Technique Report of Gold Phoenix Tech Co, Ltd

Website: ca.goldphoenixpcb.com, Email: sales@goldphoenixpcb.com

Cost Analysis of Printed Circuit Board

By Gold Phoenix's North America Service and Tech Center Team

Abstract: With the development of modern technology, PCBs are widely used; and with the cost down of electronic equipment, everyone wants cheaper and cheaper prices. The heavy cost pressure can also be felt in the PCB industry. Of course when making the PCB boards, we can find a lot of ways to reduce the cost, like changing the base material, changing the material of solder mask…By using all these methods, the prices of printed circuit boards can be reduced by 10% to 50%. But there is no "free lunch". The reduced cost of PCBs may come at the expense of the increasing the cost of assembling PCB, reducing the long-term reliability of the equipment. So it may increase the final cost of the equipment.

With our increased business in North America, a lot of customers ask us how to reduce the cost of their PCBs. In order to help our customers to know the PCBs better and give some guidelines on reducing the cost of PCBs, we will introduce the propriety of different PCB materials and important prosperity index of them. In this tech report and we also give some detailed information on the cost of PCBs. This tech report is written by the tech center of Gold Phoenix in China and its North America Service team, we hope this report can answer some questions of our customers.

I. INTRODUCTION OF PCB MATERIALS

The materials used to make PCBs are very important. And because the price of different material can vary from 10% to 100%, so the price differences of PCBs made by different materials can be huge! But as we said before there is no "free lunch", usually low cost material come with low quality. In this section different PCB materials will be introduced.

1. Glass transition temperature Tg: A high Tg is very important for the PCBs to guard against barrel cracking and pad fracture during soldering operation

2. Coefficient of thermal expansion (Tce): SMD assembly process exposes the printed wiring assembly to more temperature shocks than typical through-hole process. At the same time, the increase in lead density cause the designer to use more and more

layers, making the board more susceptible to the problems concerned with the base material's Tce. This can be a particular problem with regard to Z-axis expansion of the material since this induces stress in the copper-plated holes, and becomes a reliability concern.

Dissipation factor: A measure index of the tendency of an insulating material to absorb some of the ac energy from electromagnetic field passing through it. Low values are important for RF application, but relatively unimportant for logical applications.

Electrical breakdown voltage DBV: The voltage per unite thickness of an insulator at which an arc may develop through insulator.

Water absorption factor WA: The amount of water an insulating material may absorb when subjected to high relative humidity, expressed as a percent of total weight. Absorbed water increases relative dielectric constant as well as reduces Electrical breakdown voltage

There are 6 major different materials--- FR1 FR2, FR3, FR4,CEM-1, CEM-3--- used to make PCBs . They are introduced below:

a) FR1-FR4

FR1 is basically the same as FR2. FR1 has a higher TG of 130°C instead of 105°C for FR2. Some laminate manufacturers who produce FR1 may not produce FR2 since the cost and usage are similar and it is not cost effective for having both.

FR3 is also basically FR2. But instead of phenolic resin it uses an epoxy resin binder.

FR4 (FR = Flame Retardent)is a glass fiber epoxy laminate. It is the most commonly used PCB material. 1.60 mm (0.062inch). FR4 uses 8 layers glass fiber material. The maximum ambient temperature is between 120° and 130°C, depending on thickness.

In China FR4 is the most widely used PCB base material, next is FR1 then FR2. But FR1 and FR2 are usually used for 1-layer PCBs because they are not good for passing through holes. FR3 is not recommended to building multi-layer PCBs. FR4 is the best selection. FR4 is widely used because it is good to make from one- layer to multi-layer

PCBs. With only FR4, PCB companies can make all kinds of PCBs, which leaves the management and quality control much easier, and eventually it can reduce the cost!

b) CEM-3, CEM-1

CEM-3 (CEM = Composite Epoxy Material) is very similar to FR4. Instead of woven glass fabric a "flies" type is used. CEM-3 has a milky white color and is very smooth. It is a complete replacement of FR4. But it not popular in China, so it is not cost effective to use it.CEM-1 is a paper-based laminate with one layer of woven glass fabric. It is not suitable for Plated Through Hole, same as FR1-3. CEM-1 can only be used for one-layer PCB.

c) Conclusion

Form the analysis above it is obvious that FR4 is the best selection to make PCBs in China market. Some materials may be cheaper but it can only be used to make one-layer boards, and the reliability of the boards made by these materials is not good. So they are only used for one-layer and very simple boards.

