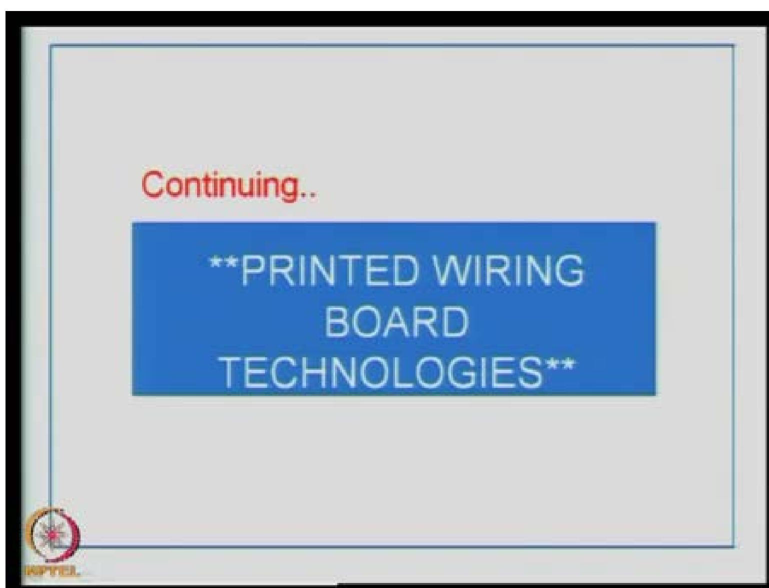
An Introduction to Electronics Systems Packaging
Prof. G. V. Mahesh
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 06
Lecture No. # 24
Process flow-chart
Vias
PWB substrates

We will continue with the printed wiring board technologies. This is the second in the series of this chapter on printed wiring board technologies.

(Refer Slide Time: 00:27)



So, we will continue with the material aspects of printed wiring board and the various processes involved in making a complete printed wiring board; there are different types, for example single sided, double sided, flexible boards, substrates for multichip modules and so on.

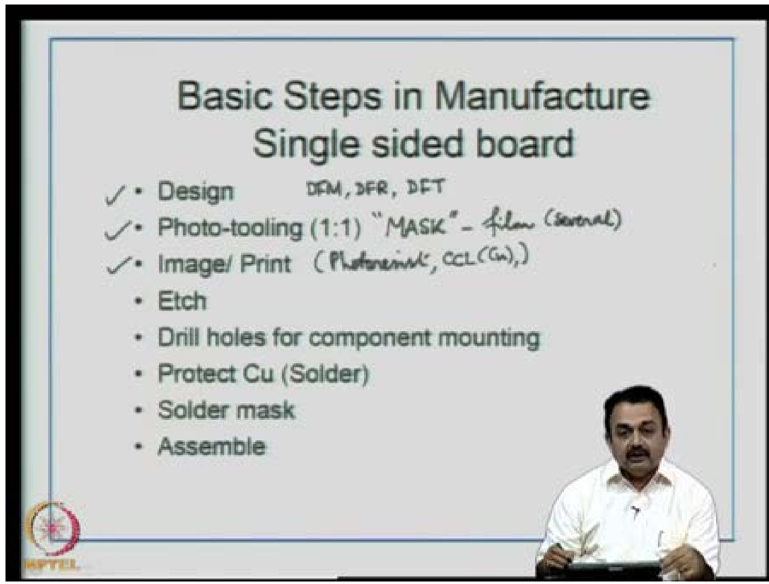
(Refer Slide Time: 00:30)

In the last class, if you can recollect, we have discussed briefly the basic steps in the manufacture of a single sided board. The first step is completing the design; making sure all the aspects of design are in place, especially from the viewpoint of design for manufacturing, design for reliability and design for testability.

So, I want to emphasize these points time and again. Then, the design is sent for creating a photo tool. We saw that a photo tool is also known as a mask. So, the output from a design will now be used to create a mask. A mask is basically a film that houses your design or contains the complete circuit; you may have to have several masks for a multilayer board. For a single sided board typically, you will have one electrical layer and therefore a single mask, but at the same time, mask for a solder mask will have to be created and a mask for a legend prints have to be created.

Now, the next step is to transfer this image onto the surface of a copper clad laminate. We go through a process called imaging or printing. Here, we are going to use materials like a photoresist; obviously, the copper clad laminate will be there, where you have the copper on the surface and then we are going to use other chemicals and materials to make sure that the image is transferred onto the copper surface.

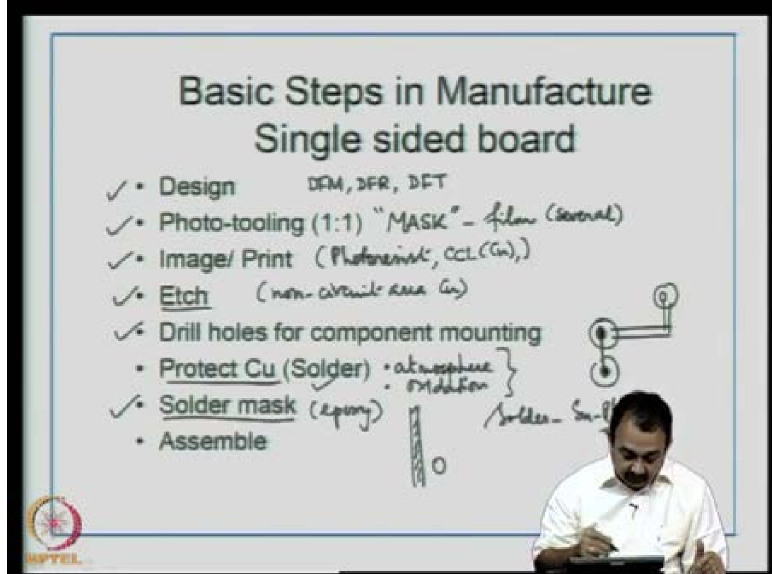
(Refer Slide Time: 03:03)



Now, once this circuitry is imaged onto the printed wiring board surface; we are interested in retaining the copper on the circuit areas. The non-circuit areas copper will have to be removed. How do you remove that? You do a process called etching. Etching is a process to remove copper from unwanted areas; that means, in this case non-circuit areas. Etching is a very general process; you can etch stainless steel using a specific etchant. In this case, copper we use specific etchants. We can also etch plastics by using certain chemicals.

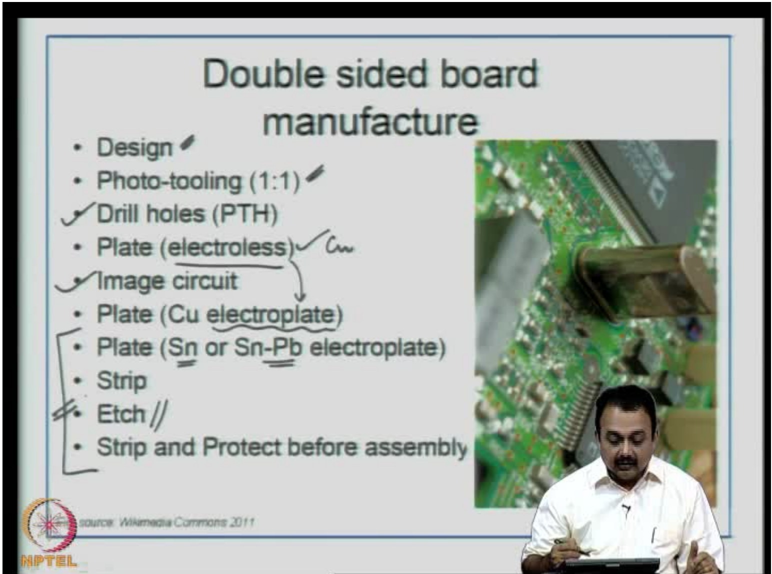
Now, after that you drill holes for component mounting; that means, in the circuit you have the pads generated then you have the tracks. Now, assume that there is a component to be mounted in this location; so you drill a hole here; you drill a hole here and that is suitable for PCB component mounting, now the copper that is generated (Refer Slide Time: 04:25). The circuit copper that is generated will now have to be protected because, the copper will interact with the atmosphere very fast and it will undergo oxidation. Therefore, you have to protect copper, how do you protect copper? You use solder material. In this case, solder material is basically tin and lead but, as I said today we are in the lead free realm of manufacturing therefore, we have to look at alternatives for lead. So, we will come to these aspects much later.

(Refer Slide Time: 05:33)



Now, we can also protect the board by using a solder mask material. I told you, this is an epoxy resin and this material is basically used to protect two adjacent tracks or a track and a pad close together and so on. So, the space between tracks is basically covered with a solder mask material. Now this board is ready for assembly.

(Refer Slide Time: 05:37)

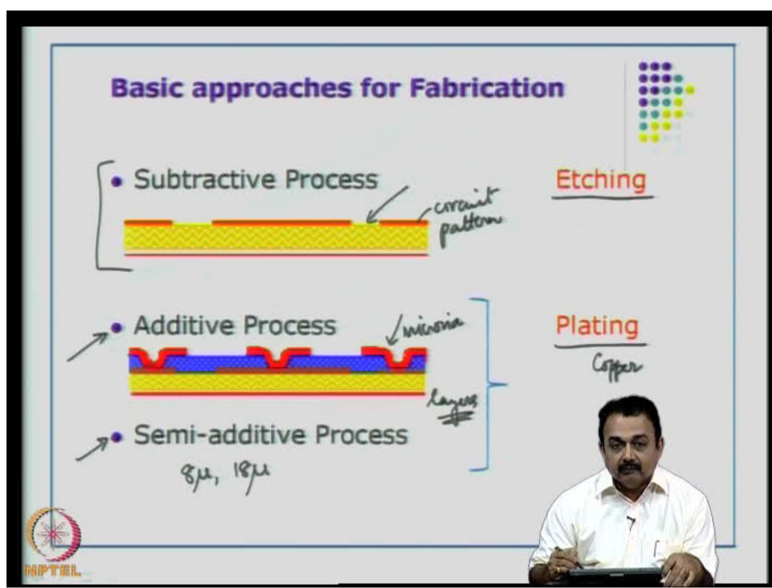


The double sided board manufacture process steps very broadly includes the design aspects, then you have the photo tooling to create the various layers; in this case, two electrical layers and then the other additional photo tool or the mask for the solder mask, top and bottom legend print for the component mounting side and so on.

