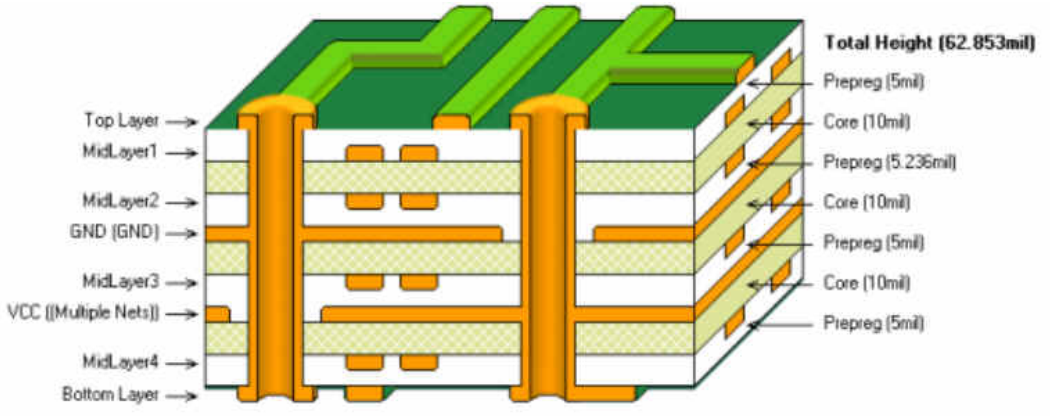
Introduction



An important component of the supplier's contribution to a good product is engineering support in the form of advice on components selection such as chipsets, PCB, housing and package, manufacturability advice such as power screening and SOP for testing, and other technical supports. Good supports in every area can make the difference between a product that is profitable and reliable and one that is not!

In this forum paper, we will talk about PCB (Printed Circuit Board), which usually has been viewed as a somewhat trivial part of electronics by the users of PCBs and good performance has not been well rewarded or recognized. As a result, less capable PCB makers have been awarded contracts while the more capable are driven out of business. Unfortunately, it is rare that the lowest price PCB bid turns out to be the lowest cost!

For GoMax, it is vital to choose PCB fabricators who can keep the risk of failed prototype and production PCBs to the absolute minimum. The pressure comes from the fact that a failed PCB costs a company, not only its cost, but also the cost of all the components mounted on it as well as a large quantity of troubleshooting labor and the time to market. Even worse, it will cost the faiths from our clients.

At GoMax, the objective of the PCB supplier selection process is to choose PCB suppliers that are capable of reliably supplying prototype and production quantity PCBs. In addition, it is also important to choose suppliers capable of providing vital front-end design engineering support in areas such as material selection, design for manufacturability, lamination and plating! The knowledge is gold!

There are a number of ways that a PCB can turn out bad. Vias and plated through-holes fail creating open circuits. Solder joints fail as a result of poor surface finishes. Shorts can be present in the PCB when it is new or grow due to metal migration. Opens can be present in a PCB when it is new or occur due to metal

fatigue. Impedance can be the wrong vale for variety of reasons. Those failures are the most insidious because they usually do not show up until PCB has been assembled or sold out. Inferior PCBs as above directly impact the durability of mass production products and therefore the image of the very product.

Sadly, the lack of visibility of these kinds of failure leads to the assumption that price is the only selection criteria for a PCB and therefore product. Normally, if a supplier quotes a price noticeably lower than its competitors, odds are it is because of shortcuts, not simply superior skills.

PCBs/Manufacturing Process

In this section, we simply presents a simple idea about PCB manufacturing process, For a typical rigid multilayer PCB, we can in general consider the process steps as:

- PCB data acquisition
- Preparation of PCB laminate
- Inner layer image transfer
- Laminate layers
- Drilling and cleaning holes
- Make holes conductive
- Outer layer image transfer
- Surface finish
- Final fabrication

Step 1: PCB data acquisition

In this step, files transferred from PCB design house to PCB manufacturing facility include Gerber files, drill files, profile routing files, fabrication drawings.

