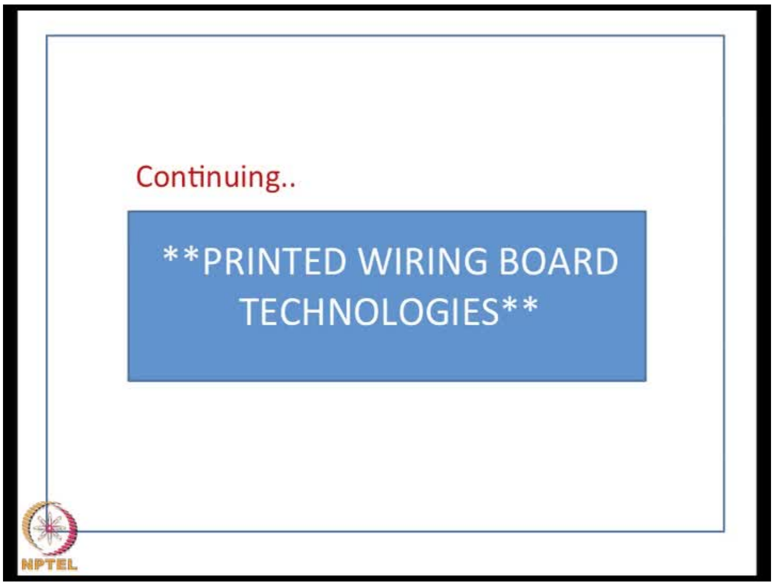
An Introduction to Electronics Systems Packaging

Prof. G. V. Mahesh
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 06
Lecture No. # 25
Substrates (continued)
Video highlights
Surface preparation

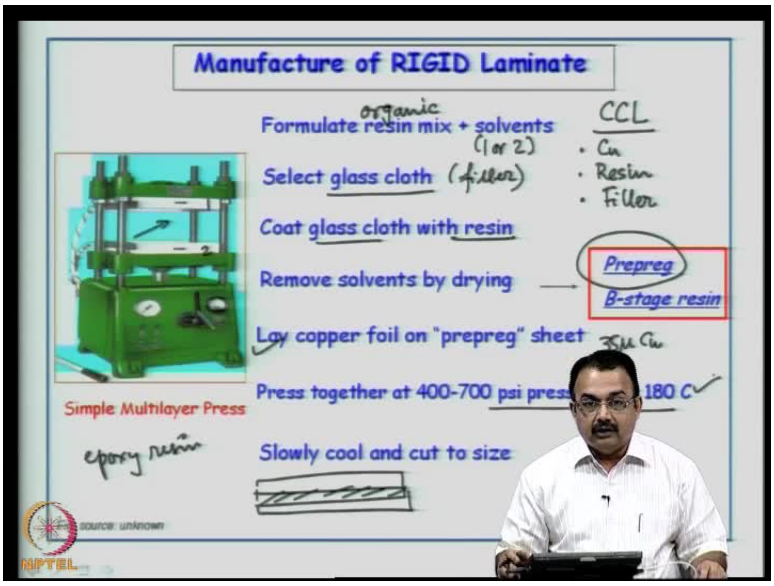
We will continue with this chapter on printed wiring board technologies. So far, we have seen the introduction to this chapter on the fabrication methodologies for printed wiring boards.

(Refer Slide Time: 00:16)



We have reached the stage where we are discussing the substrates - organic substrates. We have used the term called copper clad laminates to describe the base substrate that will be used for manufacturing or fabricating printed wiring boards.

(Refer Slide Time: 00:51)

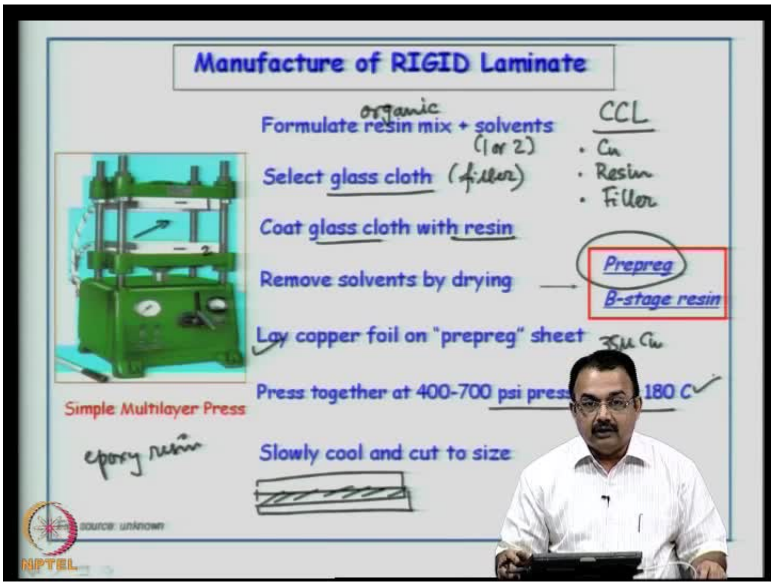


We have seen that basically when we talk about copper clad laminates there are essentially three components. If you recollect a copper clad laminate - CCL in short, it consist of three components one is the copper, second is the organic resin and third one is the filler component. We have used schematics in the last class to see what each of these denotes. Now, we are going to discuss in simple terms, the manufacture of a rigid laminate, we will discuss about flex substrates or flex copper clad laminates much later.

When you talk about manufacturing rigid laminates, essentially you require an organic resin; here the resin would be an organic resin. We have also seen different types of organic resins, let us say in this particular example we are going to use epoxy; we are going to use epoxy resin. When you talk about the laminate structure, you require a copper file to bond with the epoxy resin, but when you formulate the organic resin there will be solvents. You will have one solvent; but in some cases you may have two solvents, so you need to understand what kind of materials are involved. You can have a epoxy resin, you can have a polyimide resin, it could be a Teflon - poly tetra floro ethylene, and esters. We have discussed and listed various materials in the last class.

The next component that is required for the manufacturing will be the glass cloth or glass fiber, essentially this is the filler material that we are thinking about. Now, the glass cloth or the fiber is coated with the resin material, these two are mixed, how does it happen? You will have a container of resin into which the glass woven structure will be dipped into the tank and then comes out coated with the resin. These solvents will be air dried; sometimes, you can also use force drying by allowing it to go through a conveyorised oven at fixed temperatures that will remove the solvent. In some cases, partially the solvent is removed, because we are now going to prepare what is known as the prepreg material or the B stage resin.