Now, you have to drill the holes first, because you are going to plate the through hole. Therefore, making the through hole is an elaborate process; you have to make the non-conductive hole walls conductive. For that, you use a electroless copper plating, because you are going to make the non-conductive board area inside the hole wall conductive. So, you cannot do electroplating directly; we go through a procedure called electroless process. We will see about this process in much detail as we go by.

Then, you do the imaging of the circuitry copper areas; then you build up the copper with electroplating copper, so electroplating can only be done after you have done electroless. Now, you can protect the copper with tin or tin lead electroplating; so that the nascent copper is protected; after that you can remove the unwanted copper by doing the process of etching. Finally, if you want you can also remove this tin material - if you do not require it and you can have the copper protected with solder mask material.

(Refer Slide Time: 07:49)



So, depending upon the process that you require, you can modify these sections and add tin and finish the process, etch away the unwanted copper and then the board goes for assembly. The basic approach for fabrication has been mentioned earlier, but again I would like to emphasize, since we are in the process of describing the various steps in manufacturing.

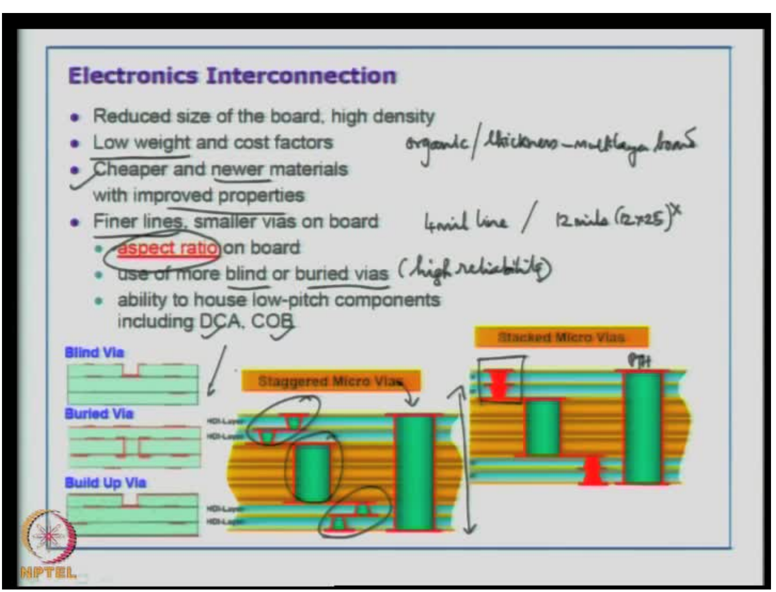
Essentially, two important processes have to be identified: one is etching, the other is plating. Etching is removal of copper and plating is addition or adding copper to the surface of the board. Now if you look at this picture, what I have described here is, a

subtractive process, where you are talking about removal of copper from this area specifically to create a circuit pattern.

Here, etching becomes the focus; it really decides on how the feature sizes of the copper should be. We are taking initially a large amount of copper and then removing unwanted copper. So in this process description, you will not really focus on plating, but in certain processes like semi additive process or an additive process, you may have to add or focus on plating copper.

How do you do that? That means, you start with, let us say 8 micron of copper or 18 microns of copper, as the starting material and build the required thickness; let us say up to 32 microns by electroplating process. The sequential build up technologies which we are going to describe will essentially rely on electroplating process. Electroless plating is also a method of adding copper.

(Refer Slide Time: 10:02).



So, the semi additive and additive processes to create these Micro Vias and then additional layers on a printed wiring board will be done by plating process. In the electronics interconnection of a printed circuit board, what are the focal points? We are interested in the reduced size of the board obtaining high density, which is what the designer is doing and the manufacturing should be able to complement these kinds of requirements for global high density interconnects.

So, reduced size of the board, high density of the board, low weight and cost factors. How do you make sure that you have a low weight? You have to choose organic materials, which are low in weight compared to a ceramic substrate and also, you have to

look at the right thickness of the substrate that is required. As a designer, you do not want to add more organic material as a substrate core - core substrate thickness, because that is going to increase the weight of the board. So, you can interact with manufacturer and see, what kind of thicknesses of the final board can be achieved especially, if it is a multilayer board.

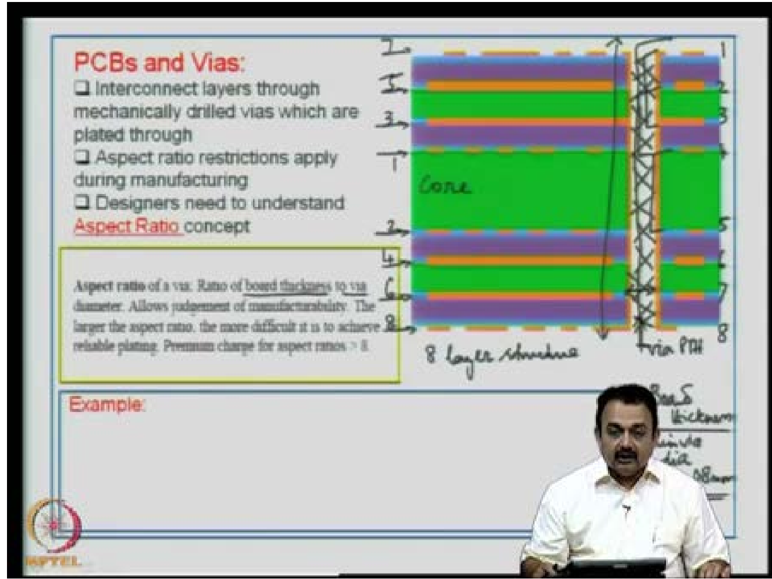
Then, we have to look at economics. Obviously, you should go in for cheaper but better. Cheaper does not really mean, you can use any cheap material for high performance applications. You have to make sure, you work on the economics and use new materials that are cost effective and which have improved properties, what are the properties? Dielectric constant, glass transition temperature, low moisture absorption and low C T E - Coefficient of Thermal Expansion.

So, these are some of the properties that as a designer you will look for. As a manufacturer, you must be capable of having a line in your workshop to produce fine lines. So, if I say I am a manufacturer and if somebody asks me, what is your fine line capability? For example, if I say that I can do four mil lines definitely; I am on the leading edge in the market today but, if I say, I can do only twelve mils that is - 12 into 25 micron - then people are not going to do business with me.

This is the kind of global leading edge manufacturer requirement, for which you have to enhance the chemistry and the technology capabilities in your industry. Then, both designer and manufacturer should be aware of a very important aspect called aspect ratio of the board, very shortly I am going to describe that. You must be able to use more blind or buried vias in your design and as a manufacturer. The manufacture should be able to turn out manufacture blind or buried vias with very high reliability.

The design should be able to house low pitch components including direct chip attach chip on board. If you look at this figure, which we have seen earlier in some other context; this is a cross section of a board and you can see, this is a via and you can see these are the Staggered Micro Vias. This is a via in the inner core structure; this is again a staggered via; this is a Stacked via - you can see one via sitting on top of the other - this is a plated through hole via and so on. So, this is again another method of describing a buried via and a blind via (Refer Slide Time: 13:57).

(Refer Slide Time: 14:05)



Now, this is a cross section you see, which describes or which shows the various interconnect layers. You can see, this is the core structure; you can see copper 1 and copper 2 here, then you can see copper 2, copper 3, copper 4, copper 5 then you have copper 6. 1, 2, 3, 4, 5, 6, 7, 8 those typically, this is an eight layer structure, so you have 1, 2, 3, 4, 5, 6, 7 and 8 layer copper structure. In this you have via a which is plated through hole, connecting top electrical layer to the bottom electrical layer.

In some other notation, you can call this as 1, 2, 3, 4, 5, 6, 7 and 8. So typically, now you can say layer 1 and layer 8, electrical layers are connected by a mechanically drilled through hole structure. Using mechanical drilling, you can connect any of the inner layers; you can connect layer 1 to 3; you can connect layer 2 to 4 typically, layer 4 to 5 only and similarly, at the bottom.