Step 2: Preparation of PCB Laminate

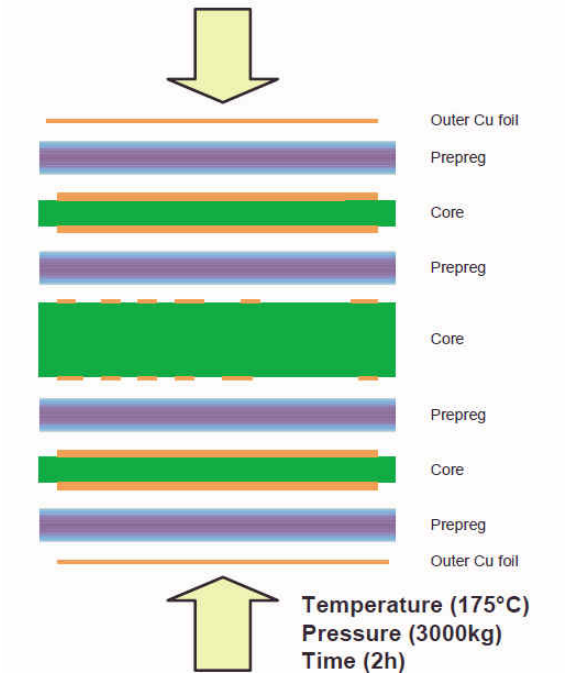
Dielectric material, copper foil, core material, coat dielectric material with resin & harden.

Step3: Inner layer image transfer(photo-lighography)

Coat copper foils with photoresist, place phototool and expose to light, etching!

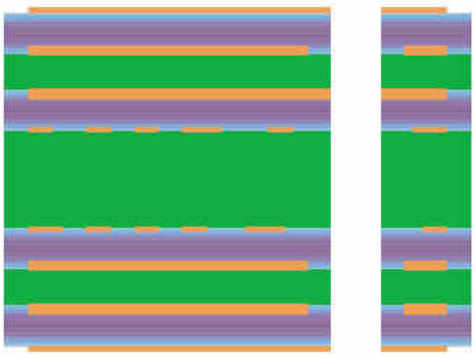
Step4: Lamination

Cores are pinned in a stack with sheets of prepreg separating the copper layers. In this stage, horizontal alignment is very critical.



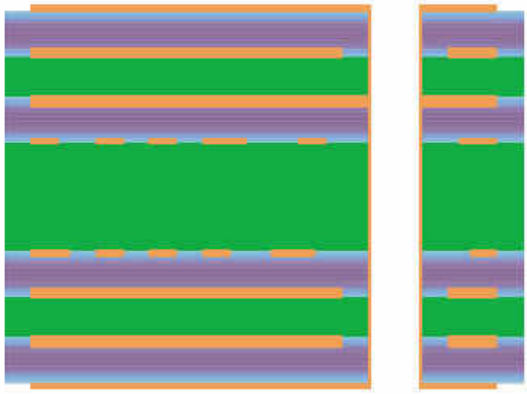
Step 5: Drilling and cleaning

Holes are drilled through PCB to interconnect layers (vias), and to allow the insertion of PTH components.



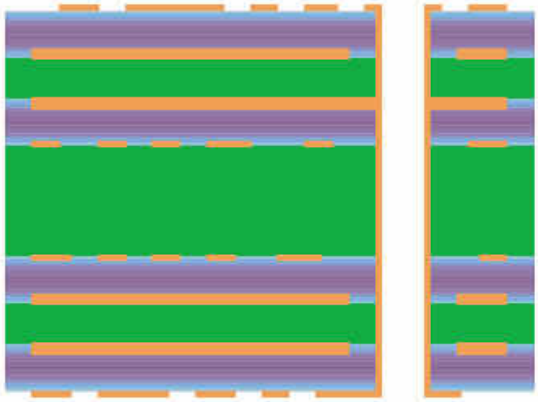
Step 6: Make holes conductive

PCB substrate is not conductive. Therefore, a non-electrolytic deposition method is required.



Step 7: Outer layer image transfer

Print, pattern plate, and etch!

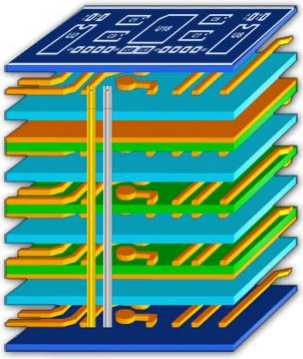


Step 8: Surface finish

The purpose in this step is to prevent copper oxidation.

