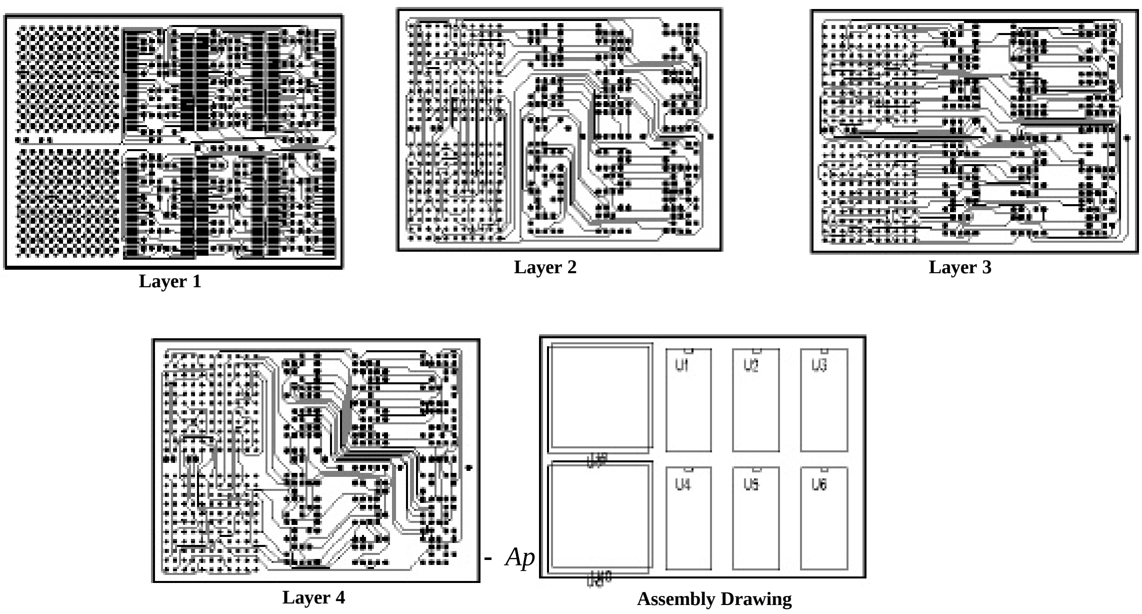

Fig. 3 Routing Layers Through Via Design

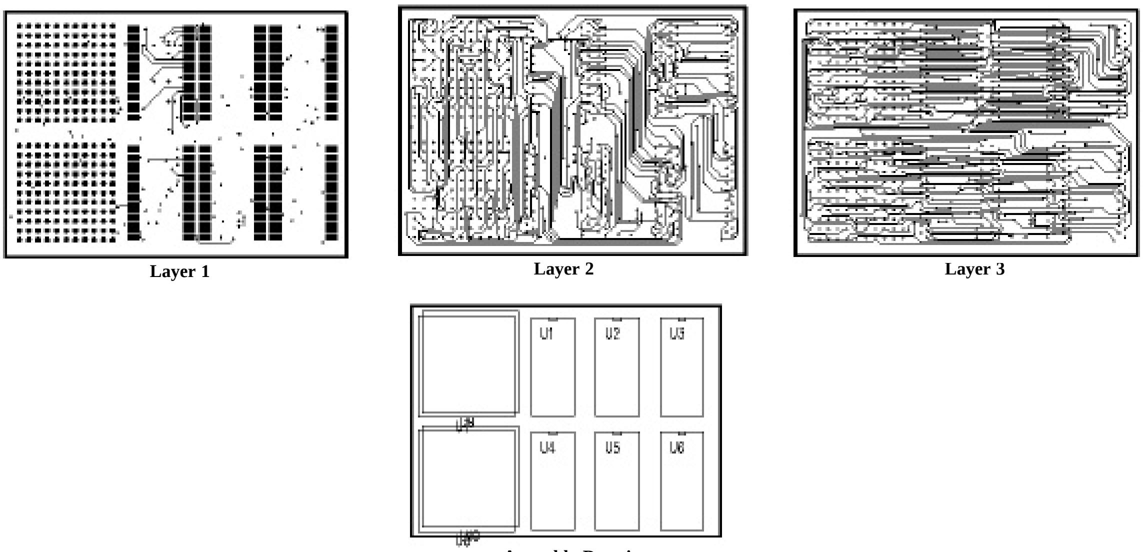

Fig. 4 Routing Layers Blind Via Design

Comparative Results

Here is what the route efforts produced. For the through hole via board, the autorouter required all 100 route passes and was successful in completing 91% of the boards connections on four routing layers. This turned into a classic via starvation situation. Possible routing channels remain on all layers, however, the few remaining via locations are not adequate to allow 100% completion on this board. This board took slightly over two hours routing time to get to this level of completion.

The results from the laser blind via board, however, are quite different. The autorouter was able to achieve 100% completion after only twelve routing passes. In addition, nearly all of the routing was condensed onto the two inner layers, with only a very minor amount of connection spilling over onto the top layer. This design could easily have been completed on only the two inner layers compared to the four required for the through hole version. This effort required only sixteen minutes of routing time.

Fabrication

The CAM operation to bring the circuit design into the fabrication area is rather straight forward for producing laser drilled blind vias, however, it is necessary to make sure the outer layer is fabricated with etched windows that act as a mask for rapid laser drilling.