Then there comes a question--- If the FR4 is the best choice, then how to reduce the cost of the PCBs. This question will be analyzed in the next section.

II. COST ANALYSIS OF PCBS

From the analysis above it is clear that FR4 is the best cost-effective material for multi-layer PCBs and the reliability is the much better compared with other base material. Then how to reduce the cost of the PCBs if we use FR4 material?

One simple way is to use cheaper and low-quality FR4 material. At the first sight this may be seemed as a good way if the board's quality is acceptable, but think twice before you make the final decision. High-quality FR4 material with a higher price is more stable and it will be much easier for you to control the quality of PCBs later, which is especially true in mass production. Big PCB manufactures usually only use high quality FR4 material, because it is easier to control the quality of their products. They reduce the cost by going really high volumes (let's say 20,000m^2/month). The prices of high-quality

FR4 at high volumes are not necessary higher than that of the low-quality FR4 at low volumes.

Then why do some companies still use low-quality FR4 material? The reason is that this kind of PCB manufactures is usually quite small and they can not go high quantities to reduce the cost. In order to survive in the market they get to reduce the cost. And for prototype it is a very effective way to reduce the cost by using low-quality FR4. Also quality control is not a problem in this case. In addition big PCB manufacturers usually are not interested in making prototypes, so small PCB manufacturers can still make a living in the severe competition. In mass production some companies still use the low quality FR4 material, but the final price of the board they made are not necessary has a lower price, because their quality control cost will increase, and percentage of the qualified boards will also reduced.

Then if we use high quality FR4 material how can we reduce the cost? In the flowing paragraph we will give some datum to show how the final prices come out.

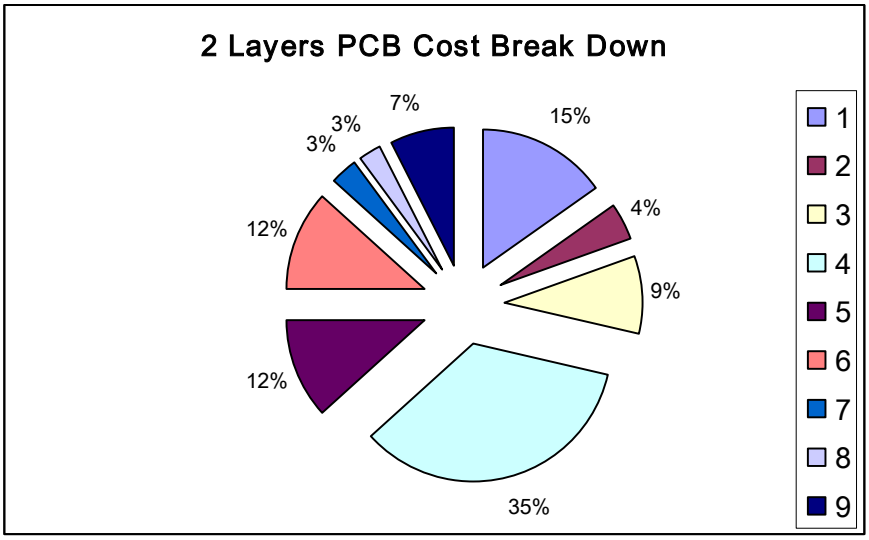

Fig 1 2 Layers PCB Mass Production Cost Break Down (PCB size 4.13X4.19inch, 40m^2/order)

1.Base Material(FR4) 2.Chemical Material 3.Drilling Fee 4.Plating and Finishing 5.Photo Etching 6.Solder mask and silk screen 7.testing and Shaping 8.Managing Cost 9 Profit

From the analysis shown in Fig 1 it is obvious that the highest cost in the mass production is plating and finishing 35%, next is FR4 material 15%, solder mask and silk screen 12%, test and shaping 12%.

So the most effective way to reduce the cost at mass production for 2-layer PCBs is to reduce the number of components and vias, to reduce the complexity of the PCB!

2-layer PCBs without solder mask and silkscreen will reduce the cost by around 12%. If less expensive material is used for solder mask and silkscreen, the price can be reduced around 5%.

