PCB COST REDUCTIONS

by Markku Jämsä

SUMMARY

Although the entire toolbox of PCB cost reductions are beyond the limits of this article, Jamsa Markku takes a look at some strategies, which include design steps, total cost considerations, and of course, keeping it simple and doing it right the first time.

There is an old joke about the PCB Commodity Manager having one brain cell only, either occupied by the idea of price reduction or by the idea of capacity allocation. If the commodity manager looks angry when you meet him, he is thinking about cost reduction; if the CM smiles, it is allocations. As a polite PCB supplier, I must make myself clear that all the CMs I know have several brain cells, and therefore it is not always so straightforward.

At the moment, the allocation topic likely occupies the cell, since, for the second year in row, the capacity of PCB manufacturing has been well booked. The PCB industry finally has some possibility to compensate some of the increased costs of the PCB. The industry has seen significant price increases of basic raw materials of the food chain like copper, nickel, gold, raw oil, electricity, water, not to mention labor cost. These cost trends have been even stronger in Asia than in Europe. Therefore, the PCB cost reduction by the price itself seems to be not the path to follow for the commodity managers. They would rather continue with the existing prices and try to minimize the cost pressure coming from the bottom of the chain.

On the other hand, the cost reduction projects through the design of the PCB look more promising. Aspocomp, as many other PCB companies do, receives every now and then some requirements to assist to re-design some existing high runner PCB. The entire toolbox of possibilities of how to make cost reductions are beyond the limits of this article, but let's have a look at some examples.

PCB Design Steps

Normally, the PCB shop sees only the output of PCB layout design, the Gerber zip, PCB specification sheet and some mechanical drawing. However, there are a few more design steps before layout design. These may also have a great impact to the PCB cost. Quite often the engineers who are responsible for these earlier steps have limited knowledge of PCBs.

“ MINIMIZING THE NUMBER OF PCBS WILL ELIMINATE CABLES AND CONNECTORS, SHORTEN AND SIMPLIFY THE SIGNAL ROUTING AND, IN ADDITION, SAVE COSTS INVOLVED IN MECHANICS. ”

Schematic design is a phase when the hardware (HW) engineer selects the components and creates the wiring diagram of the system. In wiring diagrams of any complex system, the designer has to decide which functions will be placed into the same PCBs. Minimizing the number of PCBs will eliminate cables and connectors, shorten and simplify the signal routing and, in addition, save costs involved in mechanics. However, if the single PCB tries to digest too much, meaning too many functions are being integrated into the same PCB, it may become too complex and large. In that case, it will have a high cost due to low yield of PCB and of the assembly, but also it may become complex to debug and rework.

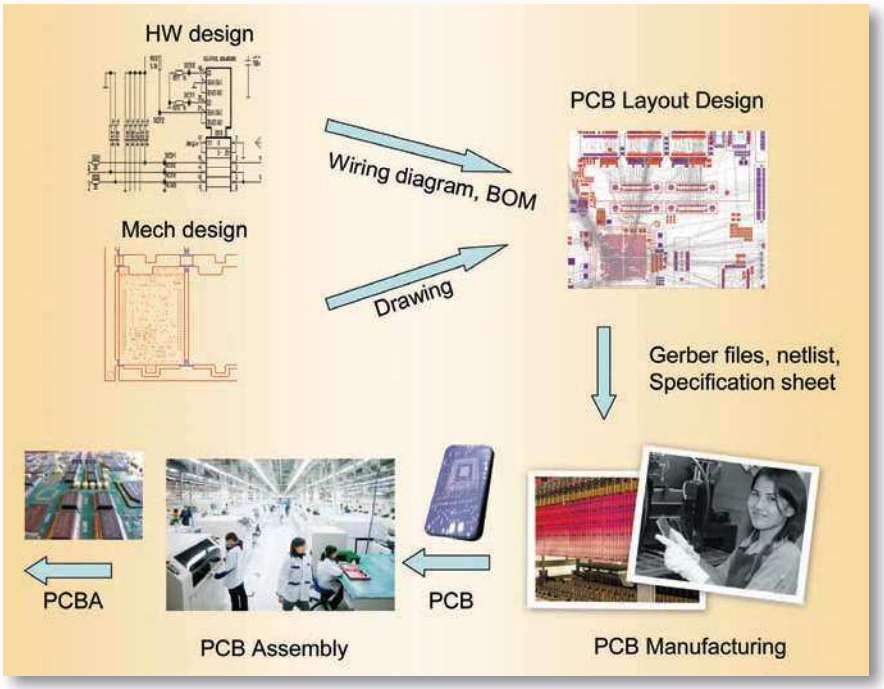

Figure 1. Simplified process flow how to make a printed circuit board assembly (PCBA). It is quite typical for all of the steps to be done by different companies.