(Refer Slide Time: 04:10)

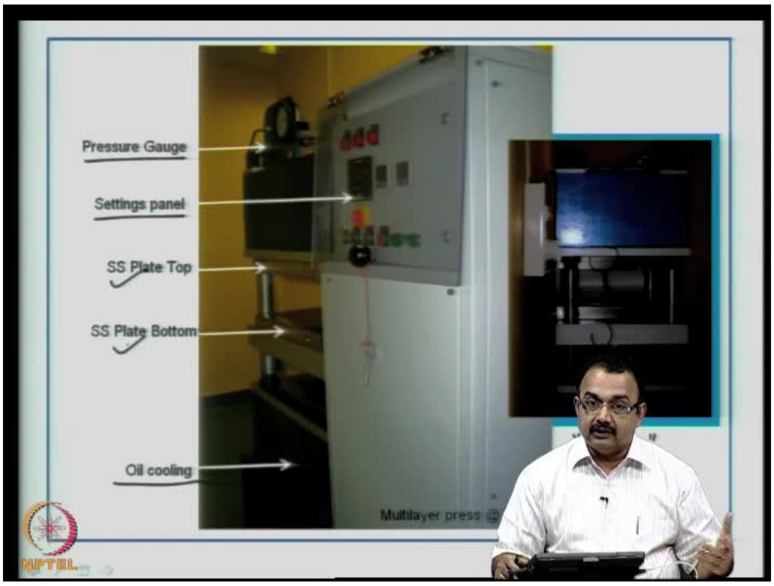


At this stage most of the solvents are removed by drying. At this stage prepreg is very important material that is used for multilayer boards in PCB manufacture. It will act as the inner layer dielectric and it can easily be sandwiched to two copper foils by lamination process. The next step will be to lay the copper foil onto the prepreg sheet, so the prepreg thickness may be 200 microns, 300 microns, 400 or 500 depending upon the requirement. The copper foil thickness again is dependent on the various requirements in the market.

Assume that you are going to use a 35 micron copper foil on both sides, by then suitably you can take copper foil and bond it with the prepreg sheets that has just been defined as a prepreg or a B stage resin. This structure of the dielectric material with the copper foil on top and at the bottom is now fed into the multilayer press. This picture shows a typical example of a small multilayer press, where you have two stainless steel plates (Refer Slide Time: 05:31).

Your buildup of the dielectric material and the two copper foils in this case is fed in between the two stainless steel plates, then they are pressed together at a suitable temperature and pressure; the timing is also very important. As I said previously, the heating has to be slow as well as the cooling has to be controlled, you cannot ramp up the temperature quickly to the required final temperature. Remove it out of the multilayer press and give it into the thermal shock, this will affect the physical properties of the copper clad laminate. Finally, it is cut to a size depending upon the market requirement.

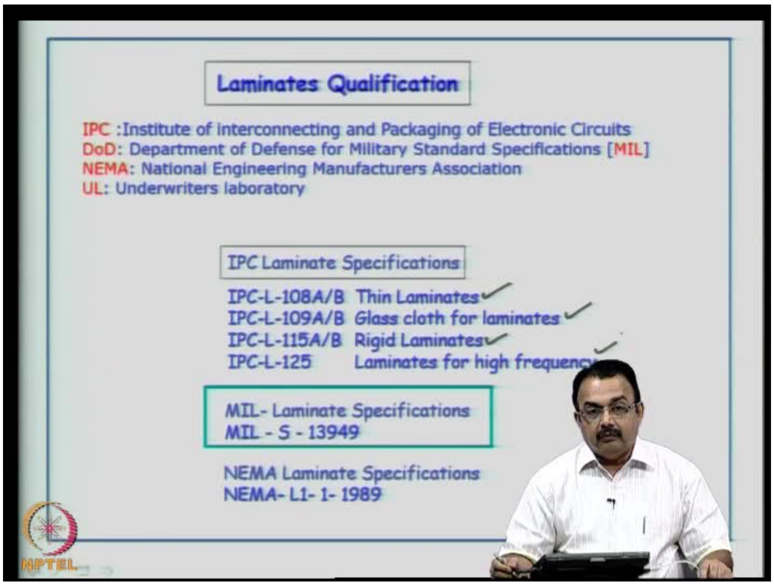
(Refer Slide Time: 06:29)



What you see here is basically a multilayer press, it can be custom made. Various models are available in the market depending upon your requirement and the volume of production. In a PCB industry, again we are going to use this multilayer press for laminating different layers that we have designed. If you look at this picture, you will see on the right side there are stainless steel plates here. In between these books, as we call it; each buildup structure is known as a book of laminate material, this will go inside the in between the plates, then they will be subjected to the appropriate pressure and temperatures (Refer Slide Time: 07:26).

So, essentially the components are; there will be a pressure gauge, you can see on the right side, there is a settings panel for the pressure and the temperature, you can program it for the required timing so that you can reach the maximum temperature in the required slow heating and slow cooling; there will be oil based cooling typically. Then you can see there is a stainless steel plate at the top and at the bottom, this is very typical of a simple multilayer press. For a large volume PCB manufacturing, the sizes can be large to accommodate the number of books that you require.

(Refer Slide Time: 08:09)

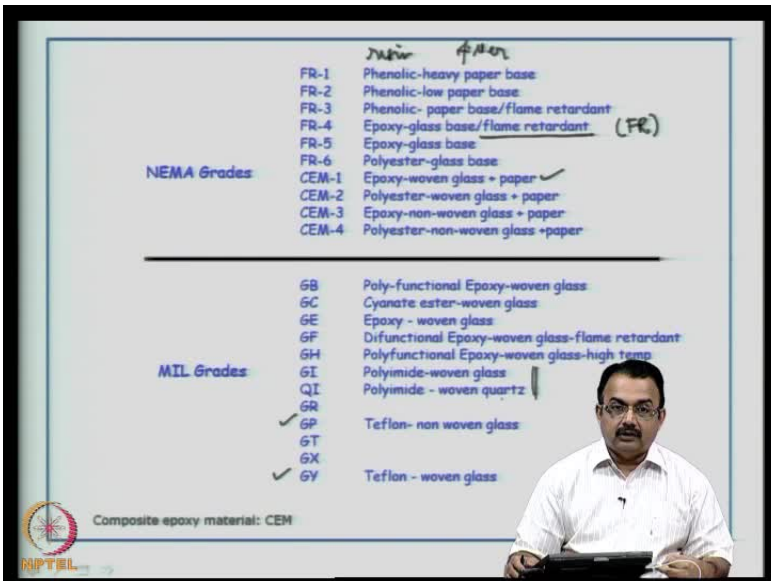


Now, let us look at the laminates qualification. IPC that is the Institute for interconnecting and packaging of electric circuits in USA, which is one of the leading institutions that define or give periodically the qualification and the test procedures for qualifying a copper clad laminate. This is very useful for the industry. Now, the department of defense military has also brought out MIL standards for laminates qualification. If a company A produces these laminates they have to follow some of these standards like the IPC, the MIL standard, the NEMA standard that is National engineering manufacturers association, or the underwriters' laboratory. Either of these or in some cases a combination of these qualification and test procedures have to be carried out and certified for it to be able to sell in the market.

The IPC laminates specifications will pertain to; there are various specifications some of the things are listed here. For example, 108A/B pertain to thin laminate structures, then 109A/B pertain to glass cloth laminates that means the physical properties - the thicknesses. The test to be carried out to qualify when such a material is used, 115A/ B is for rigid laminates, 125 laminates for high frequency applications. So epoxy for example, cannot be used for all applications, if you want to use high frequency application laminates, then you may have to look for better structures better material; for example, Teflon or polyimide, you will look for those kinds of materials.