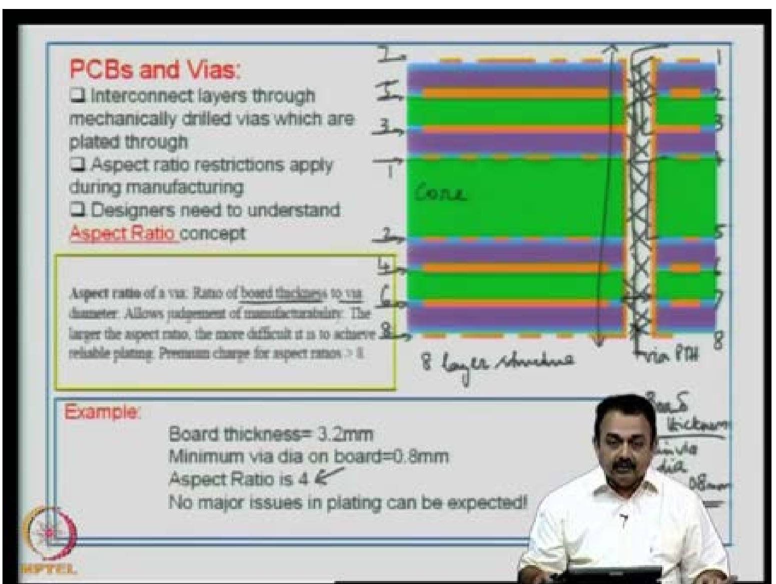
So, this depends on the design and your assignment for ground plane and so on. These are the mechanically drilled vias. Aspect ratio restrictions apply during manufacturing and the designers need to understand the aspect ratio concept. So, with this background what is an aspect ratio? It is the ratio of the board thickness to the via diameter. You have aspect ratio is equal to the board thickness by minimum via diameter on the board (Refer Slide Time: 16:17). So, in a board typically you would have used different dimensions of a via; it can vary from 0.2 mm to let us say 0.8 mm.

If in a board, you have used all these dimensions then for calculating the aspect ratio, you have to choose the minimum via diameter that gives a lot of indication, that gives a lot of information about the possibility of the board getting manufactured or not. Also it

indicates some kind of a reliability assessment that you can make. As a designer, as well the manufacturer, will be able to tell you whether with this aspect ratio, he can manufacture the board with high reliability or not.

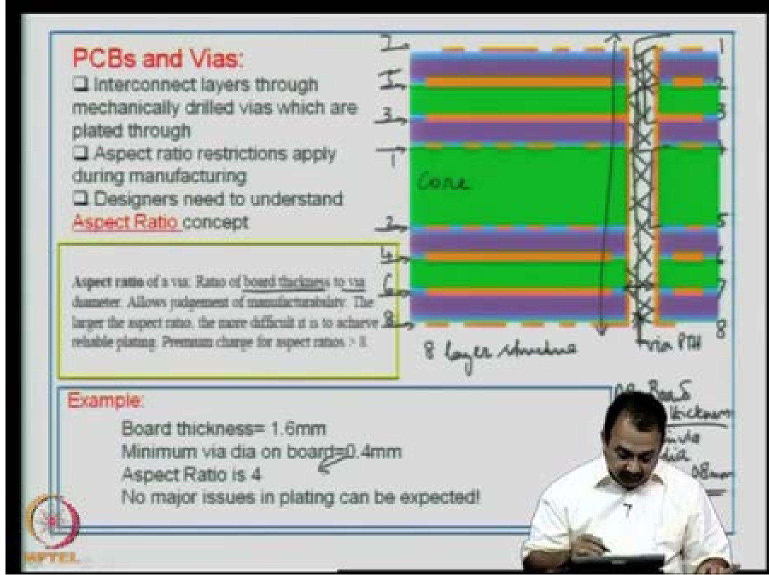
Let us take an example, larger the aspect ratio the more difficult it is to achieve reliable plating. That means, if you look at a structure like this and if the thickness of the board is very large and then, you have a very small dia of the hole in the structure, the plating process has a requirement that it wets the entire hole wall surface. If the hole is very small, then the wetting process inside the hole wall is going to be very tough. That is why they say, aspect ratio has to be normal. If it is abnormal like, having an aspect ratio of 8 or more, you can imagine that it is going to be a very small dia of the hole and the reliable plating inside the barrel of the hole is going to be extremely difficult, even with the best of chemistries and technologies.

(Refer Slide Time: 18:19)



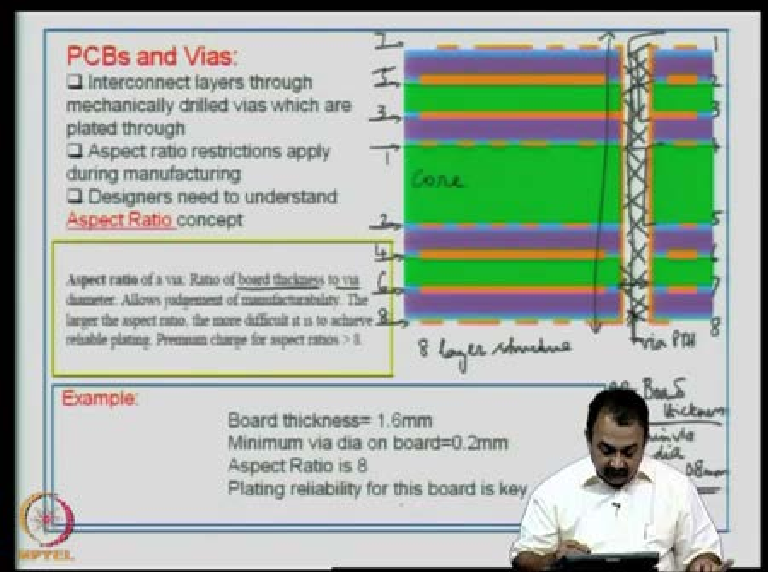
I will give you an example, how to calculate aspect ratio and what to look for in an aspect ratio as a designer? If you look at the board thickness, for example it is 3.2 mm in this case, the minimum via dia on the board is 0.8 mm. Therefore, the aspect ratio is 4. Now typically, with this kind of a scenario no major issues in plating can be expected; it is very easy to plate a hole wall - that is 0.8 mm in size of a thickness of 3.2 mm, so you can accept this board for manufacturing.

(Refer Slide Time: 19:05)



Now, if you take a situation where you have the board thickness to be 1.6 mm, the minimum via dia on the board is 0.4 mm. Here again, the aspect ratio will be 4 but, note that your board thickness has reduced compared to the earlier one and then your via is the same as in the previous example the aspect ratio is the same. In the earlier example it was 3.2 mm and 0.8 mm drilling here both have reduced but, look at the aspect ratio it is 4 therefore, no major issues in plating can be expected;0.4 mm on a smaller thickness of the board can be achieved, plating on this hole can be achieved fairly well with the current technologies.

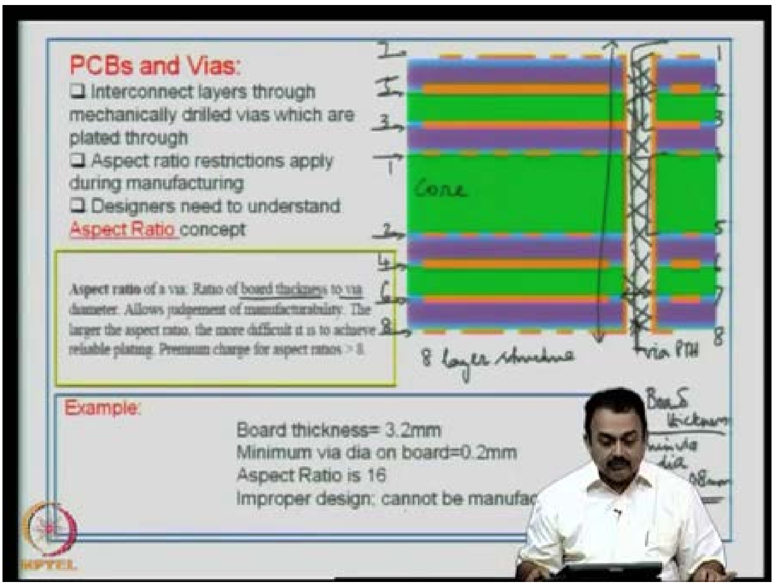
(Refer Slide Time: 20:07)



If you take another example, where the board thickness is 1.6 mm but, your dia of the vias are 0.2 mm - 0.2 mm mechanically drilled vias - the aspect ratio has increased to 8. It is going to be very difficult to achieve very high reliability for these kind of boards.

The plating possibly can happen but, you have to spend a lot of time on the quality check and reliability for this board. Hence, the cost for this board is going to be enormous. The manufacturer definitely will charge premium rates for you; if the aspect ratio is going to be 8 and above.

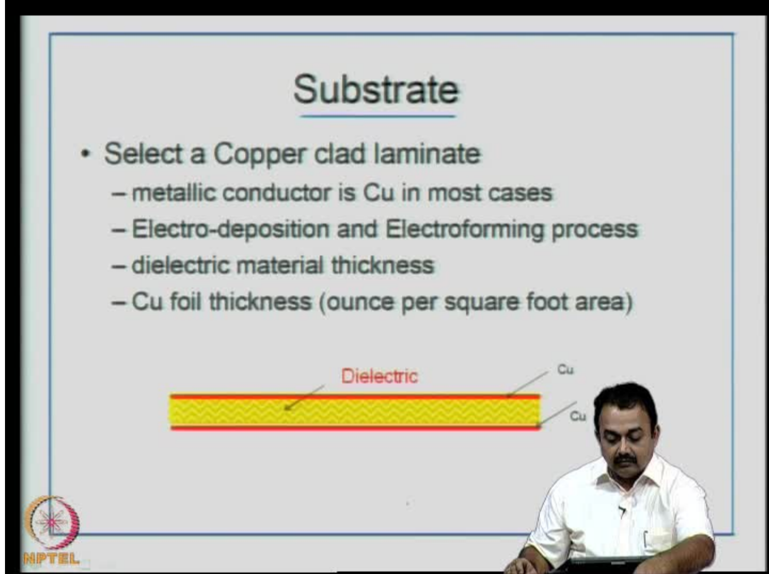
(Refer Slide Time: 20:55)



Another situation, board thickness is 3.2 mm minimum via dia on the board is 0.2 mm aspect ratio is very high 16; it is an improper design first of all and it cannot be manufactured with any kind of reliability built around via structures, because it is a fairly thick board and drilling a mechanically drilled hole of 0.2 mm through the 3.2 mm structure itself is a big challenge, because if you talk about 0.2 mm drill bit, they are very thin, flimsy, fragile and it produces a lot of heat during drilling and entering through this 3.2 mm structure and coming out without breakage of drill bit is a big challenge.