Step 9: Final fabrication

Mechanical features are added to the PCB (mounting holes, cutouts, etc)



PCB Fabricator Selection

Printed circuit board fabrication is one of the most complex processes involved in the manufacture of electronic products. What to look for in a PCB fabricator for GoMax and therefore GoMax's customers become essential for the entire supply chain! GoMax's PCB fabrication partners have plenty of experiences in a variety of industrial and commercial products for over 10 years in Taiwan. As well known, good Taiwanese PCB fabricators can produce high quality PCBs among the best in the world. Below are the fundamental rules when GoMax constantly checks on PCB fabricators on a half yearly basis:

Sales Engineering Support

The best engineering support comes from fabricators who are routinely building PCBs of the complexity. They will have seen many things that can improve the yield of a PCB that a less experienced fabricator has not.

Materials Engineering Knowledge

There is a very broad range of potential materials from which PCBs can be built. They all have been developed to meet a particular need. When a material designed for one application is mistakenly used in another, the result is often a failed product! PCB makers must work with how a material shrinks when it goes through the heat of lamination and cool down, how it behaves in the lamination cycle, how it drills, how it plates, how the dielectric constant varies across the various thicknesses of laminate and how it varies with frequency.

Front End Processing and Tooling Creation

This is often referred to as the computer aided manufacturing part of PCB fabrication. Well-run front-end processes effectively lower the failure rate of the fabrication process.

Staff Training

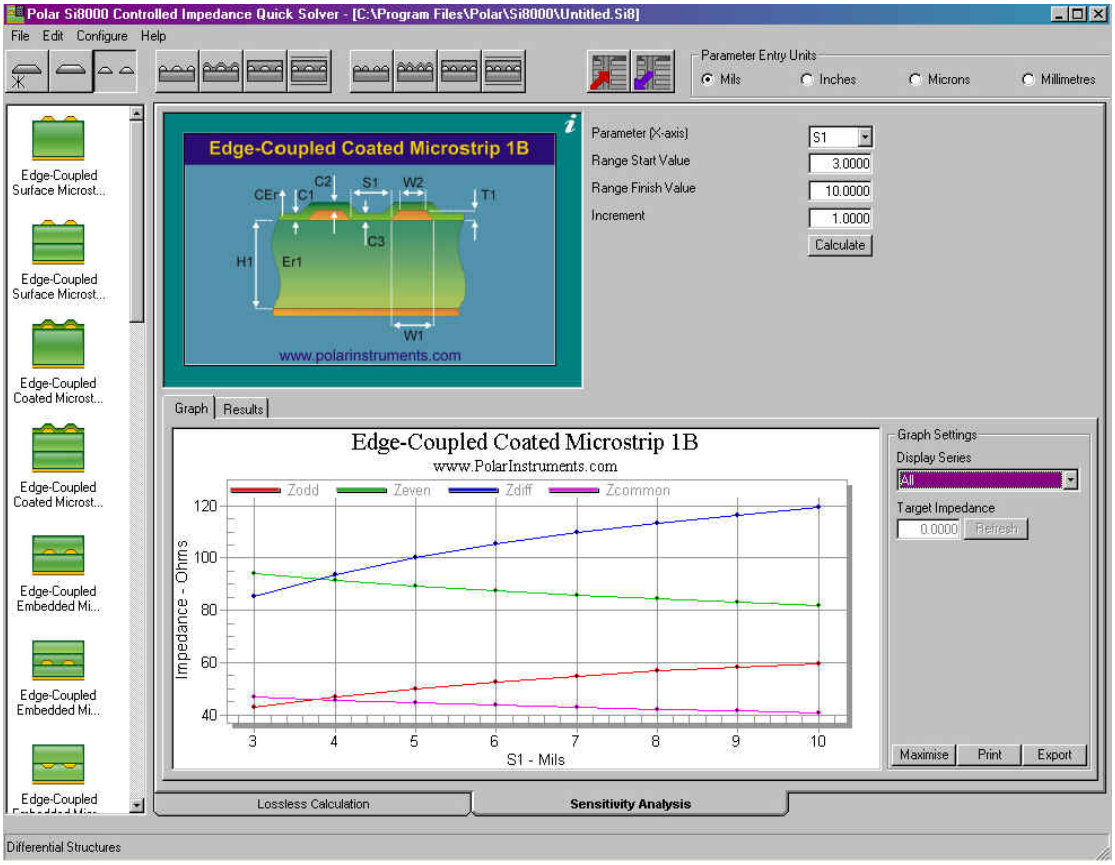
PCB fabrication relies heavily on the skill of the people operating the machinery and maintaining the chemistries involved in the process as well as the skill of the personnel preparing the tooling sets. Only with well-trained and well-motivated people, high quality PCBs can be produced.