MIL specifications, MIL-S-13949 is very common and the NEMA standard pertaining to laminate I S 1989, it is better to be aware of these qualifications. If you are doing research or building a product using laminates of various types, you have to be aware of some of the qualification procedures and the certifications, so you need to buy certified laminates.

(Refer Slide Time: 11:00)



NEMA grades are commonly known as FR-1, F R-2, FR-3, FR-4, FR-5, FR-6. Now, we will see what FR-1 is; FR-1 is a phenolic heavy paper base. Here, you can easily identify that the resin is phenolic based and the filler is paper based, so from these grades, FR-2 for example, is a phenolic low paper base. The phenolic is a resin, it indicates for what resin is used; organic resin. It contains paper filler, but the quality and the thickness of the paper might vary from FR-1. FR-4 for example, it contains an epoxy resin, it contains a filler which is glass base and more importantly it is flame retardant that is why we use the term FR, flame or fire retardant.

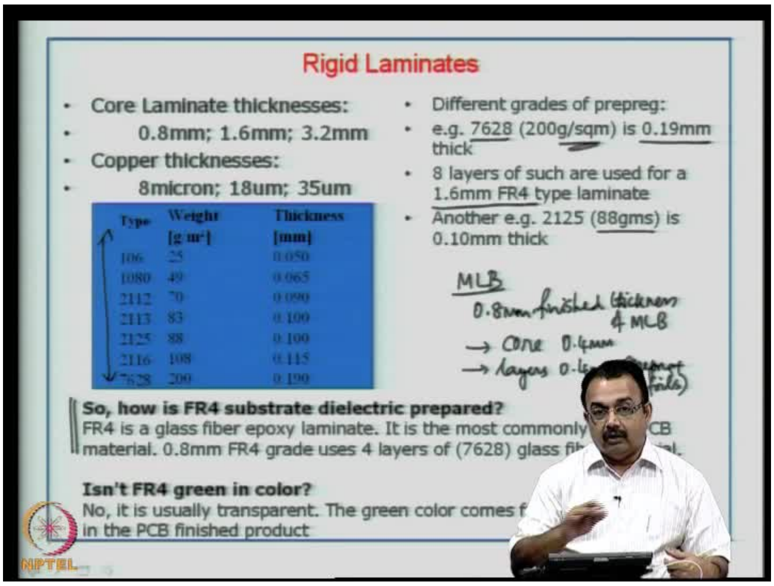
If the material has to be flame retardant or fire retardant, there are again qualification procedures for the material to be certified, as when it can be FR based. When an equipment using a PCB of FR-4 or FR-5 qualifications are used, and if there is some kind of overvoltage or short circuitry in the equipment, then the PCB, because of being organic can catch fire or because of the fumes that emanate when the temperatures exceed certain limits, it can catch fire, but it should have the ability to retard the flame by itself; that means, it should have a self-extinguishing property. The material combinations like epoxy or glass base with the fire retardant material in that can be used for such applications.

FR-5 is again epoxy and glass base, FR-6 is the polyester and glass base. You have what is known as CEM material, which is a composite epoxy material; that means, in all of these cases, the organic resin is epoxy or it could be a combination of other resins, but it contains different types of fillers. You can see for example, in CEM-1 you have epoxy but, at the same time you have woven glass and paper as the composite filler material.

CEM-2 is the polyester woven glass and paper, CEM-3 is the epoxy non-woven glass and paper, and so on. Then you have MIL grades, in certain cases people follow MIL grade or buy MIL grade copper clade laminates. You have GB, which is polyfunctional epoxy woven glass. Then in polyimide category, you have GI and QI; GI means polyimide with the woven glass structure and QI is the polyimide in woven quartz material. For Teflon, you have GP, GY and so on.

Now, the cost variations are always there. If you look at this big list of NEMA grades and MIL grades, all of them are not economical; some of them are expensive because it targets certain specifications and applications. For example, it could be high frequency application or it could be for space applications. So, very common consumer electronics typically use the most economical FR-4.

(Refer Slide Time: 15:06)



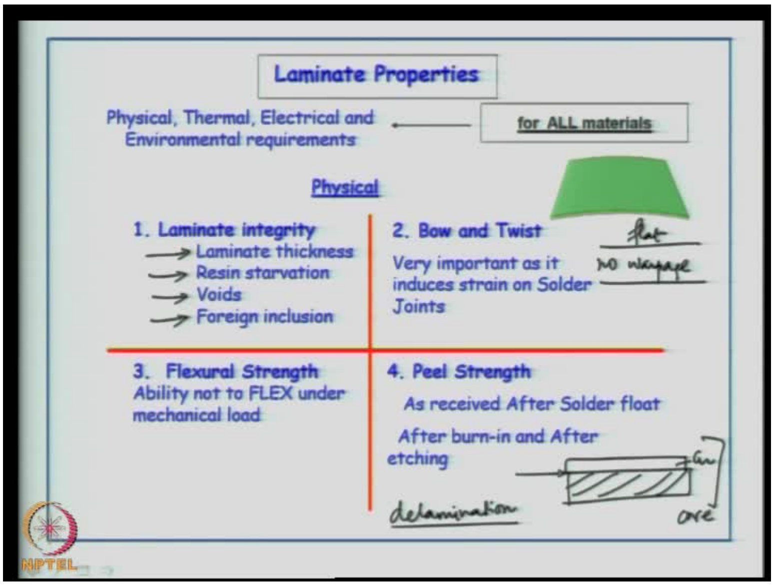
Now, let us come to some other important details for a rigid laminate structure. Typically, core laminate thickness is 0.8 mm, 1.6 mm or 3.2 mm; the copper thickness in the core can be 8 micron, 18 micron or 35 microns. In the case of prepregs there are again different grades; for example, a 7628 very common number in the PCB industry is basically 0.19 mm thick and the weight is 200 grams per square meter, this is evolved over the years. These kind of numbering systems; for example, if you want to manufacture or use a 1.6 mm FR-4 type laminates; FR-4 is epoxy plus 1.6 mm thick, then you can use eight layers of such, because 8 into 200 micron thick typically about 0.2 mm, this will give you 1.6 mm thick FR-4 laminate.

Another one is 2125, which is 100 micron thick, FR-4 which contains 88 grams of the material per square meter. These are the kind of notations that are available, in this box you can see various types of material listed, there are more but we have listed here very a few of them 7628, 2116 for example is 0.115 mm thick, 2125 is 0.1 mm, 106 is 0.05 mm.

Depending upon the buildup of your multilayer structure, you can pick the various prepreg grades and accordingly build your multilayer board. I will give you an example, if you have a multilayer board to be made, if you want to make a 0.8 mm finished thickness of your multilayer board, then you can start with a specific core which can be 0.4 mm. Then the layers can be built for the remaining 0.4 mm, by choosing various thicknesses of prepregs and copper foils. So, prepreg plus copper foil thickness knowledge is very important for a manufacturer.