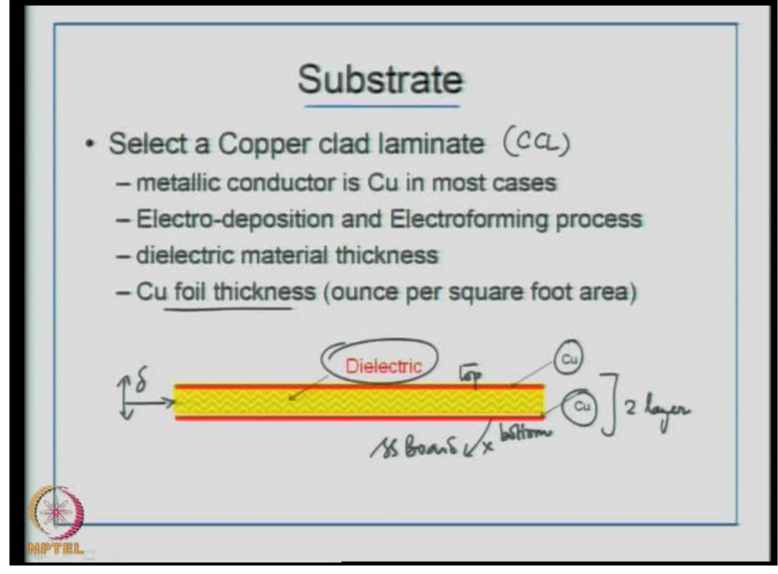
Because, if you work at lower drill bits sizes you have to increase the rpm of the drilling machine enormously. This is always a challenge in drilling; it is a challenge in plating. Therefore, this kind of design should be avoided. So, please remember aspect ratio is equally important to be thought about for a designer, as well as for a manufacturer. As a designer, we should ask the manufacturer about aspect ratio capabilities in the manufacturing line.

(Refer Slide Time: 22:26)



Now, we look at what is a substrate for a printed wiring board. In the process sequence, first we have to select a copper clad laminate. If you can look at this sample here, what you see here is a laminate structure (Refer Slide Time: 22:51); we can see copper on one side and again copper on the other side. This is a very rigid board and pressing it; it is not bending; it is not flexible; it is a rigid board out of this comes the printed circuit board that we see with the solder mask and the assembly, both sides can be used for mounting. This is a high density board and here you can see a double-sided board with through holes. So, these are the kind of structures that you can use. We are now going to talk about this laminate and its cross section.

(Refer Slide Time: 23:33)



If you look at a copper clad laminate cross section, which I have shown here, you will see that there is metallic conductor and copper is mostly the conductor used in printed circuit boards, the reason for using copper are much more than availability. The electro-deposition and electroforming processes are the methods used to create a copper clad laminate. When you talk about a copper clad laminate or a CCL in short - the terminology used in this industry. You will see that there is a copper layer on top, a copper layer on board for a single sided board typically, you will not have this copper layer; this is typically for a two layer structure (Refer Slide Time: 24:29).

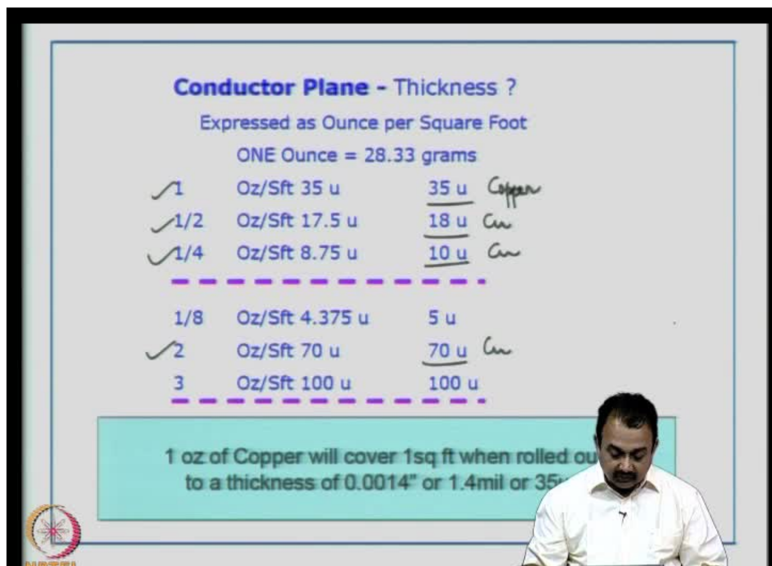
The sandwich material that you see here is the dielectric material. Now, you are worried about the dielectric thickness, very important; the thickness of this material decides a lot of things including the electrical performance of your circuit. Then, we are also worried about copper foil thickness that is basically laminated onto the top surface as well as the bottom surface.

(Refer Slide Time: 25:13)

I want to show you as an example, a copper foil; see here, this is a copper foil. Now the copper foil can also come in very thin sheets, you can see here. This is a copper foil you need to handle it very carefully. These kind of materials are basically laminated to these board structures to get a copper clad laminate. This is an example of a dielectric material, what I am showing here is a dielectric material. So, you can imagine a copper foil and a dielectric material sandwiched to produce a copper clad laminate like this (Refer Slide Time: 25:46). So, a substrate will contain or you can call it as a copper clad laminate, it contains a metallic conductor. The metallic conductor is produced by electro deposition

or electro forming; it contains a dielectric and the alignment of the copper depends on the thickness that is required for that particular application. In the market, you get different varieties of thicknesses of copper and the total thickness of the copper clad laminate.

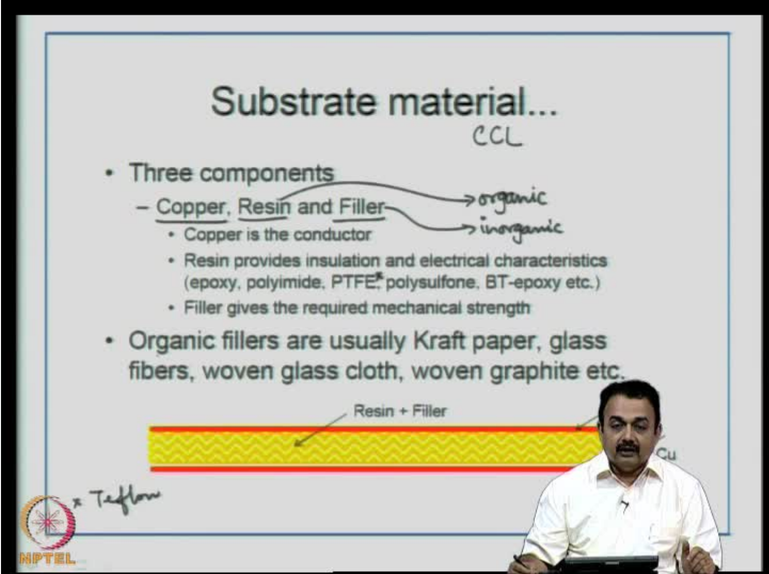
(Refer Slide Time: 26:42)



Let us look at what is the terminology used in defining this copper clad laminate. We need to identify copper clad laminate based on the thickness of copper that is used. For example, if you say that I have bought a copper clad laminate with 35 micron; copper 35 micron defines the copper thickness. It is known as one ounce copper; that means, one ounce which is 28.33 grams, one ounce per square feet of the total laminate material is known as 35 micron foil or copper.

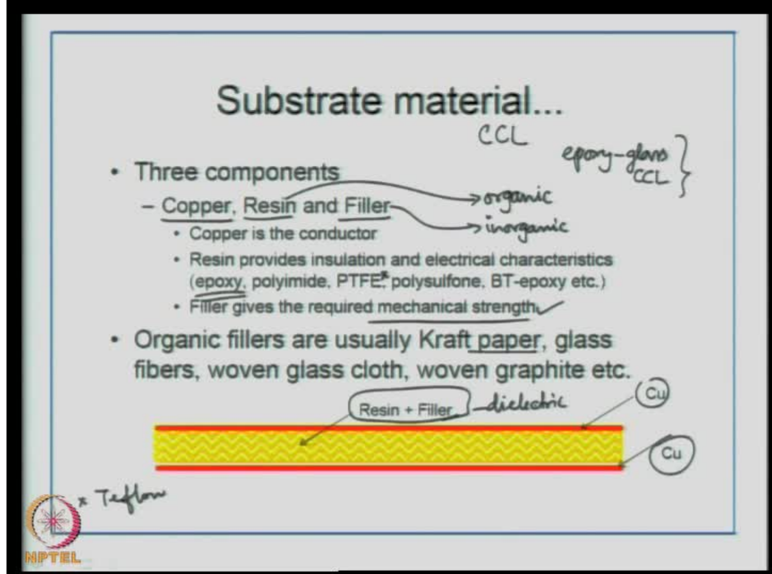
Then, you have half ounce copper clad laminate which contains 18 microns of copper. Quarter ounce which contains 10 micron of copper. For special applications like power applications, you tend to use two ounce copper where you have 70 microns of copper and so on. One ounce of copper will cover one square feet when rolled out to a thickness of 1.4 mil or 35 micron. This is the basis for defining or using standard terminologies like 35 micron copper clad laminate or 18 micron copper in a copper clad laminate and so on.

