

The role of PCB fabricators in the PCB design process has always been an important one. And, with the complexity and speed of today's PCBs, the contribution of fabricators in ensuring that a board is built right the first time becomes even more crucial. Ultimately, the engineer who designs a board depends on the fabricator to provide counsel on the manufacturability of any given design. In order to provide adequate counsel, a fabricator must be well informed on several key technological aspects that include having a clear understanding of the available materials and their limitations as well as of the process steps involved and their limitations. It is useful to spend a little time reviewing the basics of what constitutes good manufacturability input.

## **The Basics of Manufacturability**

At the top of the list of manufacturability counsel, a fabricator must be able to tell the difference between what is possible and what is reasonable. While it may be possible to manufacture almost any board design it's only reasonable to manufacture a board that can be produced in volume at competitive costs. (Note: All of the following points in this article refer to the manufacture of high-speed PCBs, which is rapidly becoming all PCBs as component speeds continue to climb.)

In terms of possible versus reasonable, there are a number of errors that arise. I have encountered these errors with the clients for whom I provide consulting services. One of the most common errors is attempting to transfer a set of design rules from one type of board to another. For example, some fabricators think it is possible to transfer the design rules from a small, thin board, such as a memory DIMM, and map them onto a large, thick board. The result is attempting to build a board with two traces between 1mm pitch drilled holes or attempting to connect 0.8 mm pitch parts with through holes. There is simply not enough room on the board to accommodate these features and still make a manufacturable PCB that also has good signal integrity.

A second common error is for fabricators to manufacture a board with unreasonably small, drilled holes. Some fabricators claim that they can build a board with high aspect ratio holes greater than 10:1 (diameter vs. length). These fabricators will brag about the extreme

capability of their processes in terms of being able to build a board with a 15:1 aspect ratio or higher. While it may be possible to build such a board, it is not reasonable to do so nor is it practical to try and produce this type of board in volume. Many of these PCB fabricators claim these high aspect ratios as a way of demonstrating the quality of their operation regardless of their ability to produce PCBs in volume with these aspect ratios. The problem is, engineers often assume these claims to mean reasonable and then they build designs around such capabilities.

One of my recent consulting projects readily illustrates this concept of possible vs. reasonable. In this particular instance, the fabricator encouraged an engineering group to design a PCB with the following characteristics:

- A 22-layer board consisting of two 11-layer boards with blind vias drilled through each of the two 11-layer boards. Laminating the two 11-layer boards created the final 22-layer board.

On this particular board, two traces ran between 1mm pitch holes with a drill diameter of 10 mils. First, it is difficult if not impossible to hold the registration across all of the drilling operations with two-step lamination as in this example. This resulted in a yield that was almost zero. The board in question measured 6" x 12" and cost more than \$2,000 to build. The engineering group should have been counseled that the board as designed would never be manufacturable in volume. Instead, they were led to believe that it was not only possible but also reasonable. The result was a failed project.

## **The Elements of Fabrication**

With the foregoing basic premise of fabricators providing counsel of what is a reasonable design to consider building, it's useful to go over some of the things that comprise a manufacturable design. Among these are:

- Lowest price that meets all other goals
- All processes and materials second sourced
- Plating just good enough to meet reliability and assembly goals
- Tolerances loose enough that special process control is not

required

- Laminate spacings large enough to meet reliability targets

## **Laminate Material Types**

There are a wide variety of dielectric materials that can be used in the PCB fabrication process. Determining which laminate material is best for a given design is based on the following criteria:

- Dielectric loss (loss tangent)
- Ability of the material to withstand high temperatures during assembly and rework (reflected by glass transition temperature,  $T_g$ ).
- Cost. The material that gets the job done at the lowest cost is the best selection
- Uniformity of dielectric constant across a layer (required to achieve uniform impedance)

The role for a fabricator in material selection is to know the characteristics of all of the available materials including their limitations. An important piece of information that a fabricator needs to supply the design engineer is an accurate relative dielectric constant of these materials as a function of glass to resin ratio and frequency. This is vital to ensuring the impedance and velocity or time delays are correct.

When the subject of “high frequency” material arises, what is meant by this term is low dielectric loss. It is important for each fabricator to know the loss tangents of the available materials. Unfortunately, it is not possible to determine what loss tangent is “good enough” without performing complex simulations of the circuits that will be contained on each PCB. From this information, the lowest loss tangent material that is just good enough can be selected. This kind of analysis is beyond the skill set of a PCB fabricator and is the responsibility of the design engineer.

It is not automatically necessary to use the lowest loss tangent

material in a design. For example, I have done designs with 9.6 GB/S paths in the material commonly called FR-4 while I have had to use Nelco 4000-13SI in a 2.4 GB/S path. Which material is needed is determined by path length, trace width and ability of a circuit to tolerate loss. None of these characteristics are known to the fabricator so, again, the design engineer must provide this data.

## **Lamination**

It is well known that foil lamination results in the lowest cost multilayer PCB. All other lamination choices--cap lamination, two-step lamination and build-up lamination--are more expensive, with two step lamination usually being the most expensive. A fabricator should make it very clear to a design engineer the costs and limitations of each of the lamination processes and provide as much help as possible to achieve a design that uses the least expensive process possible.

Encouraging two-step lamination, as happened in my example, is as close to malpractice as one gets in this industry. It has a place only where cost and yield are less important than the complexity.