Now, the question is; how is FR-4 substrate director prepared? We have seen FR-4, I want to reemphasize that it is a glass fiber epoxy laminate, most commonly used PCB material. 0.8mm FR-4 grade uses four layers of glass fiber material. Is FR-4 green in color, because I have shown you various samples over the last few lectures; is it green? No, the basic FR-4 material is not green in color; the green color that you have seen in most of the samples is basically coming from the solder mask color, which is green. Solder mask is one of the final applications or final processes on the finished PCB, so this FR-4 material is usually transparent in color.

(Refer Slide Time: 18:55)



Now, we look at some of the laminate properties; we look at physical, thermal, electrical and environmental requirements. The physical properties and the laminate integrity are the properties that a manufacturer will check and pass it onto the consumer – end-n consumer. Laminate integrity; what does it mean? We are worried about the laminate thickness; if you say it is 1.6 mm there will be a tolerance, but the dimensions have to be maintained.

Resin starvation; we talked about resin plus the filler mixing together in the initial stages of the b stage resin manufacture, at that stage the resin should be completely wet. The filler material for example, the glass cloth in FR-4 should be completely wet with the epoxy resin. At that stage there should be no voids or starvation of the resin material, it should fill all the openings in the glass cloth.

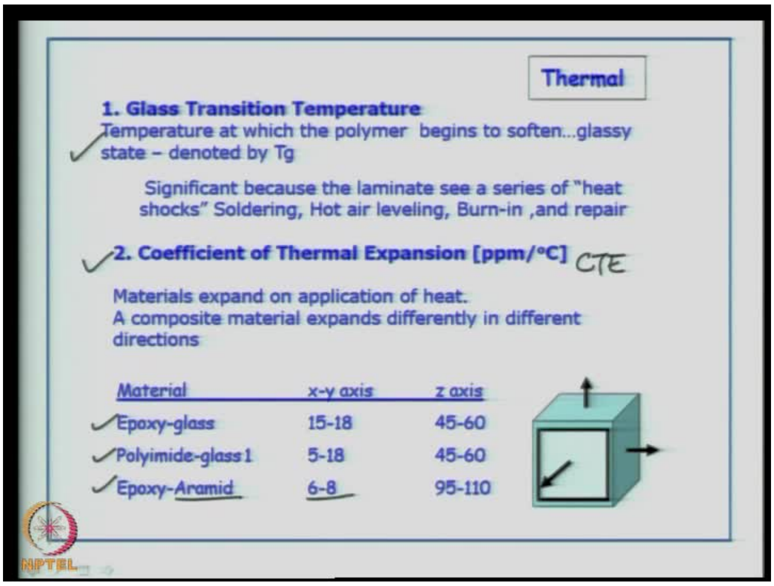
There should be no foreign particle inclusion; dust, moisture, other foreign materials like hair, or other impurities, because once it is embedded in the structure, it is going to be very difficult to remove that and it is going to be carried away to the final PCB. The second point is bow and twist, it is very important as it induces strain on the solder joint.

Now, the laminate should be very flat and there should be no warpage. The storage also became very important; how you can store your laminate? It should be stored flat; that is horizontally, it should not be kept standing. Now, the bow and the twist property is important because, it also indicates how the manufacturing has taken place in terms of the correct thickness of the dielectric and the copper on both sides if it is double sided board. If there is a bend, if you have started fabricating the PCB, when the board experiences thermal shock during soldering and so on, then there will be dimensional changes, there will be heat shock, thermal shock. This will induce the strain at the solder joint because of this warpage. So, make sure that your PCB is flat.

Flexural strength is another important property; it is the ability not to flex under mechanical load, because we are talking about rigid laminates. Peel strength; now the copper, as I have drawn many times the structure, there is a dielectric and there is a copper here. Now, the peel strength at this point is very important, we talk about delaminating.

The material should not delaminate from the base substrate; the base substrate is the dielectric. This is a core, the core should not exhibit any delamination and how do you test that? In the manufacturing of the copper clad laminate basically they take samples of this laminate and then allow it to float on molten solder. It gives a thermal shock and that is allowed to cool, then they look at the peel strength on those areas of copper. They will try to see if the copper is peeling away from the resin material; resin plus filler material. This will give an indication of the peel strength of the electro deposited or electro formed copper. They can also do burning test. During etching, you can see if there is any delamination, etching should take place only on the exposed areas of copper material to the etch end.

(Refer Slide Time: 23:18)

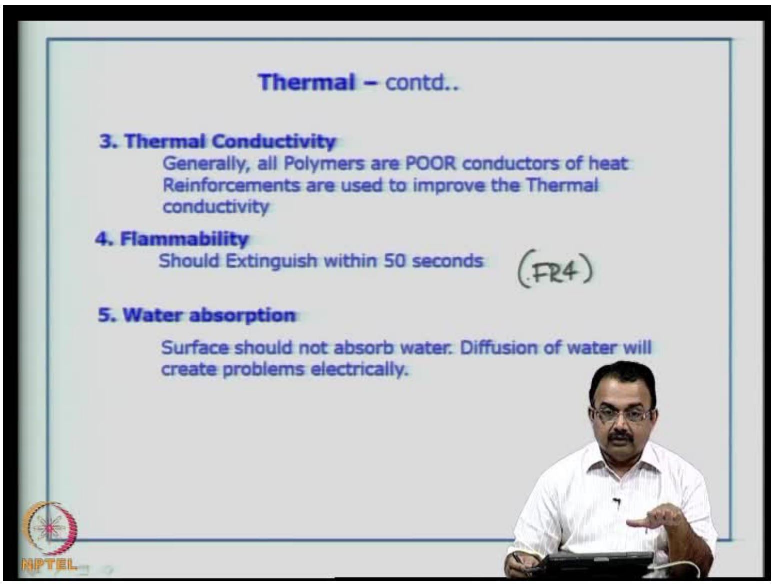


The thermal properties that we are interested are the glass transition temperature denoted by Tg, we have discussed this briefly in the last class. It is the temperature at which the polymer begins to soften, there is a glassy state denoted by T g. It is significant because the laminate will see a series of heat shocks during the process of PCB like soldering, hot air leveling of solder, burn in test for reliability and repair - repair can be manual soldering; manual soldering can in fact give more thermal shock compare to machine soldering.

Then we are interested in CTE - coefficient of thermal expansion. This property indicates the integrity of the material; materials expand on application of heat. A composite material expand differently in different directions, so you have CTE in the x, y and z axis, how is this important? Your PCB for example, if it contains the epoxy glass as the core substrate, then the CTE in the x-y axis is around 15 to 18 and the z axis is around 45 to 60.