Additionally, the HW has to consider how to integrate the functions. For example, it may be attractive to integrate power supply to the same PCB rather than signal processing. In some cases it may turn out too complex to make high copper and high-density wiring in the same PCB since high copper thickness will limit the usage of narrow line/space for signal routing.

Designing the mechanics is also one of the key processes from PCB cost point of view. The PCB outline is usually coming from mechanical design and therefore quite a lot of material utilization is actually decided already in this phase. This is mainly true with large size PCBs, when the dimensions exceed 210x280mm. Just a few mm in x or y dimension can make a difference of 10-30% of the PCB cost. Unfortunately, quite often this consideration occurs too late, typically during the quotation process for the volume supply. At that point, the mechanics have already been ‘frozen’ a long time ago; there is no chance to change any dimensions.

In case of small PCBs, mechanical design is not similarly important regarding the PCB cost. In this case, the main focus should be to optimize the delivery panel for assembly and also at the same time for PCB manufacturing. Usually, the small size PCBs will be delivered in the delivery panel having several PCBs in the same ‘array.’ This panel approach reduces the handling cost in PCB manufacturing and provides more savings during the assembly and soldering of the components. Preferably, the final design of the panel remains the electronics manufacturing services (EMS) company’s responsibility. The EMS and PCB supplier can design the panel together to find the optimal cost.

Good Old Tricks

I think most of the experienced layout designers are well aware of the basic PCB guidelines, like keeping the design rules as robust as possible and avoiding any idle surface area of the PCB. Keeping the copper layers as homogenous as possible is also a good idea. To some extent, minimizing the layer count and keeping the via technology as simple as possible will be a big cost factor. These are just a few; there are several more on the designer’s check lists.

Have a Look at the Total Cost of the PCB

Sometimes we see PCBs where the designer has been looking at only one factor of the cost. Let me make an example of one project some time ago where the designer focused on via technology only, knowing that the microvia PCB is more expensive than multilayer PCB. The initial situation was 8L HDI (high density interconnection) PCB of 1-6b-1 construction, having Cu fill for microvia in order to ensure flat surface for some SMD pads. As such, this PCB was not the cheapest one and there was obvious price pressure. After a while this PCB was replaced by a new revision, having 8L construction without microvia, having PTH (plated through hole) only. In order to solve the flat pad issue, he had to use POFV (Plated Over Filled Via) technology. Also, the wiring density increased from 100/100 to 75/75u in order to make required routing of the signals. 75/75 was also the case in the surface layers which have a bit thicker copper due to POFV technology. Also, the so-called PTH to Cu parameter went right to the border of the capability.

The customer was surprised when, in spite of all design efforts, he received quotations showing almost the same price as before. In principle, ML (multilayer) should be about 30-40% cheaper than HDI with the same size, material, thickness and layer count. As you can see in Picture 2, this modification did not simplify the PCB much, but considering the yield impact, it actually ended up to the same cost as before. A significant effort was made and, in the end, no cost saving was available, but a lot of discussion and frustration was.

As we can see, it is important to consider the entire PCB when doing the cost reduction projects. Good practice is to make a PCB specification sheet of the alternative construction before the layout effort, having all of the new 'spices' included. Then, you can simply ask for a budgetary quote from your main PCB suppliers. It is important to invest some of the designer time to make this specification draft complete enough, considering the main cost factors:

- layer count
- via technology (PTH, microvia, filled via, buried via and such)
- size of the PCB and size of the delivery panel
- copper thickness in different layers
- dielectric material
- line/space in different layers
- pad sizes and PTH to Cu clearance
- min hole sizes and estimate of the number of holes
- surface finish

This may sound a bit heavy to do—designers are busy people. Also, a few designers even feel that the specification sheet is just something a layout designer has to do, after the layout work itself, to 'keep the PCB supplier happy.'

