Presented in the ECWC 10 Conference at IPC Printed
Circuits Expo®, SMEMA Council APEX® and
Designers Summit 05

Printed Circuit Board Architecture for the Use of Optical Interconnection of Components

James Howard
WUS Printed Circuits
Carson City, NV

Greg Lucas
Circuit Solutions Corporation
Fremont, CA

A Brief Review of the General Architecture of Optical PCBs

Surface Based Optical Transmission:

The current technology includes transmission of between optical units (typically modulated laser sender and receiver units) and fiber optic cables or flexible kapton fiber optic cables, which are all relatively familiar.

At present many of the optical and digital devices may be mounted on opposite sides of the PCB and all connections are on the surface of the PCB. (Figure 1)

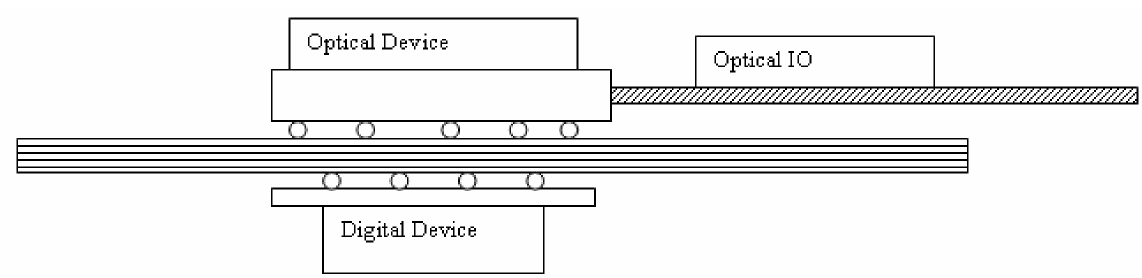


Figure 1 – Typical Device Placement

The limitations for this technology for on board transmission of signals are size, surface space requirements, and limited channels available.

The interposer model (Figure 2) is based on the requirement for reducing surface space and communication time between elements. The technology and limitations are not significantly different from the current systems deployed.

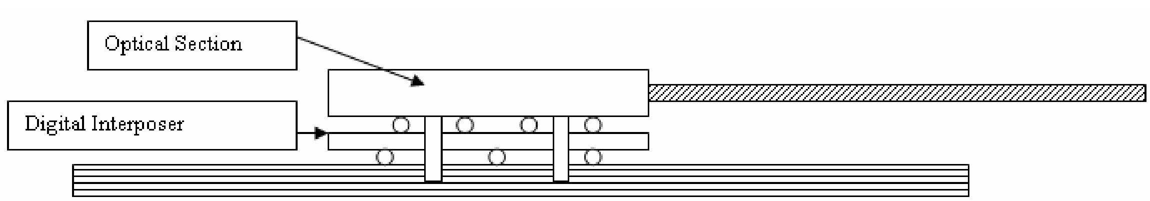


Figure 2 - Interposer Digital System

The primary change in this evolution of technology is the incorporation of the optical and digital sections in a chip and the inclusion of transmission elements in the PCB (Figure 3). A number of new techniques and technologies are available for IP development in this model. These will be correspondent to additional changes in the PCB process methods to incorporate these developments.