We are also talking about this material being used in conjunction with silicon, so there can be huge CTE mismatch, because of various thermal cycles, board should have good dimensional rigidity and reliability. It should be stable - dimensionally stable, so if the CTE - coefficient thermal expansion is going to be very large, then it will affect your integrity of the board especially in terms of registration, during assembly and so on. You can see various materials like epoxy glass; polyimide is only 5 to 18 epoxy aramid, which is a different type of a glass material. Cloth material has got 6 to 8, it is better for a designer to know these characteristics.

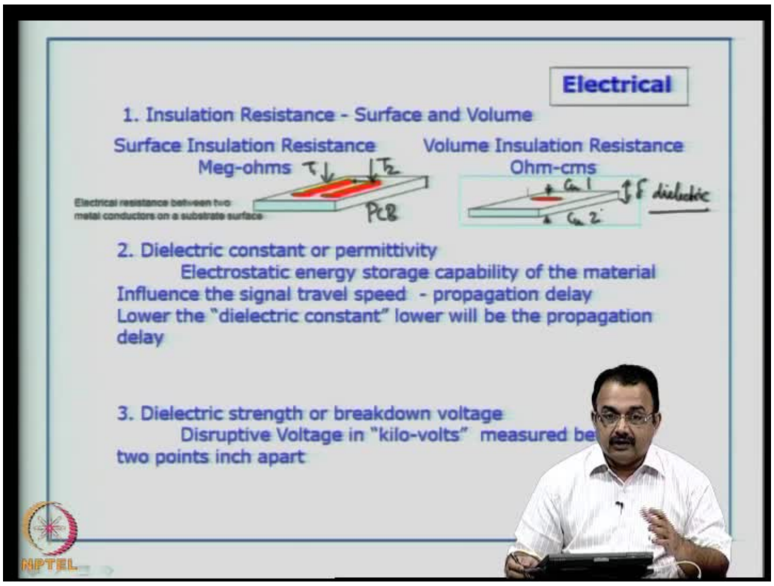
(Refer Slide Time: 25:35)



Thermal conductivity: Generally, all polymers are poor conductors of heat reinforcements that are used to improve the thermal conductivity. We have seen how the metal core is also used in the core structure of a PCB to remove heat, to act as a heat sink, because some packages like your processors can dissipate lots of heat. It is better to remove the heat from the package to the body of the PCB using core metal structures, or other fillers can also be used. Flammability should extinguish within 50 seconds, I was talking about this particular point when we talked about the classification of FR-4 and so on.

Certain tests are done to find out that if it follows this flammability or the self-extinguish property for glass epoxy, other materials are also available. The other important property in a laminate is it should not absorb water, moisture absorption should be less and diffusion of water into the structure could create problems electrically. That is why we say the resin starvation should not be there, the copper to dielectric bonding should take care of these kinds of problems.

(Refer Slide Time: 27:05)



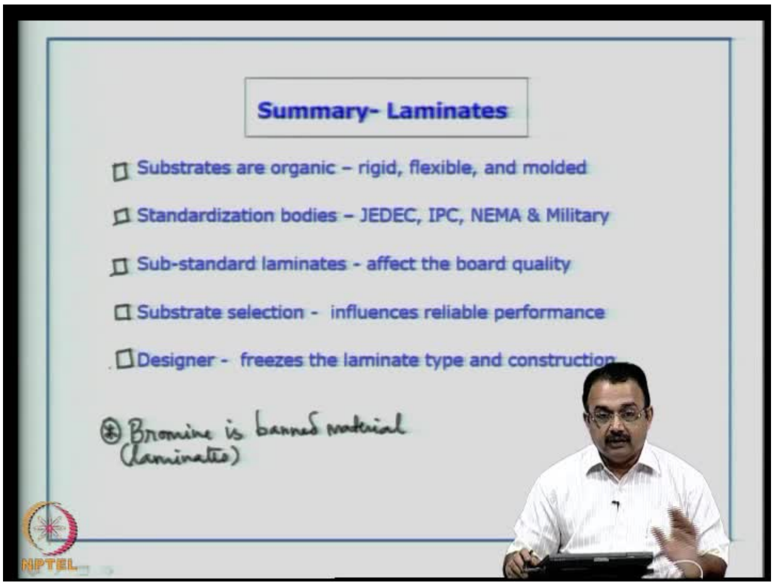
Electrical properties; we are interested in understanding insulation resistance, both surface and volume resistance. Surface insulation resistance is basically the electrical resistance between two metal conductors on a substrate surface. If you look at PCB like this and if you have two parallel tracks; tack one, track two let us say. We are interested in the electrical resistance that this material - dielectric material can provide.

In addition, as you know we are coating at solder mask material as a dielectric, these are all very important manufacturing steps that will take care of the integrity of your board in terms of electrical parasitic. In volume insulation resistance here, we are worried about the thickness of the dielectric material. Lower the thickness; we are interested in the dielectric constant being very low. How copper on plane 1 and copper on plane 2 can function electrically in a much better fashion based on the thickness of the dielectric. The volume insulation resistance is measured in ohm centimeters, this is also a parameter of importance for the designer.

Dielectric constant or permittivity; all of you were aware of this basic concept of dielectric constant in a material. It is basically the electro static energy's storage capacity of the material that influences the signal delay in a system or the signal travel speed or you can call it as a propagation delay - signal propagation delay. If you talk about delay in structures, structure like copper, copper and dielectric, you can measure the dielectric, you can measure the signal propagation delay and this in turn is related to the permittivity of the material.

Lower the dielectric constant lower will be the propagation delay, so that is very clear. The other point is dielectric strength or breakdown voltage property; this structure is sandwich structure, it can be subjected to a destructive voltage in kilo volts measured between two points, an inch apart. There are certain threshold values which we can utilize and check if it can qualify the current industrial standards.

(Refer Slide Time: 29:49)



These are the important things that we look for in a laminates. In addition, today I talked about the environmental aspects of laminates, most laminates contain bromine as a fire retardant or a flame retardant material, but today bromine is banned. Bromine is banned material in the PCB or in the electronics industries; laminates which contain bromine are slowly being faced out. Environmentally if you talk about requirement for a laminates, we are moving into era where bromine is replaced by materials like phosphorus and so on. We are going to discuss about this when you come to lead free chapter, but it is better to understand the implications of the new legislations that come into the electronics industry, it has affected the laminates also.

As a summary, we talk about substrates that are organic, rigid, and flexible, they can be molded. The standards or the standardization bodies that are JEDEC for components; IPC for components, assembly and manufacturing; NEMA for manufacturing; military also covers components, manufacturing and solders. Do not use substandard laminates, electrically they are going to affect the board quality, thermo mechanically also they will affect the board reliability.

Substrate selection; a designer spend a lot of time in understanding what a substrate is, because it influences the performance of the board, especially in high frequency applications, high performance applications and strategic applications. So designer, once again if I can remind you, freezes the laminate type, it is not the manufacturer who is going to take the FR-4 or a polyimide or a teflon, it is you; the designer, who has to decide what laminate you want for your electrical applications or electronics application.