There are two methods for producing these windows and they can be different as defined as:

1. Foil Lamination - where copper foil is used in the same fashion as Mass Lamination; or
2. Core Cap Lamination, where the outer layers are treated as if they were innerlayers and the windows are etched on the outer surface and the traditional circuits are etched on the inner surface.

The preferred method for building blind via circuits for interconnections down to a third layer is the Core Cap Lamination technique. With Core Cap Lamination the critical registration between the outer window and layer two donuts can be best aligned with innerlayer alignment techniques. The pad on layer three can be adjusted in size to match the alignment capabilities and material movement knowledge, of the fabricator. Figures 5 and 6 show the typical material layup for each method.

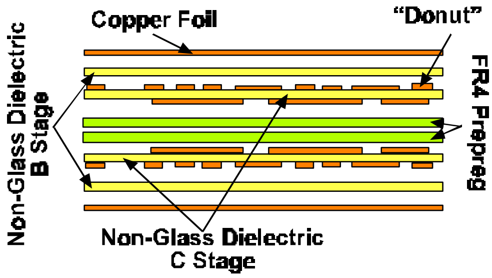

Fig. 5 Foil Lamination

These two methods for producing a conformal mask for drilling blind vias with a laser can work with any number of layers. The two diagrams within this paper are designed around a six layer board design. It is generally not necessary to design blind via boards for less than four layers, with the best value or payback for boards in the six and up layer count.

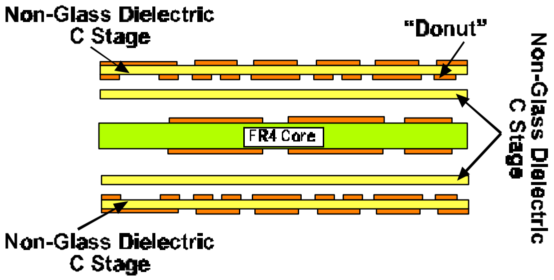

Fig. 6 Core Cap Lamination

A test design with both a Ball Grid Array (BGA) and a fine pitch Quad Flat Pack (QFP) is available for use in evaluating the fabrication process capability for the introduction of laser drilled blind vias to multi-depths. The design has all nets interconnected so that a simple continuity test can be preformed to check the entire circuit or to find an open area.

The interconnection scheme is defined in the manner displayed in the two drawings that follow. Both drawings depict only one side of the circuit board. Figure 7 shows the interconnecting scheme for the BGA and Figure 8 the interconnecting scheme for the QFP.

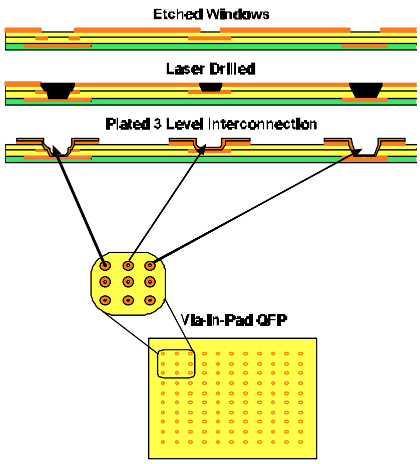

Fig. 7 BGA Multi-Level Interconnections

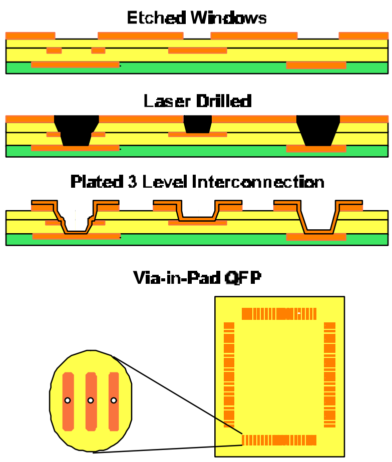

Fig. 8 QFP Multi-Level Interconnections

Conclusion

The advantages of multi-depth blind via interconnections saves significant time for the circuit board designer, easing his layout, and routing effort. The most significant savings however comes in the time saved, allowing more circuit boards to be designed by a given circuit board designer.

Component densities are only going to grow over the next few years, placing more demand on the circuit board to make the necessary interconnections. The circuit board designer will not be able to meet the increasing interconnect demand with out the use of blind via-in-pad technology.

All of the elements now exist including the design CAD tools, knowledgeable circuit board designers, fabrication processes and now production lasers to meet the current and upcoming rapidly advancing dense interconnect demand.

Circuit board fabricators have recently become open to developing new technologies and are testing new advanced dielectric materials to help the up coming microvia technology movement.

Biography

Larry W. Burgess has over thirty years experience in the interconnect packaging disciplines. He holds a Bachelor's Degree in chemistry and has held Management and Engineering Management positions at fortune 100 electronic companies. He is President and Chief Technical Officer at MicroPak Laboratories, Inc., where he has licensed technology to Sandia National Laboratories. MicroPak has recently formed a joint venture with Pluritec. Currently Mr. Burgess is preparing to open a series of Laser Drilling Centers in North America to support the upcoming demand for laser drilled blind microvias.

Paul D. Madden has over twenty-five years experience in the circuit board deign and electronics packaging industry. At Loyola Marymount University Paul's major course of study was Psychology. In ten years at Tektronix Inc., he worked as a PCB Designer, PCB Design Group Manager, and CAD Development Group manager. In 1985 Paul became a cofounder and Design Group Manager at Interconnect Technology Inc., where he developed techniques to optimize the use of laser blind and buried vias in high density PCB's. In 1989 Paul became a partner and Design Group Manager at PCB West, where he remains with the merger into Praegitzer Design.