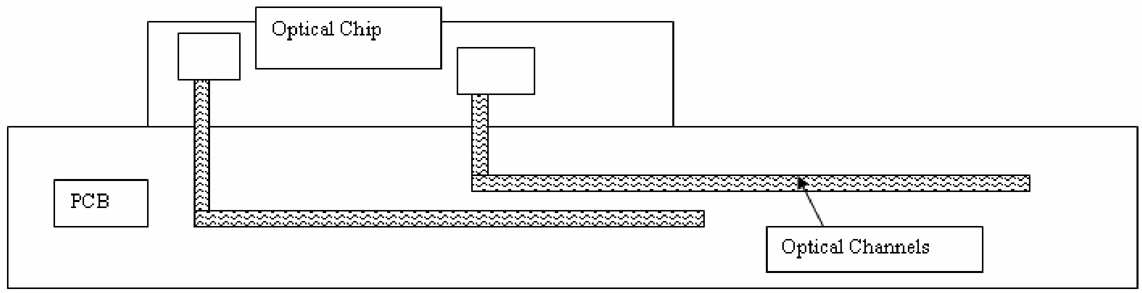


Figure 3 - Digital Optical Device On Board

Printed circuit boards have previously been formed as laminated structures and have been populated with devices such as integrated circuits and the supporting elements, which may be used in a wide variety of electronic applications. More recently optical devices have been added to the printed circuit boards to transmit optical signals to locations remote from the printed circuit board and to communicate with other devices located on the same printed circuit board. Typically, these optical devices and interconnecting transmission medium, such as an optical fiber, have been located on the surface of the printed circuit board because of the necessity to make contact with the integrated circuit, or other device, and the physical difficulty in placing an optical transmission element, such as an optical fiber, within the printed circuit board.

Substantial effort has been expended to design an appropriate optical interconnection scheme between the electrical devices mounted on a printed circuit board, due to the faster speed of transmission of an optical signal in comparison to the speed of an electrical signal within a conductor, and the elimination of such electrical effects such as the generation of electromagnetic interference or EMI within the conductor as the current passes through.

Generally an optical transmission design includes the elements of a light generation device such as a laser diode which receives an electrical signal from and may be formed within or on an active device such as an integrated circuit. A light transmission element such as an optical fiber then carries the light to a distant receiving element such as a photodiode, which then converts the light signal into an electrical signal that may then be further processed.

One novel approach to this design was set forth by Muhammed Shahid of Lucent Technologies in U.S. Patent 6,185,348 in which a method for assembling a multifiber optical connection circuit was disclosed which could be used to form simple coverings for the group of optical fibers contained within. The Shahid method does not contemplate a method of integrating this multifiber optical ribbon within a fabricated printed circuit board in which holes must be drilled, circuits must be routed and following substantial chemical and physical processing must be laminated or formed into a final fabricated unit.

Additionally other methods of encompassing optical fibers within flexible materials for bundling of circuit elements have provided a better external interconnect capability.

Other techniques have also been considered such as utilizing a refractory lens or mirror to conduct light from a surface device to a channel or optical fiber within the PCB. These techniques were also not found to solve the problem that the present concept is intended to solve. The inherent instability of the position of such devices as a lens or mirror within the PCB or affixed between the PCB and a surface device would render such a solution impractical and unreliable. Additionally, the additional electromagnetic interference or EMI from the high-speed optical devices might provide a significant degradation to the operation of the assembled device.

Summary of the Concept

Accordingly, it is an object of the present concept to provide an effective and workable solution to the problem of interconnecting devices on a PCB by means of optical channels within the printed circuit board.

The concept of the present concept is to connect the surface device in the Z direction or into the PCB with a standard electrical connection, such as a blind via, and place the laser diode or other optical generation device on the surface of a capacitive power/ground sandwich layer. The laser diode would then generate optical signals which would be transmitted through an optical fiber or other optically conductive material, the fiber or conductive material all lying on the same plane, to a photodiode or other receiving device which would convert the optical signal into electrical signal for transportation in copper conductive through holes or other conductors in the Z, X or Y directions within the PCB. By designing the PCB in this fashion the problem of bending light to be channeled inside the printed circuit board is solved as the electrical signal proceeds a very short distance into the PCB. The optical generation transmission and reception devices are all formed or mounted on the same plane, eliminating the problem encountered in the mechanical changes in size and shape that a PCB undergoes when it is exposed to heat or other stress.

It is the object of the present concept to place the capacitive power/ground plane as near to the surface components as is practicable to shorten the path of electrical flow. It is contemplated in the current concept that the plated through holes or PTHs from the surface devices may be formed as blind vias with short length, low via inductance and inherently lower EMI generation. In typical applications these blind vias may be .005 inches or shorter to enable an electrical connection to the power ground plane upon which the optical elements will be formed. In contrast the optical portion of the circuit (as an example) may be 12 inches in length, with a ratio of 10:12,000 electrical to optical length of connection, making the time of signal travel in the electrical signal portion irrelevant.

It is more specifically the object of the current concept to form channels in the conductive coating of the power ground layer for the placement of optical fibers or the formation of channels that may be filled with a optically transmissive material in a subsequent process. These optically conductive fibers or channels may additionally receive coating to prevent the unintentional transmission of light to other elements. This coating may take many forms including plating, polymers, or epoxy materials.