(Refer Slide Time: 32:14)

Now, we will come to the board fabrication process step by step, before that you should know the safety procedures before entering the lab or your work spot. Safety procedures are almost the same, but depending upon the concentration of certain processes in the lab the safety procedures need to be less or more. Understand the safety procedures before entering the lab, you need to know about the material safety; you have the rights to know about material safety, so any work place have to give you material safety instructions and you can demand for it.

Read the materials safety data sheet that is available for every material. If you buy a chemical, along with it comes a material safety data sheet, you have to read them carefully to understand what are the implications of that chemical in terms of usage, health, hazard. In terms of the nature of the chemicals, whether it is a poisonous chemical or it is a carcinogenic material, which needs to be carefully looked into in terms of handling and so on.

Follow clean room work procedures; clean room work procedures will again vary depending upon the classification of the clean room. We talked about clean room classifications when we talked about semiconductor manufacturing, it could be a class ten clean room, class hundred, class thousand, ten thousand and so on. Depending upon the clean room levels that you are going to work, you need to use proper gowning methodology; proper clean room equipment needs to be in place.

Make sure firefighting equipment is in place; make sure of the nearest health centre, security, telephone numbers are made available to all personal working in the lab or in the work spot. This is a very general guideline I would like to give for all students, personal working in RND and other institutions. Now, we will have a look at typically what it means to have a clean room work procedure or gowning procedure. Have a look at this video, I will also try to explain you some of the highlights depicted in this video.

So essentially, your clean room can be as I said class ten, hundred, thousand, ten thousand and so on (Refer Slide Time: 35:08). This is the ESP lab in CEDT, what basically we are trying to show here is the gowning that you need to undergo. You need an apron; make sure that the gloves are used when you are handling the boards and the chemicals. There are different materials that are being used in the PCB manufacture, for example in our lab there are different sections of clean rooms in the lab. Nevertheless gloves have to be worm; you have to wear a facemask that can protect you from fumes, acid fumes, alkaline fumes or other organic solvents and so on.

You need to wear a head mask that will prevent hair falling into the lab, as you know hair thickness is very small, but hair falling on a photoresist will be translated onto the PCB. Use goggles, very important for safety. Normally in a lab or in a manufacturing setup there will be an air curtain and also an air shower, any person entering into the lab need to have air shower.

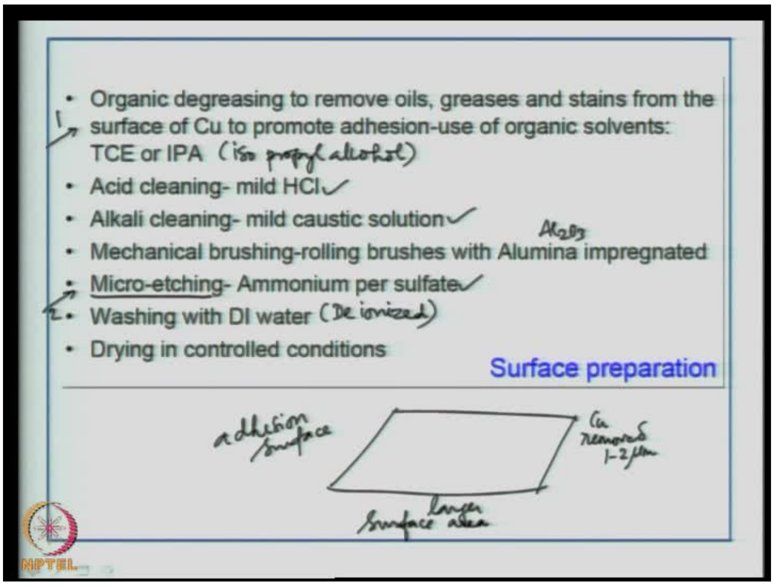
The air shower is used because it will blow away all the dust from you, from your hair, from your clothing and so on. This will prevent all the dust entering into the lab, because as I said a dust or a pick of dust falling on a material in the surface of the PCB or on the surface of the PCB is going to get translated along with the design.

This is yellow room that you can see sometimes materials are very sensitive, temperature and humidity conditions need to be maintained (Refer Slide Time: 37:13). For maintaining the clean room, here you can see Hepa filters are used, basically these are filters that are used to filter the dust; this is basically used to filter the dust and then these are removed by an exhaust.

So you have a laminar flow structure, this Hepa filters can basically remove all the dust from your body and from the clothing. This video has demonstrated typically how you have to enter the lab, because some of the yellow room activities; we say yellow room because some of the materials cannot be exposed to white light.

We talk about board preparation or surface preparation; it is the first activity in the board fabrication sequence. In the inset, you are seeing a printed circuit board, the color of the laminate as you see here is almost transparent, the green color that we have normally used to say as a finished PCB, it is basically the color of the solder mask material. We will now see how a copper clad laminate which is the starting material needs to be readied for transfer of image from your mask that you have manufactured, pertaining to your design, on to the surface of copper.

(Refer Slide Time: 38:54)



If you look at the copper clad laminate on the surface, you will see lot of organic grease. If you touch the copper clad laminate, your thumb impression or finger impression is going to be attached because your hand is greasy or because of the natural sweat or along with it there is dust and it is going to be transferred on to your copper clad laminate. Because of large storage there can be accumulated particles and other dust materials, there can be stains, because as you know copper is very reactive or moderately reactive to atmosphere, copper can react with the oxygen in air to form copper oxide.

So, oil debris from the equipment, greases from your hand or other chemicals, oxides, some other stains; chemicals falling on the surface of the copper if unattended to. Generally, laminates need to be protected until it is used, but if not, nevertheless you have to treat the surface of copper. Now, the treatment is required because we are going to promote the addition on the surface of copper, how do we do that? As you know, we are now going to transfer an image on to the copper surface using a photoresist material, so this photoresist has to adhere nicely on to the copper surface. First, we use organic solvents like trichloroethylene or isopropyl alcohol. Trichloroethylene is almost banned nowadays from being used, so I would rather suggest use isopropyl alcohol.

IPA stands for iso propyl alcohol; it is a very safe chemical. You can even dilute it with water, it is mixable with water and therefore you can use this I P A for cleaning the surface of your copper. If you use organic solvents all the grease is removed, you can use hand cleaning or you can use machine based cleaning. You can also do acid cleaning using mild HCL - hydrochloric acid, alkali cleaning using mild caustic solution; caustic solution is basically hydroxide based solution. You have mechanical brushing using rolling brushes with alumina - alumnium oxide, Al_2O_3 - alumnium oxide impregnated in to the fiber structure of brushes. Now this will act as a very good mechanical aid to remove oxides from the surface, dust material from the surface.

Another option is to use micro etchant, micro etchant means removing very small thicknesses of copper from the surface. Typically we use ammonium per sulfate for washing, then with deionized water, DI water is known as de ionized water, there are no ions except hydrogen and oxygen, in this water no salt is present. Tap water contains a lot of solvents so you have to treat it to get DI water and finally, dried in controlled conditions. These are the number of steps that you can probably do for keeping a copper clad laminate surface clean.