Overall Process Controll

PCB fabrication involves more than 100 process steps. These range from creating

photo-tooling to a variety of chemical cleaning steps, plating steps, drilling, lamination, testing, coating, materials management and tracking the location of each order in the process. A good fabricator will have the activities involved at each step documented at each station!

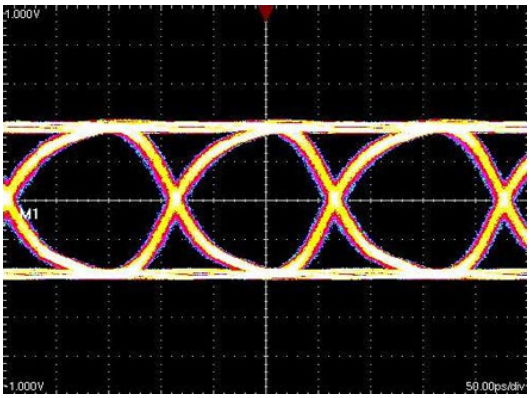
Impedance Control



This section we will discuss the impedance of the circuit, and why and how PCB has a deep impact in impedance! So, what is the impedance? Beginners with electronics get down Ohm's law pretty quickly. The concept that Voltage, Current, and Resistance to electrical flow are related by the simple expression V=IR seems to set pretty easily. Impedance is simply the generalization of the concept of resistance from DC to AC. That is, it's a way to represent how much current will flow with a specified (AC) voltage across the impedance. That is, if you have one volt AC across an impedance that lets one ampere of AC current flow, the impedance is defined by the AC version of Ohm's law and is one ohm. Since AC has not only amplitude, like DC, but also frequency and phase, this introduces the possibility that an impedance will not only allow a current to flow, but will change the phase of the signal, and

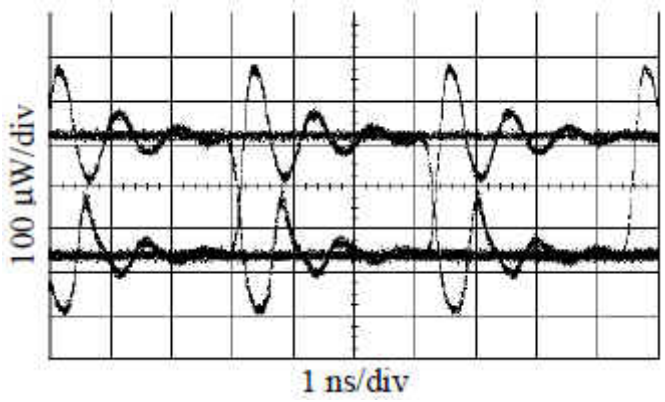
respond with different amplitudes and phases as frequency changes. One of the most important factors to determine the overall performance in today's high speed circuits is related to ***differential signaling*** and therefore the impedance match or control for ***PCB manufacturing***.

Differential pairs are usually found on a PCB, in cables, and in connectors. Differential signaling is a method of transmitting information electrically by means of two complementary signals sent on two separate wires. This technique can be used for both analog and digital signaling. Taking HDMI as an example, 4 differential pairs are required for single HDMI channel high speed A/V transmission. There are many reasons for people adopting differential signaling technique, such as high noise immunity and lower energy consumption, and therefore a very standard way to measure how the differential signaling quality has been developed and well known, which is the so-called eye pattern or eye diagram.