It is an additional object of the concept to provide a capacitive base sandwich for the operation of the electrical devices that form the optical elements to suppress EMI from those elements. Acting as a high speed capacitive buss element the power/ground layer will capacitivly couple some of the high speed electrical noise, reducing the electrical interference that may affect the devices on the PCB or be radiated as EMI.

An optical printed circuit board with electrical connections in the Z axis and optical connections in the X and Y axis according to the present concept is described in greater detail below. For the purposes of this embodiment, the Z axis of the printed circuit board is the direction through the printed circuit board layers that the via holes pass through and is typically very thin, as an example thickness ranges from .002 inches to .5 inches are not uncommon. The X and Y dimensions are the dimensions of the surface of the printed circuit board or “PCB” which are typically the approximate size of the internal layers as well. Many different embodiments may be incorporating the same PCB features described. Types of devices used, layer count, line width, layer thickness, types of holes used or methods of manufacture are all examples of variations that may occur to the PCB without changing the basic application of the current concept.

Referring to Figure 1 the PCB shown has an internal capacitor layer, preferably a power ground sandwich with a thin dielectric core. Formed upon this core are the devices used to generate and receive optical information, usually as high speed pulses. These devices may be existing devices such as laser, LED or newer organo-metallic or other devices. They may include known devices as photo detectors for receiving elements or may be composed of other materials that will perform the same task. These devices will transmit and receive electrical signals from the surface devices, which are devices such as BGA, CGA or other such digital or analog processing devices. An advantage of the present concept is the reduction of electrical noise generated by the optical devices, as well as the general reduction of electrical noise generated by the digital portion of the PCB as noted in US Patents 5,079,069 and 5,155,655 both to Howard et al. Additionally, the preferred embodiment views the mounting of these devices on the conductive layer, most preferably copper as a good method of removing heat from the devices due to the large amount of a very good heat conductive material in the copper plane.

The optical transmission element may be formed through processing through several different methods or may be a premade optical fiber that may be incorporated on the layer during the manufacturing of the PCB. Some of the manufacturing methods may include etching paths in the copper foil on the surface of the capacitor layer, then filling the paths with an optically clear material and curing that material to provide stability. Optical fibers may be secured in the etched paths or on the surface of the copper or other conductive material to accomplish the same task. The etched paths may be formed as open channels with no filler material, or filled with gas or vacuum to promote optical clarity. It may be advantageous in the most preferred embodiment to only partially etch through the conductive surface to provide an optically reflective channel on three sides, which may be fully encapsulated with a reflective surface on the top surface to completely enclose the light transmissive channel. It is fully contemplated within the preferred embodiment that a highly internal surface is preferable within the light transmissive channels, which are formed in or on the conductive surface. All these methods are contemplated within the current concept.

The construction method may include the method of using resin coated copper or “RCC” to facilitate the use of laser or other processing to form the via holes in the most preferred method. Additionally, Figure 4 shows a common connection method of assembly using solder balls to connect device to pad.