(Refer Slide Time: 27:56)



So, to emphasize once again, there are three components in a substrate or a copper clad laminate. There is a copper with a defined thickness, there is a resin and there is a filler. So the resin here, defines an organic material and we use filler, which is typically inorganic in nature. You cannot use just resin and copper because, it will be very flexible; it will not be rigid and you cannot use in mounting applications that we normally use for a PCB. The physical and chemical properties are different with a filler and without a filler. So, Copper is the conductor resin provides the insulation and electrical characteristics. For example, epoxy, polyimide, poly tetra fluoro ethylene PTFE or it is also known as Teflon, polysulfones, BT-epoxy that is a modified epoxy, bismaleimide tetra azine epoxy and so on.

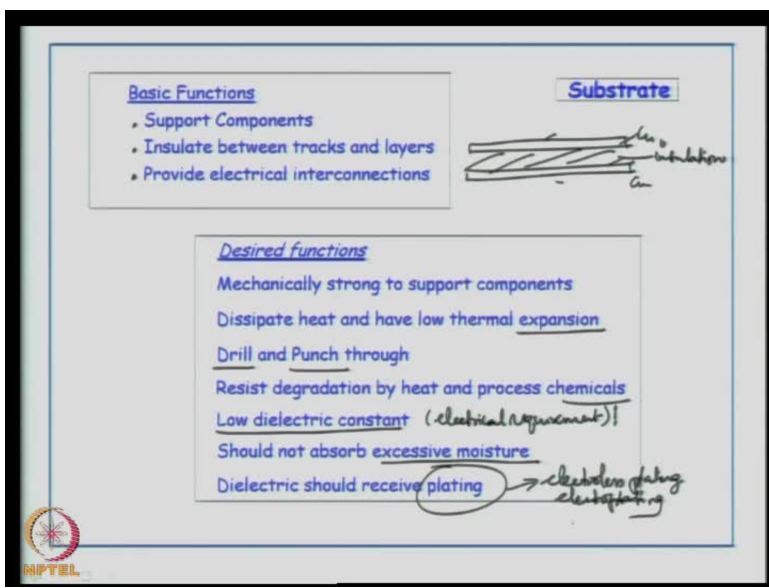
(Refer Slide Time: 29:49)



There are scores of material that are available that can be used as a resin in your copper clad laminate. For example, the material that I showed you here, if you can focus on this is an epoxy resin; it contains a filler which is basically glass (Refer Slide Time: 29:39). So this material, if it is used in a copper clad laminate will be known as copper clad laminate which contains a epoxy resin. So, I will call it as epoxy glass copper clad laminate or you can call it as glass epoxy copper clad laminate.

So, that is the kind of classification. If you use epoxy and some other material filler like a Kraft paper. So, fillers can be used like Kraft paper, glass fibers, woven glass cloth, woven graphite etc. In some cases, inorganic fillers are also added to modify the properties of the substrate material. The filler gives the required mechanical strength, so you can imagine a foil of copper with a resin will not give not give the mechanical strength. The fillers required to be added to give the mechanical strength for various applications. Once again, you look at the structure, you have the copper at the top, you have the copper at the bottom an then in-between you have the resin plus the filler which is again now the dielectric material.

(Refer Slide Time: 31:06).



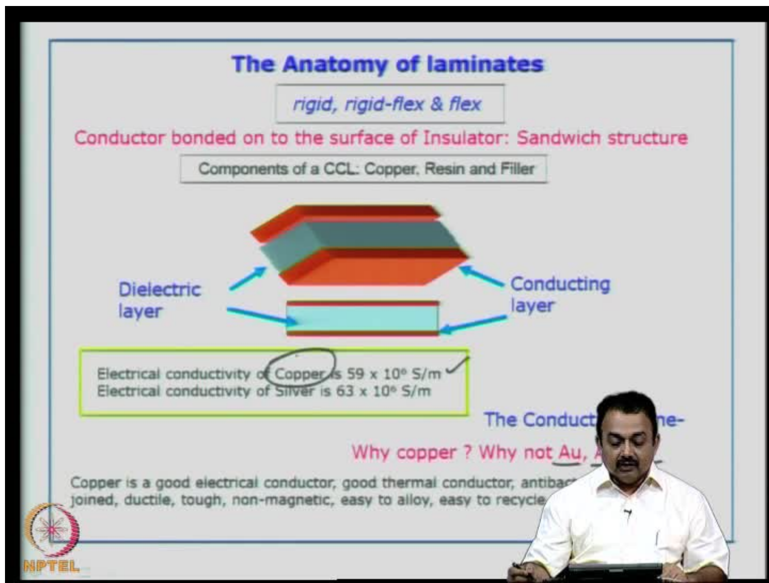
What are the functions of a substrate? The basic functions of a substrate is, it supports the components. When you mount - when you finish a PCB, you are mounting the components; so it is very rigid and it supports the components.

It supports the components to be placed and interconnected by using copper tracks. It provides a very good insulation between the tracks and the layers. Imagine a dielectric material; you have copper here and then you have the copper here at the bottom.

So, if you run tracks on top and bottom this provides the insulation and it also enables the electrical performance a very good electrical performance for your system, by minimizing cross talk and other electrical parasitics. It provides electrical interconnections by routing the copper. What are the desired functions? Mechanically strong to support components, dissipate heat and have low thermal expansion, because if it is going to expand on cycling - continuous thermal cycling - during the operation heating, cooling, expansion, contraction then there is going to be stress built on the board at the solder joint and also at the copper tracks. So, you will end up with a war page on the board which will end in a electrical failure.

It should be able to be drilled and punch through, so it should have the characteristics of accepting the drilling - mechanical drilling - of various sizes and in some cases punching is also done. Resist degradation by heat and process chemicals, because as you will see the process steps for making multilayer boards. The board goes through various chemicals and therefore, it has to withstand all of these chemicals. It should have low dielectric constant, a very important electrical property - electrical requirement. It should not absorb excessive moisture, by nature epoxy absorbs moisture but, there should be a limit. So, we look for different materials also, if you are worried about excessive moisture which affects the electrical performance and dielectric should receive plating.

(Refer Slide Time: 34:28).

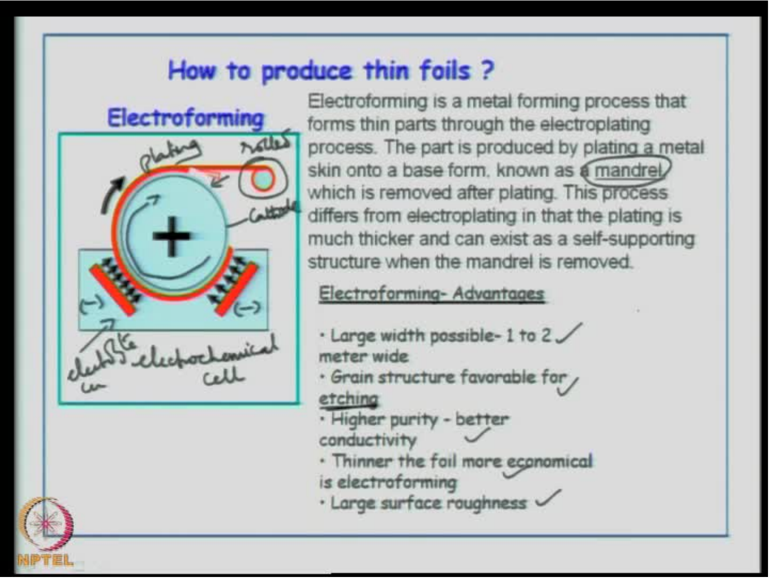


Typically, we are talking about electroless plating followed by electroplating. So, there should be good bonding between the dielectric and the added copper. This is a very essential key point that manufacturers will look for. So, laminates can be rigid, it can be

rigid-flex combined or it can be flex. You can create a rigid circuit. You can create a rigid-flex PCB or you can create a completely flexible PCB. A conductor is bonded onto the surface of a insulator and that is how you get the sandwich structure.

Now, we are very clear about the components of a copper clad laminate. I have once again shown you a cross section here. The electrical conductivity of copper is very high; silver is much better but, we do not use gold or silver or aluminum in the printed circuit board industry, essentially because copper is a good electrical conductor, good thermal conductor; it is antibacterial, easily joined, ductile, tough, nonmagnetic, easy to alloy, easy to recycle, catalytic and so on. It is compatible with all the chemical processes, you get very economically available etchants for copper and the electroplating of copper is well known. If you look at various properties, we go in for copper and not other metals in the PCB industry and much more it is easily available. So, the availability of copper is much better, cost wise also it is the on the plus side.

(Refer Slide Time: 36:17)



Now briefly, I will talk about the process of producing copper clad laminates. So, in this the first thing is, how to produce thin foils? So essentially, you first require copper foils which will be bonded onto the dielectric material and that is how your laminate structure comes up. The process of electroforming is used to produce foils.

You can see here in this figure, there is a drum and you can see typically, it is a electrochemical cell - that most of you are aware of electrochemical cell, basic electrochemistry - you have the anode and the cathode and then you have the electrolyte here, which contains the copper and this drum acts as the cathode, which will receive the

copper and the drum is rotating slowly coming out through the electrolyte and it receives the plating (Refer Slide Time: 37:04).