One important thing a fabricator needs to know is how much each laminate type shrinks during lamination and have the ability to scale the artwork used to image the layers to account for this. This will insure that after lamination the images are the correct size to allow proper layer-to-layer registration.

## **Registration**

Registration is where the rubber meets the road in the lamination step of the fabrication process. Maintaining layer-to-layer registration is a crucial element in the fabrication of multilayer PCBs. The five places in the fabrication process where the registration of images to each other is required are:

- Registration of two layers to each other on opposite sides of a piece of laminate
- Registration of all the inner layers to each other during lamination

- Registration of the drill to the inner layer images
- Registration of laser drilled blind vias to inner layer images
- Registration of drill to two parts of a two-step laminated PCB

From the above, it can be seen that there are a number of places where images and drills can be misaligned resulting in possible breakout of the drill in a signal pad as well as reduction in insulation clearance between circuits and the plating in the drilled holes. It is a common error to use the same tolerance allowance for a standard foil laminated, low layer count PCB when designing a high layer count PCB that requires back drilling or two step lamination. The result of using the wrong tolerance allowance is a design that has poor yields or violates insulation spacing standards.

## Drilling

Three forms of drilling holes are used in the production of multilayer PCBs. They include:

- Standard Mechanical: used to create through holes; can be used to create blind and buried vias as well as back drilled holes.
- Laser Drilling: used to produce blind vias
- Chemical Etching: used to produce blind vias, this is often referred to as build up.

The two big issues with drilling are hitting the desired location within the accuracy required to meet clearance requirements and the ability to plate the drilled hole both with the copper and the corrosion finish plated on top of the copper.

The common method used to specify hole size is to call out the finished hole size. The fabricator then chooses a drill size that allows for the plating. The problem with this method is features are so tight on most modern PCBs that insuring proper allowances for drill wander, insulation spacing and breakout on signal pads as well as clearances in power planes, requires the PCB designer to dimension the pad sizes from the drilled hole rather than the plated hole. Therefore, fabricators need to provide the designer with the

allowance for plating as well as hole-wander so the PCB designer can dimension pads properly.

A second consideration for drilling is allowing enough tolerance for the location a drilled hole will have in a pad. This is comprised of all the registration build up described above and the ability of the drill to find its location on the PCB panel. This allowance must be larger as the PCB panel becomes larger and thicker and as the number of layers grows. It must be even larger when two-step lamination is required. All too often, when a fabricator provides this allowance to the PCB designer it is a single number, regardless of the PCB complexity and it is usually very optimistic. This is a common reason for low yielding PCBs and PCBs that are unreliable.

A third consideration is the aspect ratio mentioned above. The drill size chosen must provide an aspect ratio that can be plated successfully in the hundreds of thousands or even millions of holes.

Back drilling is becoming a popular way to reduce the parasitic capacitance of a plated through hole in a multi-gigabit PCB. When this operation is required it is important to remember that some of the insulation along with the copper plating in the hole is removed. When this happens, the clearance from internal copper to the outside world has diminished. To maintain proper insulation, the clearance from drilled hole to internal copper will need to be larger than for a normal through hole. This is rarely allowed for and results in PCBs (usually backplanes) that slowly develop high resistance shorts.

## **Plating**

There are two types of plating involved in the manufacture of PCBs. These are the copper plating needed to create the vias and connections to inner layers and the plating used to protect the copper where contacts will be made by soldering or pressure connections. The first is meant to serve as a conductor much like the copper in the layers of a PCB and the second is meant to provide a corrosion barrier over the copper.

Copper plating is what turns a PCB into a multilayer PCB. It makes all the connections between layers. Its quality has a direct effect on overall PCB reliability. The higher the aspect ratio of the drilled holes the more critical control over this step becomes. Achieving reliable

high aspect hole plating is done with reverse pulse plating. Attempting such work without it should be done with care.

The second plating step, applying a corrosion resistant plating to the copper, is just as vital because it preserves solderability. A fabricator needs to know which type of plating is rated for which level of reliability and make sure it is available in the process flow.

### **What Should Be Expected from Fabricators?**

The fabricator's engineering group is the key resource on the manufacturability of the design rules and must provide feedback to the design engineers when things are not reasonable, as well as educate engineers on what is practical and available. Today, there are essentially three classes of fabricators:

- Fabricators who know their role is to provide manufacturability wisdom to their customers so they are successful the first time and every time.
- Fabricators, fortunately not many of them, who will agree to whatever a design engineer wants to do no matter how unreasonable it is. The customers of these fabricators very often wind up with products that can never be manufactured. The worst outcome of this kind of relationship are PCBs that function properly on the production line but are not reliable, punishing the customer long after the order is complete.
- Fabricators who just don't know what their role is. These fabricators are not properly qualified to be manufacturing partners. Eventually, these fabricators disappear altogether.

Sometimes a dilemma is created for the first fabricators. They give good advice to design engineers when something is not reasonable so the design engineers go to the second group of fabricators who ensure the design engineer that the project is possible. The problem comes when that possibility never becomes reasonable due to poor yields and/or high costs.

The start-up environment of the last few years has created pools of engineers who don't have a lot of experience. In older, established companies, there are senior engineers who can provide guidance on what constitutes good manufacturability. It turns out that the best way

to know what comprises good manufacturability is achieved through years of experience. For those start-ups who don't have this experience readily available to them, they have to rely on fabricators to provide them with the guidance on what is reasonable in terms of manufacturability. This will ultimately ensure them a successful PCB that is built right the first time.