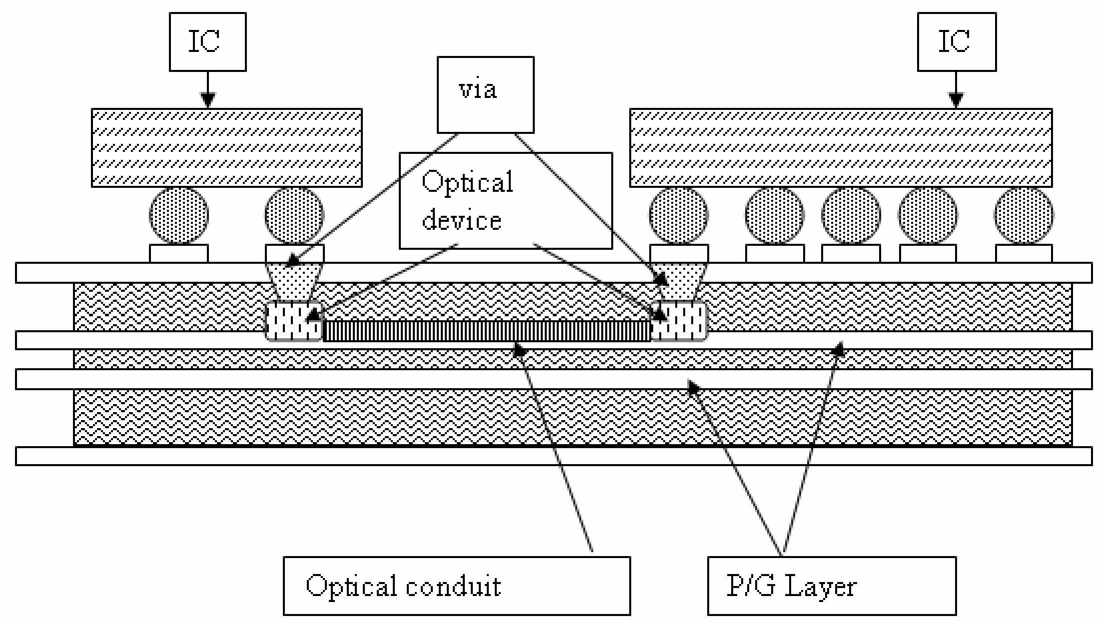


Figure 4 – Solder Ball Attachment

A blind via connects the surface pad electrically to the optical transmission and reception elements. Utilizing the shortest and most inductance free method of connection between the surface device and the optical elements is considered to be the most preferred embodiment of the current concept. It is one preferred embodiment of the current concept to use conductive polymer as a method of forming the conductive via hole. This method will allow greater process flexibility in the types of interconnection methods that may be employed in the final method.

Referring now to Figure 2, this figure shows a top view of the PCB in Figure 1 as with surface device shown as top view, solder ball, optical generation or reception device and optical transmission element. As is shown in Figure 5, optical transmission elements may move freely in the X and Y axis of the PCB but does not travel in the Z axis. The key reasons are the impractical nature of 90 degree or greater bending of light within a very unstable polymer structure such as a PCB, and the very short distances that electrical signals must travel in the Z axis in the preferred embodiment in comparison to the travel of light transmission channels in the X and Y dimensions.