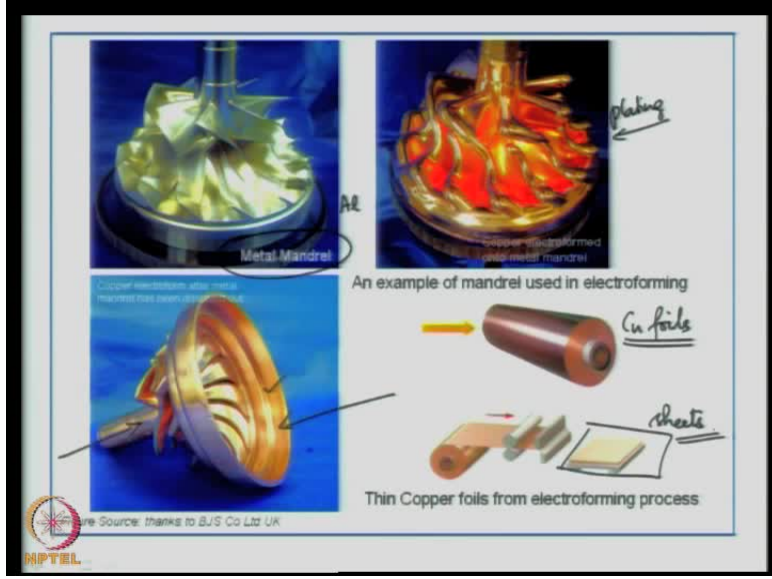
This is basically a plating process, it receives the plating and it is rolled here as a foil. It is slightly different from the set of that you would have seen for a electrochemical cell. Electroforming is a metal forming process that forms thin parts through the electroplating process. The part is produced by plating a metal skin onto a base form. In this case, the base form is the huge cylinder, because we want to produce foils in large dimensions, in large sheets, for large volume manufacturing.

This base form is known as a mandrel. This is a important requirement, because the plating or the plated metal will take the shape of the mandrel. So, mandrel is essential for electroforming and this mandrel will be removed after the plating is over. Once you remove the mandrel, you will get the foils in this case. Here, you can see that it is detached from the base cylindrical drum.

Now, how does this process differ from your normal electroplating? In this case, the plating is much thicker and these current conditions are different; it can exist as a self-supporting structure, when the mandrel is removed. So, one of the process for producing thin foils for PCB is electroforming and that is why we are discussing this. So, you can fine tune electroforming to the kind of thicknesses that you want for PCB applications.

So, in electroforming the advantages are large width of the material to be plated is possible like 1 to 2 meters and continuous roll structures are possible. Grain structure during plating is favorable for the etching process, because as you know, when we do the etching process in PCB manufacturing; etching is a very essential requirement you cannot avoid etching, but at the same time etching has to be perfect. You cannot over etch a material, you cannot under etch a material. Therefore, the grain structure has to provide a very quick etching without affecting the non-circuit areas or the pattern that is intended to be etched. High purity, better conductivity, thinner the foil more economical is electroforming and it has got a large surface roughness. So, all of these are some of the advantages listed especially in the context of PCB.

(Refer Slide Time: 40:20)



If you look at this picture here, which will explain to you what electroforming is about and mind you, electroforming is not only used for PCB applications, it can be used for decorative jewelry, automobile machinery parts, various mechanical parts that need to be generated.

You can see for example, in this the first figure on the left is a metal mandrel for a automobile structure or a mechanical part; it has got very intricate structures, which is very difficult to normally mechanically mill. If you are going to mill every part like this, it is going to take a lot of time. The best solution is electroforming; so if you make a mandrel for a structure which finally should be this (Refer Slide Time: 41:17). So, this is a mould like structure, you make out of aluminum for example, in this case which this company has done, then plating is done on this mandrel.

This describes the plating; this figure shows that complete electroplating has been done using electroforming process onto the metal mandrel. Now, you can release the mandrel by dissolving out the base aluminum. Now, you will get a structure like this which is basically what is required. So, you will be spending a lot of time on creating the mandrel but, if you look at the post mandrel fabrication, you can see that the intricate - the small detailing on the part - has been nicely reproduced during the electroforming process. Similar to this process, these copper foils can be generated, as I explained in the previous slide figure and then you will get these rolls, which can then be cut into sheets of various dimensions and sent to the manufacturing. So, copper foils are produced by electroforming process.

(Refer Slide Time: 42:43)

The four important characteristics of an organic substrate need to be known here. What are these? One is dielectric constant, second is coefficient of thermal expansion, glass transition temperature and water absorption.

For example, if you talk about the common FR-4 glass epoxy; flame retardant, fire retardant material that is why it is known as FR and 4 is basically a classification. Classification by a standard body and this is most commonly used PCB material, you can see the dielectric constant varies from 3.6 to 4.2, CTE is between 13 and 18, Tg today you get from 125 to 190, in fact that is the glass transition temperature. Transition temperature is the temperature at which the polymer in the material begins to soften; it begins to become soft and it enters into a glassy state.

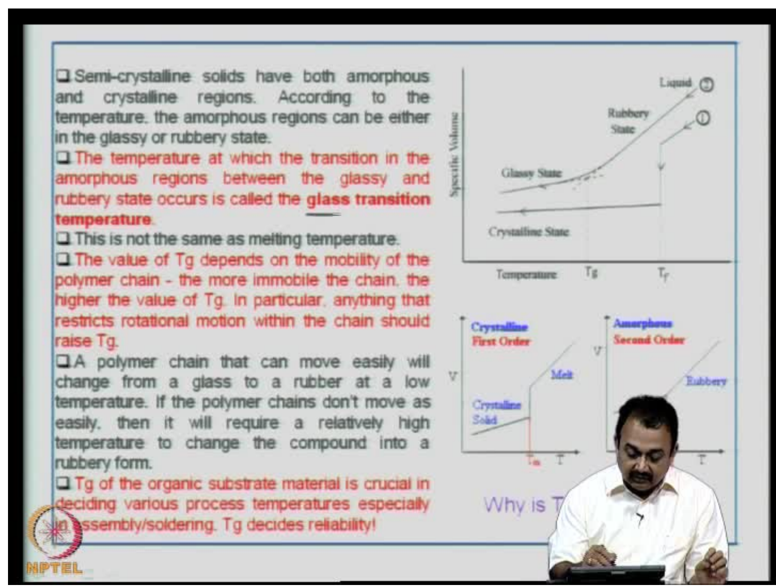
So that temperature is very critical because that can change the properties of your core substrate. If the property is not reversible then the board is totally damaged because of excessive thermal load. So, if you know the Tg of a material then you can work around the Tg even during the soldering process.

Then, the other important property is water absorption 0.10. So, for other materials like a BT epoxy which is bismaleimide triazine epoxy, you can see the comparison; water absorption is fairly low. If you look at polyimide, very good material, low CTE, you can get at high Tg material but, you can see that the water absorption is fairly high. So, this is one of the disadvantages but, the electrical performance is much better compared to an FR-4.

Similarly, if you look at teflon - poly tetra fluoro ethylene fluoro polymer, you can go up to 327; low moisture absorption and low dielectric constant but, you have to look at economics, how many people will prefer PTFE in terms of cost compared to an FR-4 which is abundantly manufactured because of low cost. Then, other things like nonwoven material can be used in comparison to a woven structure and you will end up with different properties.

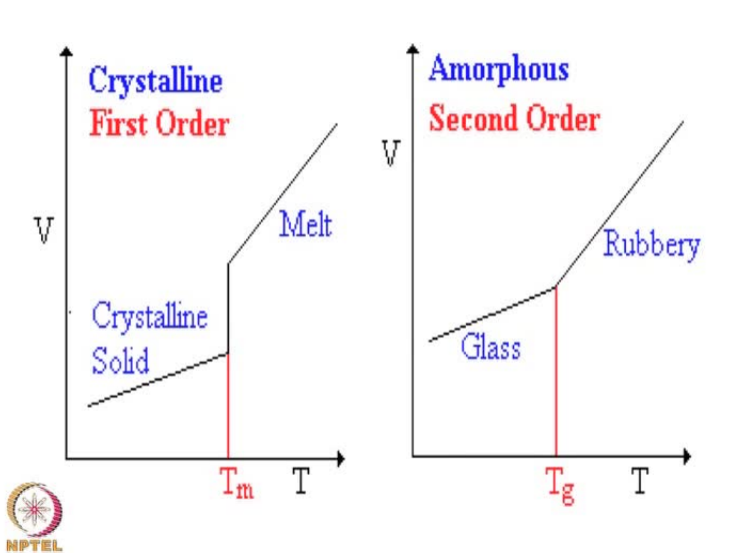
So the glass transition temperature is a very important parameter that as a designer you have to look for - the reason why I put this table is that as a designer, one should be able to select a laminate. I have mentioned before, a designers job is also to select a laminate. So, look at these properties and decide which one you want to use for your electrical performance rating that you require.