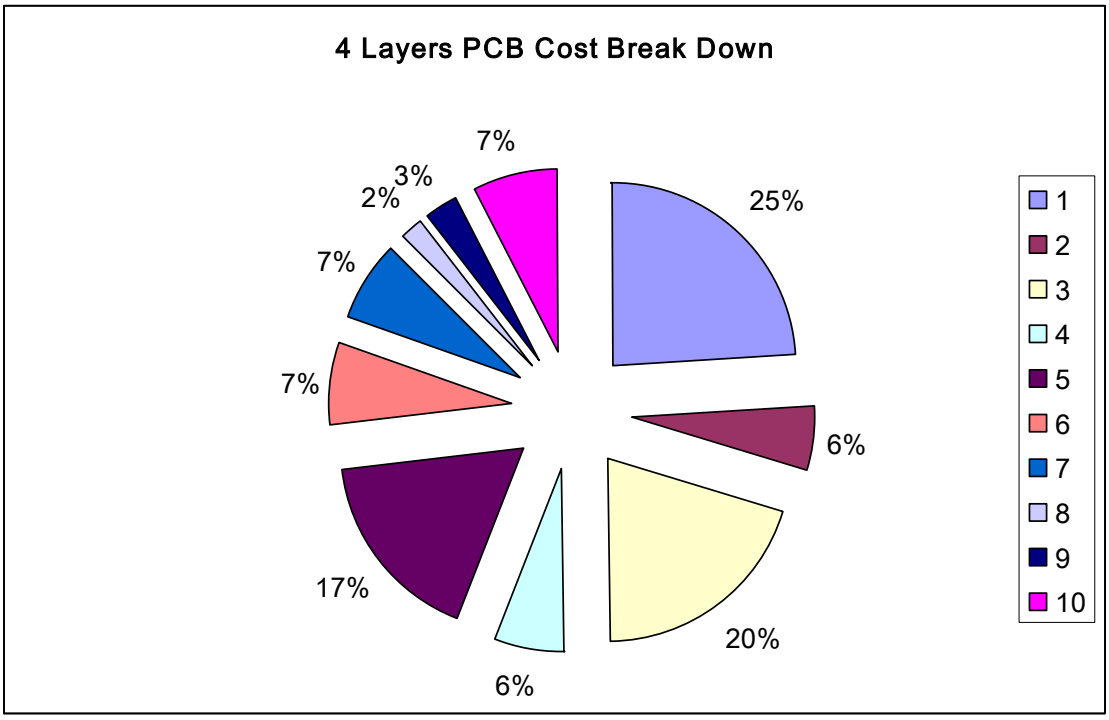

Fig 2 4-Layer PCBs Mass Production Cost Break Down (PCB size 7.08X6.65inch, 40m^2/ order)

1.Base Material(FR4) 2.Chemical Material 3.Internal Layers 4.Drill Fee 5.Plating and Finishing 6.Photo Etch 7.Solder mask and silk screen 8.testing and Shaping 9.Management Cost 10. Profit

From the cost breakdown of the 4-layer PCBs, the cost becomes more complex than 2-layer PCBs. This time the material cost is the highest (25%), next is internal layers (20%), then plating and finishing (17%).

But 20%+17% is still much higher than 25%, so again to reduce the complexity of the PCBs is the most effective way to reduce the cost!

And since using low-quality FR4 in mass production of multi-layer PCBs will increase the managing cost, the final price is not necessarily reduced. The safest way is to change to cheaper solder mask and silk screen material, and the cost can be reduced by around 3%.

Cost of other multi-Layer PCBs such as 6-Layers, 8-Layers…….have similar result.

III. CONCLUSIONS

In this tech report, the cost of a 2-layer and 4-layer PCBs are analyzed. From the datum we get from our PCB line, it is obvious that the most effective way to reduce the cost of PCBs is to simplify the design of the PCBs such as reducing the number of vias and components. The safest way to reduce the cost is to get ride of the solder mask and silk screen, by which the cost can be reduced by 5% to 10%.

Reducing the cost by using low quality FR4 material is very effective for prototypes, but in mass production the managing cost will increase, which is especially true in the mass production of multi-layer PCBs. In addition because low quality FR4 usually has lower Tg and higher Tce, the cost to assemble the PCBs may increase. So even the cost of the PCBs is reduced, the price for the final products may not be reduced.

To accurately analyze the cost of the PCBs and its relationship with the cost of final products is a very complex task; a lot of analyses will be needed. This tech report just gives our customers some guidelines and suggestions when they want to reduce the cost of their products, and we hope this report can answer most questions of our customers on reducing the cost of PCBs.

Gold Phoenix Tech, 2004-4-14