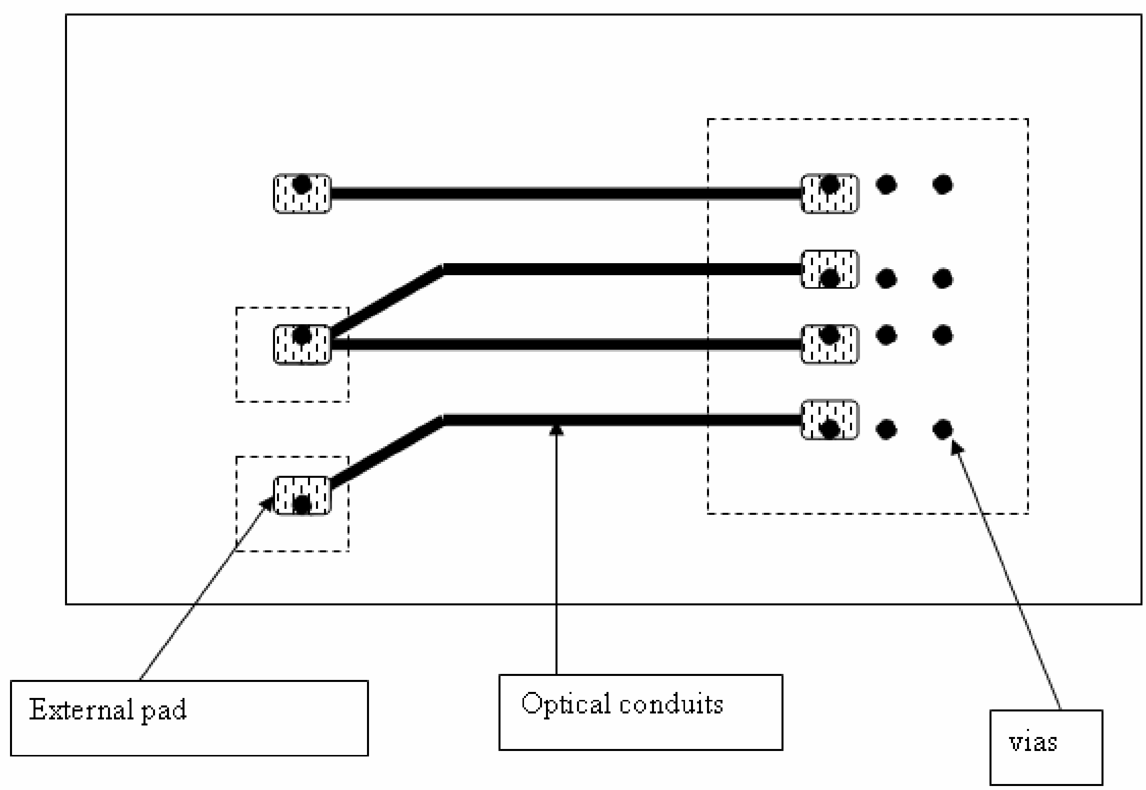


Figure 5 – Optical Transmission Element Routing Paths

In Summary

The concept revealed is a simple architecture for an optical printed circuit board, which permits manufacturing with the materials normally used in printed circuit board fabrication. This concept is that the very short distances of Z travel within a printed circuit board can be best traveled electrically, allowing the surface devices and general layout to be nearly standard and with inexpensive devices. The internal transmission of optical signals is done only within a plane, making the manufacturing much easier and the reliability as good as other embedded device PCBs. This should enable a much less expensive optical PCB.

Printed Circuit Board Architecture for the use

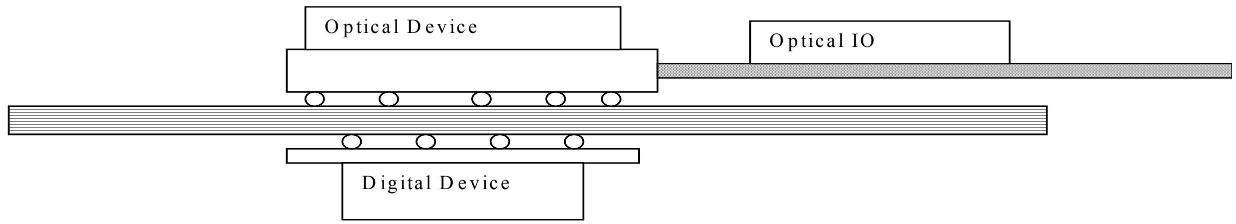
of Optical Interconnection of Components

By
James Howard, WUS Printed Circuits
and Greg Lucas

Printed Circuit Board Architecture for the use of Optical Interconnection of Components

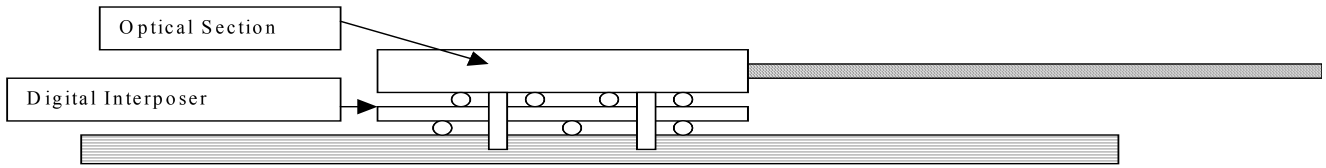
By
James Howard, WUS Printed Circuits and Greg Lucas

A Brief Review of the General Architecture of Optical PCBs
Surface Based Optical Transmission:
The current technology includes transmission of between optical units (typically modulated laser sender and receiver units) and fiber optic cables or flexible kapton fiber optic cables,which are all relatively familiar.
At present many of the optical and digital devices may be mounted on opposite sides of the PCB and all connections are on the surface of the PCB.



The limitations for this technology for on board transmission of signals are size, surface space requirements, and limited channels available.

Interposer Digital System:



The interposer model is based on the requirement for reducing surface space and communication time between elements. The technology and limitations are not significantly different from the current systems deployed..