(Refer Slide Time: 46:19)

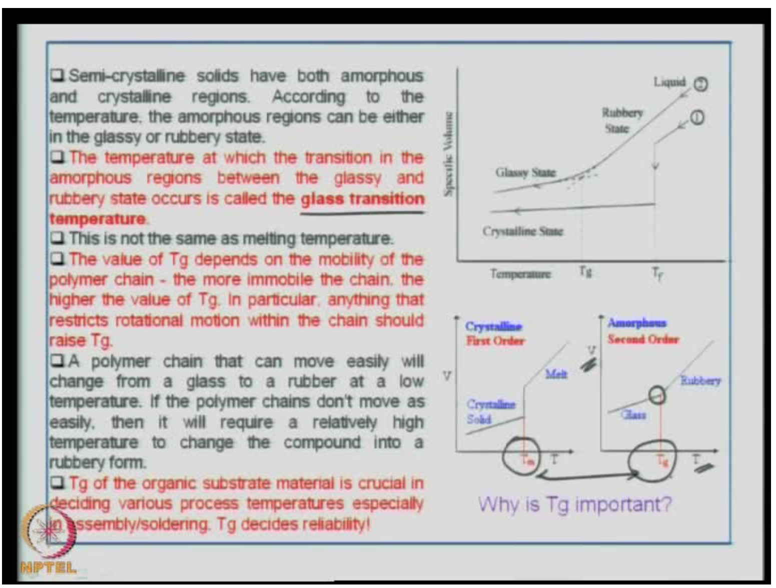


The Tg is important because the laminate will see a series of heat shocks soldering, manual repair, manual soldering and so on. So, I briefly talk about Tg because it seems to be a key issue in laminates. The first point is that semi crystalline solids have both amorphous and crystalline regions. According to the temperature, the amorphous regions can be either in the glassy or rubbery state.

(Refer Slide Time: 46:55)

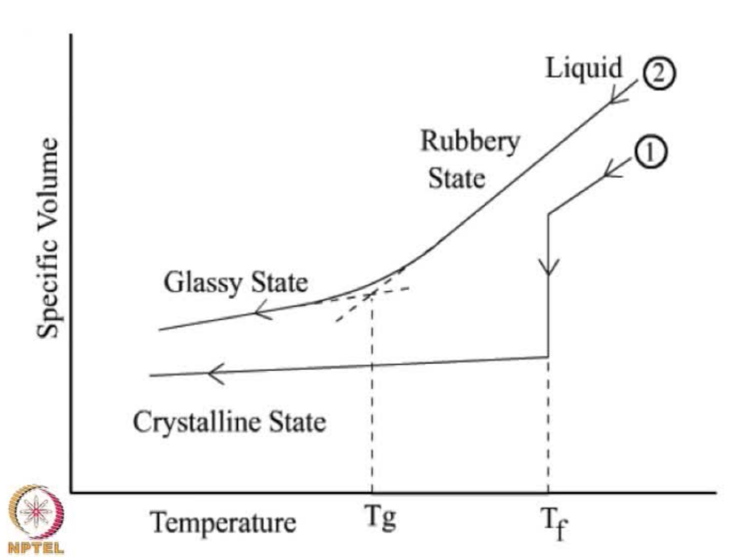


(Refer Slide Time: 47:04)

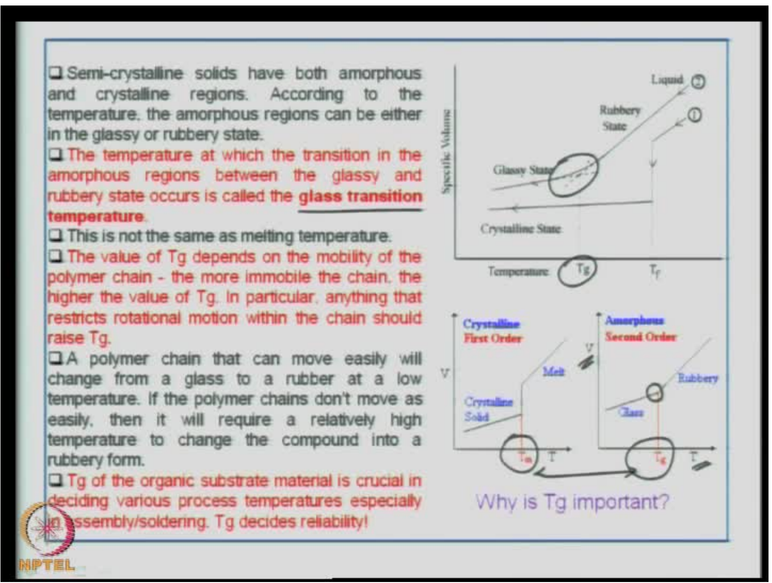


The temperature at which the transition in the amorphous regions between the glassy and the rubbery state occurs is called the glass transition temperature. For example, if you look at this figure on the right side, there is a melt region and a crystalline solid region and this is the melting point - specific volume versus temperature and then you can see here, Tg defines the very narrow region between the glassy and the rubbery state. This is a second order phenomenon that you will see, if you plot specific volume versus temperature.

(Refer Slide Time: 47:49)



(Refer Slide Time: 48:01)



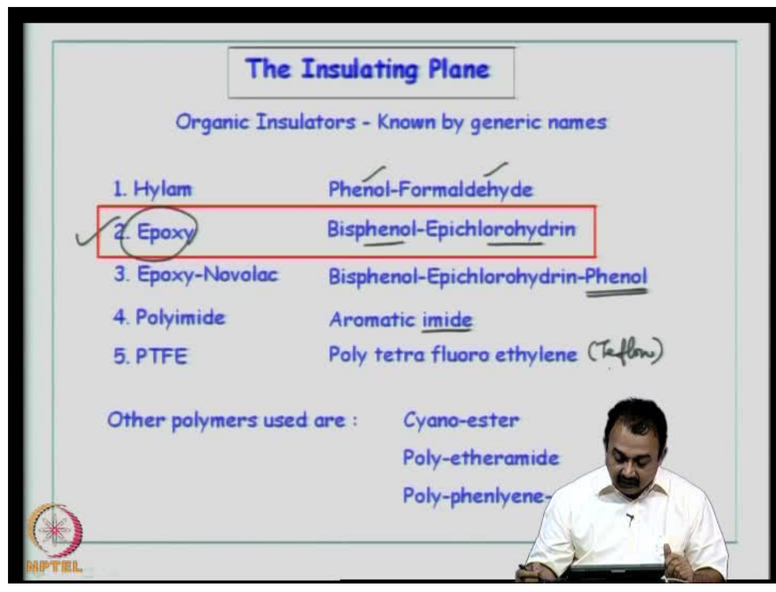
Now, this is different obviously from the melting temperature but in some cases, the properties are reversible; in some cases, it is not. Therefore, if you are very particular about the order, the crystal structure in a particular material, then you have to carefully weigh the options regarding Tg. So this figure also gives you a combination of glassy state region and rubbery state region and it also shows the crystalline state then you have the. This region is known as the glass transition temperature and this is the freezing point or the melting point.

So, the transition that takes place in the amorphous material between the glassy and the rubbery state is known as the glass transition temperature. The value depends on the

mobility of the polymer chain, so basically, we are talking about polymers and the Tg will vary depending on the polymer structure. So, the more immobile the chain is the higher value of Tg. So, in particular anything that restricts rotational motion within the chain should raise the Tg. In an amorphous structure, you may have certain branches in the polymeric chain, which will not be affected during a thermal load; which will not be mobile; which will not rotate in the chain structure. Therefore, those materials will exhibit higher Tg.

A polymer chain that can move easily will change from a glass to a rubber state at lower temperature. If the polymer chains do not move as easily, then it will require a relatively high temperature to change the compound into a rubbery form. So, Tg of the organic substrate material in our case is very crucial, because various process temperatures especially during assembly and soldering will have an impact on the organic substrate. Therefore, Tg in some sense decides your reliability of the board.

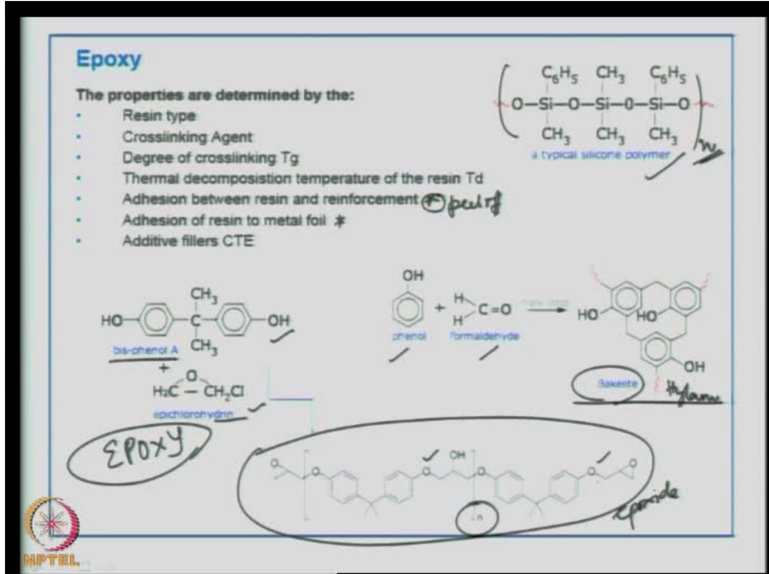
(Refer Slide Time: 50:16)



So, even when you do machine based assembly when you do a thermal profiling of a board, you will take into account the glass transition temperature, because you do not want to spend during the ramp up or the heating process a lot of time around the Tg. Now, we talk briefly about the insulating plane or the insulators. We talked about epoxy, what is epoxy? We use this very simple term epoxy but, epoxy basically is a combination of Bisphenol A plus Epichlorohydrin, these are the two organic chemicals that are used.Hylam for example, is a combination of Phenol plus Formaldehyde. Epoxy-Novolac is a combination of Bisphenol Epichlorohydrin plus Phenol. Polyimide is

basically an Aromatic imide structure. PTFE is Teflon, it is a commercial name polytetra fluoro ethylene. So all of these are organic and that is why we call these as organic substrates, because it contains an organic insulator and these organic substrates have special properties that take care of your electrical parasitics.