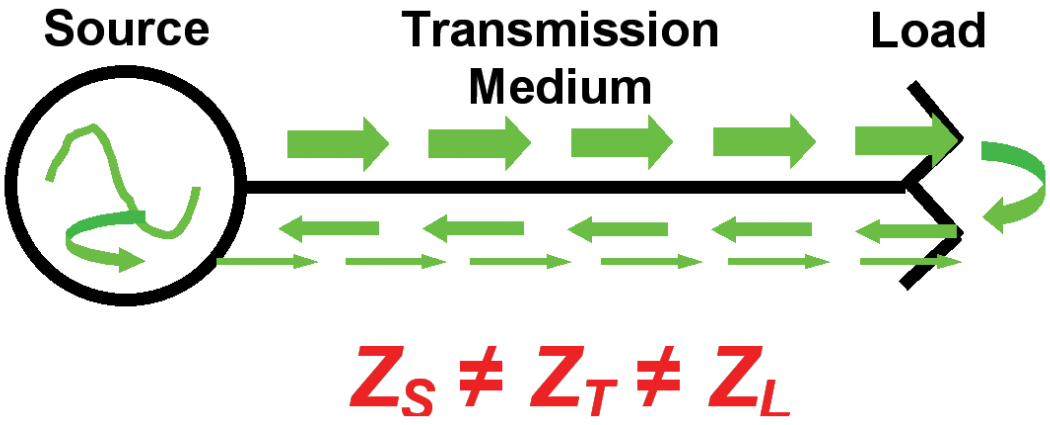


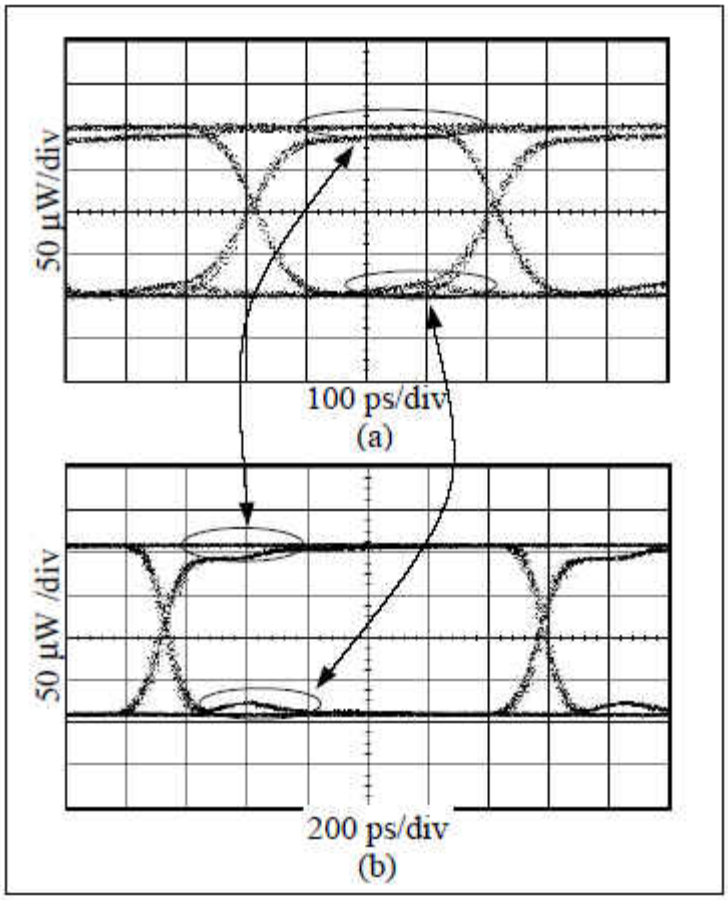
In order to maintain the performance of the transmission line for high speed A/V, the accuracy to compute PCB stack-ups so that impedance of the finished PCB is within limits becomes particularly important and the techniques to produce PCB with impedance within acceptance can no longer be neglected. Below are some examples that cause signal integrity is no longer maintained due to mismatch of PCB impedance, and therefore inferior performances in terms of transmission distance and noise immunity.

Ringing: Rising and/or falling edges exhibit ringing relative to the correct levels in a damped oscillation pattern.



Reflection: Reflections due to transmission line impedance discontinuities can appear as overshoot, undershoot, ringing or other distortions to the eye diagram.





With the ever increasing speeds of modern circuitry, the demand for high quality controlled impedance printed circuit boards is continuing to grow. Today's PCBs are not just simple electrical interconnection devices! They are complex highly specified components in their own right. As the demand for high speed circuit has quickly risen, good PCBs have been a subsequent increase in importance and therefore played a major role to deliver excellent quality products. GoMax has been paying attention to this major factor from PCB for years, and believe in the years to come, the quality of PCB will become more and more important and therefore be a significant element to the success of electronic products!

Pictures of PCB Partner