You can use both chemical cleaning as well as mechanical cleaning. The need for making the copper clad surface rough is; when you get the copper clad laminate from the manufacturer, generally the surface is very smooth. If you use it as it is your adhesive in the photoresist is not going to stick on to the surface. For yielding a very good adhesion onto the surface you have to remove a few microns of copper, so copper is removed. Let us say it is about one to two microns from the surface, it also provides a larger surface area compared to the structure that you get from the manufacturer.

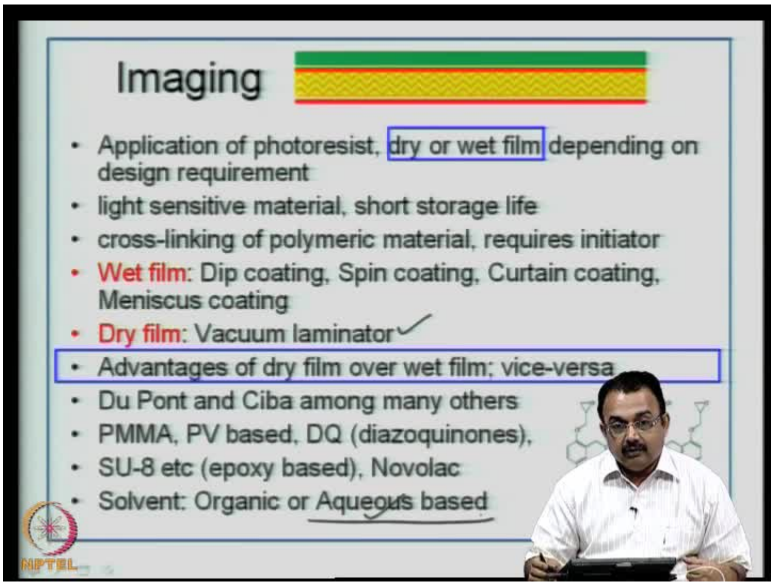
So all of these or a few of these or a couple of these, let us say you can use micro etching first; or first you can do an organic cleaning, then you can do a micro etching and then you can do a mechanical brushing. Finally, a quick acid rinse is neutralizing with an alkali rinse and then washed up with DI water and dried in controlled conditions. During drying you have to make sure that the water is totally removed, because drops of water can affect the next manufacturing step. Surface preparation is an important activity so we will now see a small video on the procedures which I have just described.

Here, you can see the copper clad laminate, there are grilled holes (Refer Slide Time: 44:41). This is now being acid rinsed, mechanically using a fiber with impregnated alumina you are scrubbing. Alternatively, you can use machine brushing where you have rollers of the same type of fibers, the laminate is fed in between the rollers and you have water being sprayed from the top and the bottom.

Sometimes, you can use jet slurry containing pumous, you can use water mixed with the pumous material and then you can form slurry. This slurry can be sprayed from the top and bottom of the brushing machine and then you have the abrasive rollers. Together it forms a very nice methodology for scrubbing and removing your copper clad laminate, providing a larger surface area, removing the oxides and so on. Now, during this process the organic greases can be removed and various other contaminants can also be removed.

Once again have a look at this video, because even for double sided board manufacture, the holes are first drilled and the holes will create bur. The bur is an important defect that you cannot ignore so these kinds of treatment and methodologies like the scrubbing can remove bur from a drilled hole. The reliability of the board gets built right from this process step, so board surface preparation can never be ignored. Finally, the board is kept in a woven; let us say at 8 degree centigrade, for about 10 to 20 minutes to make sure that all the moisture is removed. You can also increase the temperature to 90 or 100, but it is sophist to have at around 80 to 90 to remove all the moisture in about ten minutes.

(Refer Slide Time: 47:07)



These are all automated in an industry. What you are seeing is a typical prototyping activity. The next step after the board has been prepared is the imaging process, what do we do here? We apply the photoresist; it can be dry or wet film depending on the design requirement. Photoresist is a light sensitive material, it has got short storage life typically about six months only, so if you buy a photoresist material you have to consume this material in the process within six months and you need to store it carefully in controlled light conditions like a yellow room. Most of the photoresist materials are light sensitive and you cannot work in white light.

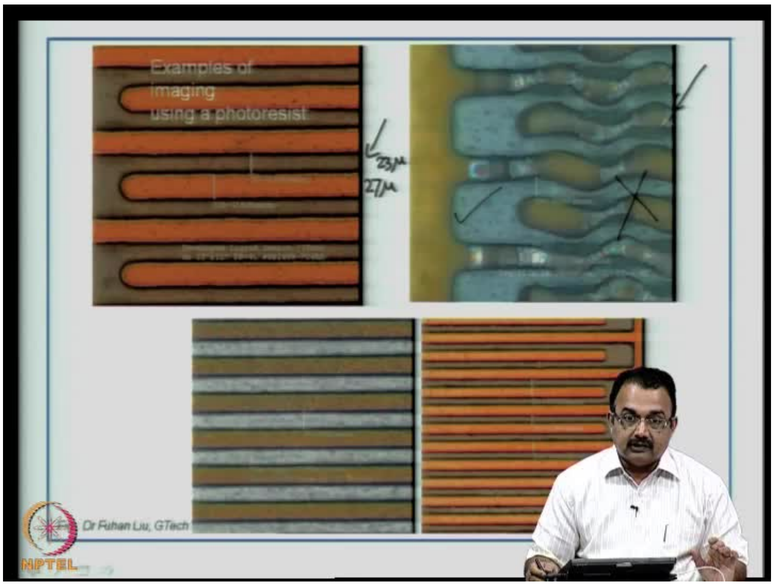
Typically, they are polymers, they are manufactured by cross linking of polymeric material and they require an initiator. In this case, U V light is the initiator that we are going to use in this photolithography process. So, imaging here is basically a photolithography process used in semiconductor industry, the same thing we are using in the PCB industry, the only difference is, the materials are different in both the cases. We are talking about different feature sizes in semiconductor and larger feature sizes in the board level activity.

The wet film can be applied by dip coating; a process known as dip coating, spin coating curtain coating or meniscus coating, for large volume some of these methods like spin coating or dip coating are not profitable or viable. For large volume, you have to go for methods like a curtain coating or use a dry film methodology which uses a vacuum laminator technology to coat the photoresist material onto the surface of the board. Now, all of these whether it is a wet film or a dry film contains an adhesive material, this requires adhering nicely to the surface of the board that is what the importance of surface preparation.

There are advantages of using a dry film or a wet film, which I will cover at the end of this section on imaging, because firstly you need to know what is a dry film, what is it look like, what is a wet film, what are the thicknesses you can get with the wet film or a dry film. When do you use a dry film? When do you use a wet film? There are many companies today; some of the leading names are Du Pont, Ciba among many others, which manufacture photoresist material in very large quantities to the semiconductor industry as well as to the PCB industry.