Doing a specification sheet before layout phase may be different from traditional layout work flow. However, this practice requires the designer to consider and document all the cost

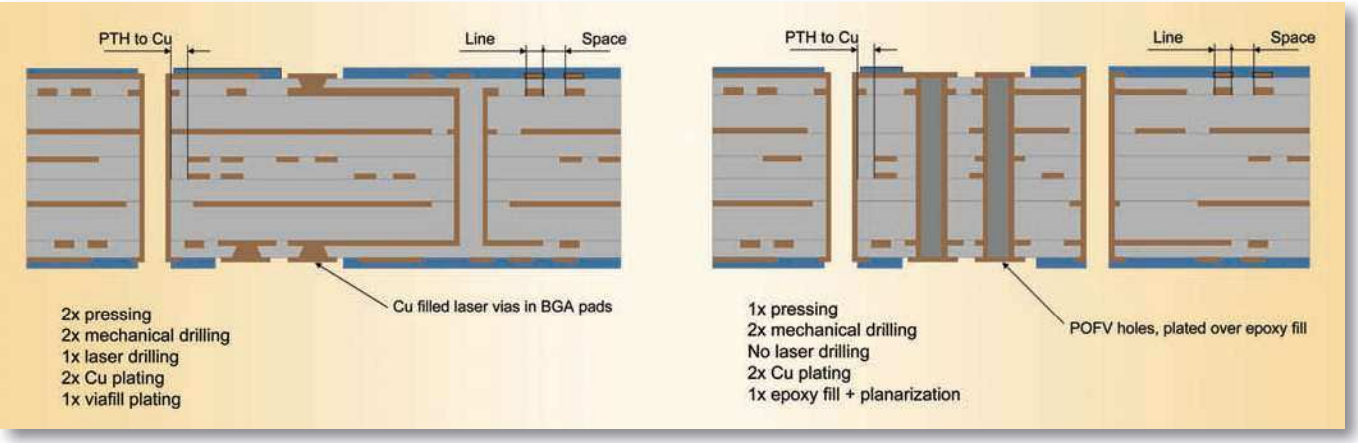

Figure 2. Cross sectional example of 1-6b-1 construction versus 8L with POFV (Plated over Filled Via).

drivers and design parameters in advance, before jumping into the layout project. Most of the PCB shops are quite familiar to process RFQ (Request For Quotation) by spec sheet only, a budgetary quote will provide quite realistic picture about the cost-saving potential. Draft specs will be much more accurate than 'what if' types of emails asking about cost impacts of some individual design parameters.

This procedure of making a specification draft first can be also very useful for any fresh designs, providing a powerful tool to estimate the PCB cost and availability in advance. Naturally, the designer will not know all the numbers in advance. For example, the number of PTHs. However, I am pretty sure that an experienced designer has a very good possibility of making a good estimate. He/she can then just finalize the spec sheet after the layout itself is finalized.

Keep It Simple

Two different PCBs in the same delivery panel may look like an excellent idea, and sometimes it is. Especially if the shape, dimensions and the laminate utilization will support the case. If the PCB technology matches 100% and the required quantity will match, it may provide a good way to provide several PCBs to the project in the same package. However, one needs to be careful. All the processes need to be exactly the same, not only the PCB manufacturing, but also the assembly process. Sometimes, in spite of an excellent initial idea of integrating, the customer wants to split the PCBs in the end. For example, the assembly yield of the PCBs in the same panel is different for some reason, or the assembly process is simply different in volume production.

The risk of integrating the PCBs too wildly is high in cases where the design partner is responsible for prototype manufacturing, including the PCBAs (Printed Circuit Board Assemblies). It may provide an immediate cost reduction for the design house to put 2-3 different PCBs into the same delivery panel in order to save setup cost of the PCBs and PCBAs. However, this short-term saving may turn up quite expensive in volume manufacturing. OEM needs to have a close look at this, especially if the PCB and PCBA supply for the volume will be different than those used by the design house.

Do It Right the First Time

Naturally, the optimal case is when the project will be executed by experienced designers in all steps. Then, there is a good possibility that the nearly 'total cost optimal' design comes up already from the first revision on, and the necessary small modifications will be done with the next revisions, resulting in a good balance of functional performance and cost performance.

In a step-by-step split delivery chain, this becomes quite challenging since the acts in the first steps generate costs in the last steps. The feedback chain is quite long and in many cases the feedback does not reach the appropriate designer. And if it does reach, he/she is probably occupied already by the next project, possibly for another customer. The motivation and the possibility to do some modification for any old project is quite remote.