Digital Optical Device On Board:

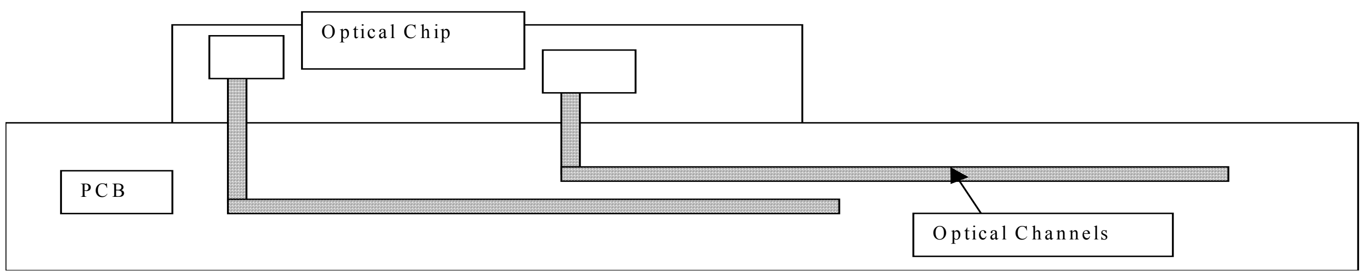


Figure 1: Side View

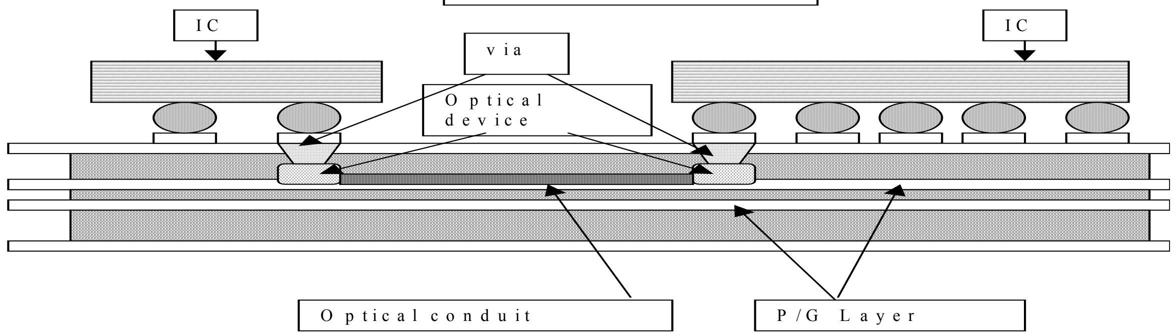
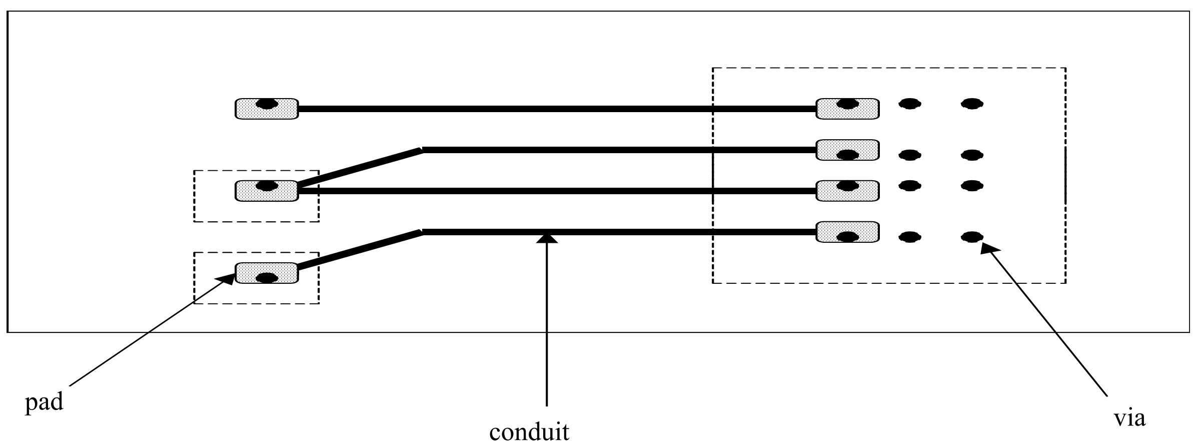


Figure 2: Top View



Electrical Concept

- It is the object of the present concept to place the capacitive power/ground plane as near to the surface components as is practicable to shorten the path of electrical flow. It is contemplated in the current concept that the plated through holes or PTHs from the surface devices may be formed as blind vias with short length, low

Optical Concept

- via inductance and inherently lower EMI an plane upon which the optical elements will be formed. In contrast the optical portion of the circuit (as an example) may be 12 inches in length, with a ratio of 10:12,000 electrical to optical length of connection, making the time of signal travel in the electrical signal portion irrelevant.

Electrical to Optic Ratio