(Refer Slide Time: 51:27)

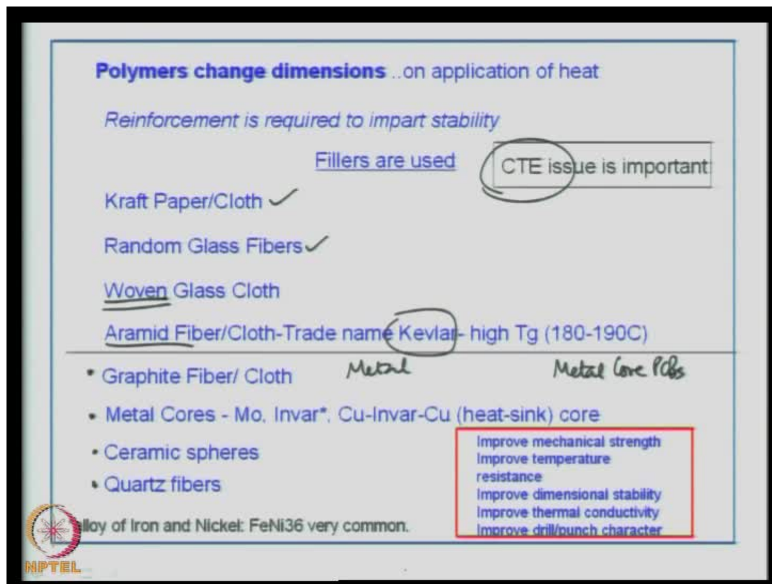


Other polymers used are: Cyano-ester, Poly-etheramide, Poly-phenlyene oxide very briefly without going is into too much of chemistry, I will describe what an organic insulator does. For example, in an epoxy the properties are determined by the resin type, the crosslinking agent; the degree of crosslinking will determine the Tg of the total system, you are also worried about the thermal decomposition temperature of the resin Td; Td is different from Tg. Adhesion between the resin and the reinforcement, I told you very important, because that once again, if there is a peel off then your board is not reliable. Adhesion of resin to metal foil, also very important. Additive fillers will improve the CTE, will modify the CTE.

In the previous slide, I talked about for example, here I talked about hylam phenol formaldehyde. You have the phenol plus the formaldehyde to give you a hylam, commercial name bakelite structure. This is a typical silicon polymer, you can see the chains are repeated in the structure monomers or repeated monomer structures to get a polymeric structure. So, you can have the length varying, so the polymer weight is determined on the number of chains number of monomers determined in the polymeric structure.

So, the molecular weight depends on the number of sequences that are available. The properties change according to the number, so here in an epoxy, typical epoxy which we are very much interested because it is economic and most of the PCB industry uses this, we see bis phenol A is used in conjunction with epichlorohydrin and you get a polymeric structure like this, where you see the epoxy groups. So, you can see this as an epoxide; you can see the epoxy groups which are very functional, very reactive and this is what gives the properties of the dielectric functionality for a polymer.

(Refer Slide Time: 54:22)



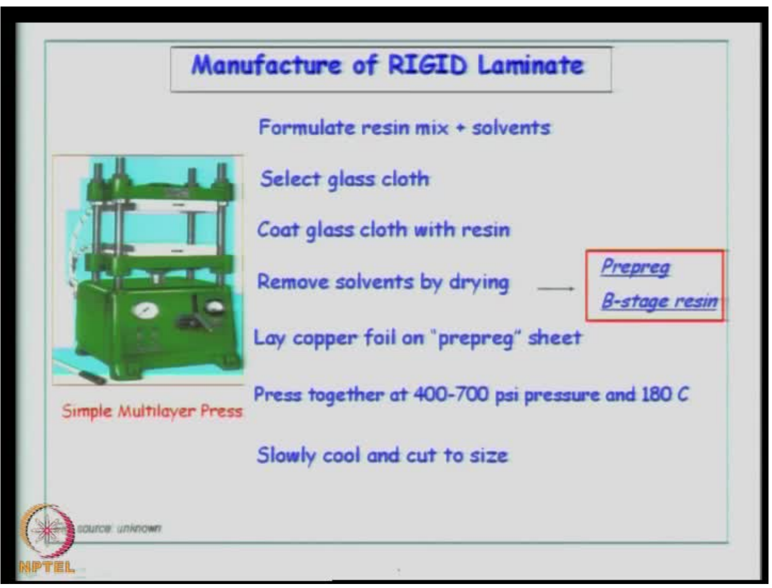
Here again, the value of n is important. You can play around with the Tg of this material by varying the number here the groups the structure here depicted in the square brackets the number of functional repetitive units, that you will see in the total chain. Now, polymers change dimensions on application of heat. Reinforcement is required to impart stability of the system; that is the reason why we use fillers.

So, when we modify the material with fillers you are modifying the CTE - Coefficient of Thermal Expansion which we have seen, when we discuss the packages how CTE of a silicon die; how CTE of an organic substrate have a huge mismatch. We try to build, for example in a flip chip system, we use a under fill to reduce the CTE mismatch. What are the reinforcements material that we can use kraft paper, cloth, random glass fibers, woven glass, aramid fiber - trade name is Kevlar, which has got very high Tg around 190 C. Then you can use graphite fiber; this is metal layer graphite fiber and cloth.

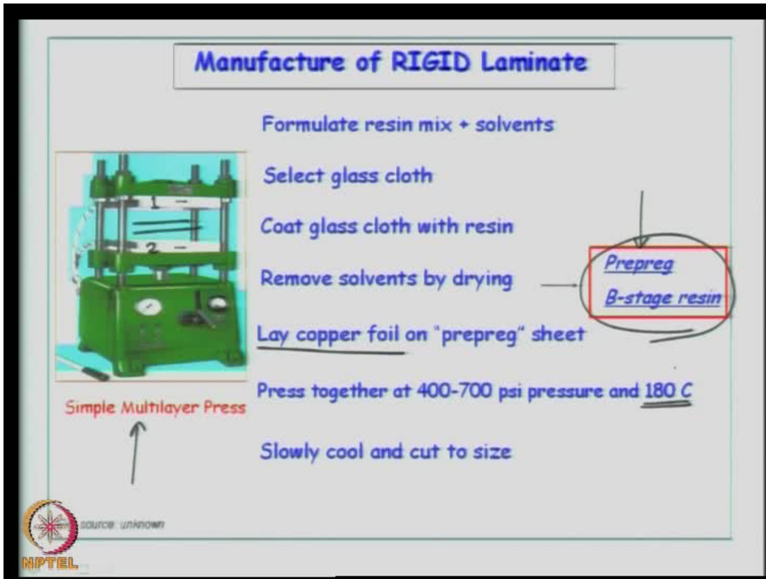
So, if you want to make metal core PCBs, you can use these kind of fillers graphite fiber, metal core structures like using molybdenum invar; invar is basically an iron nickel alloy

Fe-Ni, 36 percent of Ni is very common. Now, copper invar, copper heat sink core can be used. Ceramic spheres can be embedded into the dielectric material, resin material or you can use quartz fibers. What is the reason for using metal core PCBs? in cases of applications where too much heat is going to be dissipated, then you can choose a metal core. The other layers can be modified, you can use low dielectric constant materials or other organic materials but, if you are worried about thermal management in the PCB then you can modify structures like this and use metal cores.

(Refer Slide Time: 57:11)



(Refer Slide Time: 57:31)



Now, the filler in general will improve the mechanical strength, improve temperature resistance, improve dimensional stability, improve thermal conductivity and will

improve the drill and punch character, because this is a rigid mechanical process, that is one of the important steps in PCB manufacture. So, the manufacturer of a rigid laminate the structure that you are seeing here, how do finally manufacture? We have seen the ingredients (Refer Slide Time: 57:26). Now, what we are seeing is a rigid structure, how do you do it? We formulate the resin mix first and add the required solvents, select the glass cloth, coat the glass cloth with the resin and now you remove the solvents by drying. Typically, you remove about 40 to 50 percent of the solvents by drying and at that stage the material is known as a prepreg.

Remember, we are going to use this term very often in the following classes. The prepreg is a material known as a B stage resin, which is used in multilayer manufacture; a prepreg contains the resin; it contains the filler that is in this case, the glass cloth and it also contains some amount of solvent. So, prepreg needs to be stored at the workplace or in the lab at controlled conditions in a refrigerator because, if you keep it in normal room temperature your solvent is going to evaporate; it will become dry and further curing will not help in the bonding of the prepreg with the copper structure.

Now on this prepreg, you are going to use copper foils; that is, you will lay the copper foil on the prepreg sheet, top and bottom, let us say. Then, put it in a multilayer press, what you see here is a picture of a simple multilayer press (Refer Slide Time: 58:55). So, here in this area between the plates plate 1 and plate 2, you will keep your laminate structure; you can have multi structures and then apply pressure, apply load and keep it at constant temperature like 180 C and timings are also important.

Now, the epoxy material resin will ooze out, it will bond well with the surface of copper. This is a very slow process; you cannot suddenly heat it to 180 C and leave it for an hour and then expect it to bond well. There will be slow ramp up, reach a particular temperature it will stay there in the load applied and at the specified temperature for a certain period of time and there will be very slow cooling. So, now cooling process is equally important as the heating process. Finally, it is cooled down to room temperature and then cut to size, so this is how you manufacture a rigid laminate. So, what we have seen is the manufacture of a substrate like a rigid laminate that is used for PCB applications. We will continue with more information in the next hour.