There are some possibilities to tackle this dilemma without going back to vertically integrated value chains (which will not come back, I am afraid). One is to provide good guidelines for necessary PCB design reviews. An even more effective one is to provide adequate training for design people, including both schematic and mechanical designers. Training about PCB cost factors can be organized by the PCB shop, which needs to cover the related cost somehow. The main problem here is the availability of the designers for training. As said, they are always busy, but somehow they need to create and maintain their professional skills, including the manufacturability and the cost rules of the components they are designing.

Do It Together with Your Long-Term Partners

For a long time, the PCB food chain has been split in small slices like HW design house, PCB layout design house, EMS partner for the prototypes and sometimes different EMS partners for the volume manufacturing.

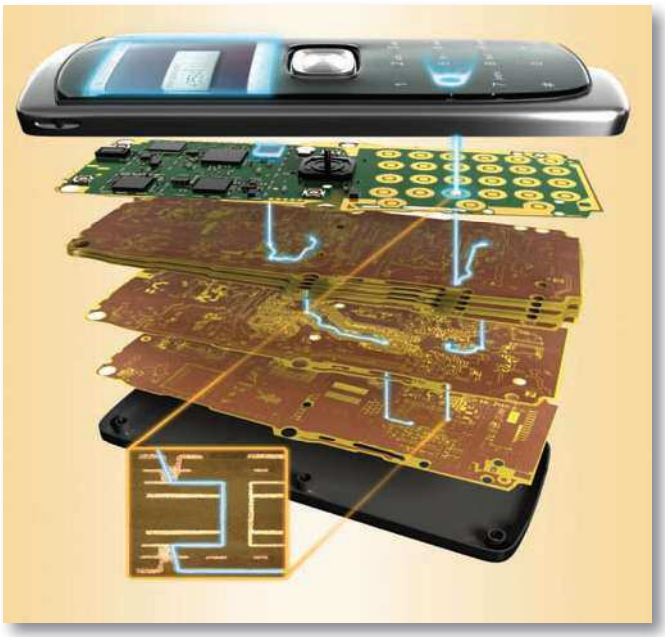

Figure 3. Please connect the partners also, not just the components!

Then, there is naturally a PCB shop for prototypes and the volume PCB shop. Still further, there are component suppliers, laminate and other raw material suppliers. Most importantly, there is some OEM to put this all together. Some of the OEMs are looking very carefully at the cost of each slice separately; others concentrate more in total cost approach.

If the approach is to get every slice in the fastest possible time at the lowest possible cost, there can be some trouble ahead. It is necessary that there is some responsible project owner, in the OEM organization, who ensures that there will be enough co-operation between the parties. Also, there must be some reasonable possibility for each partner to communicate with the designers in the early stages. Some cases are quite 'hidden,' on purpose, to protect the projects from the competition, but even then, the trusted partners can do some significant cost savings and keep the project completely non-disclosed.

How is it possible to organize all of this in order to get best total cost and functionality for the PCBs and PCBAs? In my opinion, there are some quite general and useful ways. Firstly, dear OEM, please introduce the partners to one another, and make sure that the PCB shop will know the contact persons in PCB design. The OEM also needs to ensure that the designers have good enough knowledge about the PCB and PCBA cost drivers. Also, if there are different companies for prototypes and volume supply, please make sure that these companies can talk to each other, or, alternatively, make sure that the initial DFM comments will be from the volume supplier as well. This is especially important for any high volume products; naturally, for small quantity projects it may not pay off.

Quite often, the volume supplier needs to quote something which is far away from optimal and the usual reply from an OEM or EMS is that there was no trouble getting the prototypes from some other PCB company. (It is likely true, since prototype PCB shops quite often launch excess PCBs to ensure a quick turnaround batch. The yield itself may be low, but good enough to make the proto delivery in full quantity.) In typical cases, it will then be far too late to make any important modifications, and therefore OEMs will lose a significant amount of money during the volume manufacturing. In most severe cases, there are obvious difficulties finding volume suppliers.

What happened meanwhile with our fellow PCB commodity manager? Since, in reality, the CM has an entire head full of brain cells, he will be the key person guarding the PCB and total cost through the entire process, and he will be most welcome to do so. In the best case, he has some colleagues responsible for PCBAs and components, sharing the same total cost target, not just separated targets of PCB, assembly or the components. **PCB**

Markku Jämsä is senior manager in Aspocomp Oulu Oy, responsible for PCB technology, technical co-operation with customers and sales. He has worked in the PCB industry since 1988, with responsibilities including quality, technology and production. You may contact him by email at: markku.jamsa@aspocomp.com.