Some of the common organic names are poly methyl methacrylate, poly vinyl based diazoquinones and so on. SU-8 is a very recent innovative material. New material which is epoxy based use for high density interconnects. In sequential buildup structures you can use very thin material for creating vias on structures, Novolac resins are also used. Solvent; in earlier days people were using organic solvents, but today most of the process is aqueous based, because there is no health hazard you can prepare the developing processes. One of the important processes in imaging is developing; so you can prepare an aqueous developer right in your lab.

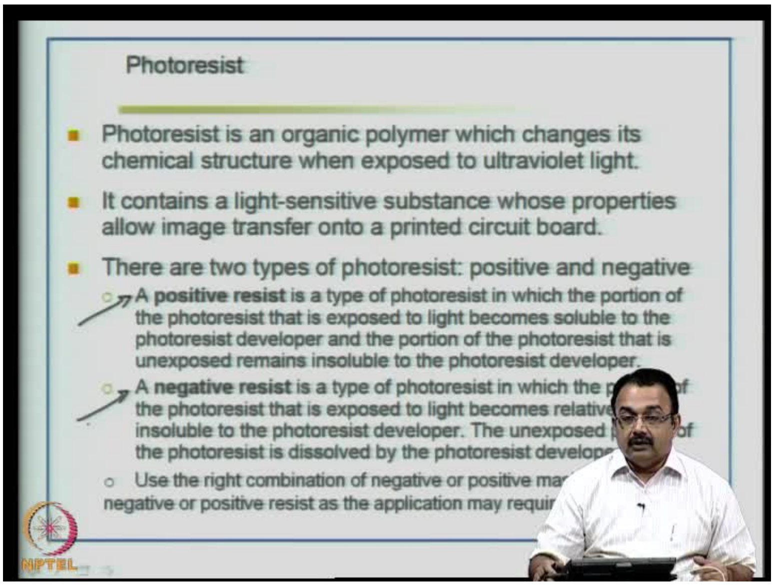
(Refer Slide Time: 51:01)



Have a look at these pictures; you can see these are some images of tracks that have been generated using a photoresist material. On the left you see, here in this particular picture a very nice image of a track which is 27 microns and then the gap between 2 tracks is about 23 microns. This is very fine line imaging, this is a zoomed up image of a photoresist imaging process completed with a series of tracks. Copper tracks need to be defined on the board; you can see that the gap is 23 micron and the track is around 28 micron. This kind of resolution is required or expected from the photoresist material, but all materials will afford this.

So you have to be very careful in choosing the thickness of the photoresist to achieve such verifying lines, we are talking about 1 milli line or 2 milli line and the spacing. If the surface of the board has not been prepared properly, this is the result (Refer Slide Time: 52:24). You can see that after the developing process of the wet or the dry film photoresist. You can see parts of it are held firmly on the surface, part of it is not held on the surface. This is a very good example to show the defects during photoresist imaging process. Here again, below we are seeing very fine line structures being done. For example, this is a 40 micron track and the gap is around 38 micron, so sometimes very fine lines are required in the PCB industry.

(Refer Slide Time: 53:04)



What is a photoresist? A photoresist is an organic polymer which changes its chemical structure when exposed to UV light. Our first job is to coat a photoresist and expose it to

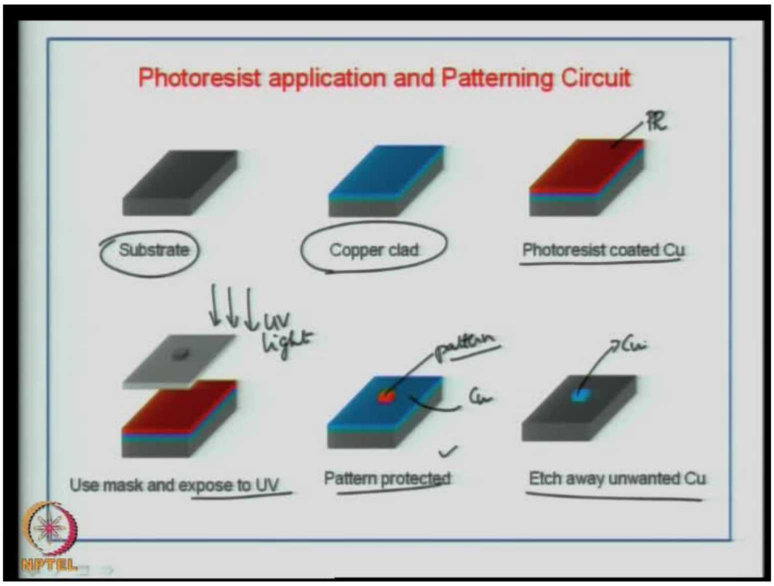
UV light using mask. Remember that the mask has been prepared for various layers, so we use a mask and expose it to UV light. Certain areas will be expose to UV light, certain areas will be not be depending on the mask, so it contains a light sensitive substance whose properties allow image transfer on to a PCB.

There are two types of photoresist; positive and negative. A positive resist is the one in which the portion of the photoresist is exposed to light through the mask and it becomes soluble in the developer. Remember that after a photoresist application the next step is to make the image visible and also stable on the surface, so we use a developer.

In a positive photoresist, once the exposure of UV light is complete the exposed areas become soluble, the portion of the photoresist that is unexposed remains in soluble to the developer and they will remain on the surface of the board, so understand what a positive resist is. In the case of negative photoresist, it is resist in which after applying, using a mask apply U V light, the areas that are exposed to U V light become insoluble to the developer, so there is a different set of chemical reaction that take place compare to the first one; the positive photoresist.

The unexposed portion in this case will be dissolved in the developer that you are using in the next process. Using the right combination of a negative or a positive mask and a negative or a positive resist you can play around with the kind of image that you require. In some cases your positive resist may be very good for fine line circuitry, but for coarse lines you can use a negative photoresist. Depending upon the sequence in your multilayer board fabrication this technology can be used for any other printing process, not necessarily PCB.

(Refer Slide Time: 56:11)



Depending upon various structures, these are very important if you are going to build microvia structures and fine lines on boards. A negative photoresist and positive photoresist have different chemical properties, when they are exposed to the U V light, it results in a different solubility in a developer, so use this very judiciously. The process sequence will look like this; photoresist application and patterning of the circuit. The first thing is you have the substrate, it is taken, then assume that it is copper clad, you coat that with the photoresist material on the copper surface, this is the photoresist, PR stands for photoresist. We use a mask, this is a mask and you can see this is exposed to U V light.

Depending on the photoresist whether it is a positive or a negative; let us assume in this case it is a negative resist. You can see the pattern; the red one, this is the pattern that is protected. Here, it depicts a pad but, you can have multiple pads, multiple tracks depicting a circuitry, this is just an example to show an area for circuit or a pattern that is protected, the other areas can be removed; the other areas can be removed after developing process. This original copper is etched away retaining only the small area of copper that you require as a pattern, as a circuit. This is a very simple schematic that you can look at carefully and understand how a photoresist works. We will now continue with the imaging of the photoresist; we will talk about different photoresist materials; we will also have video highlights on photoresist imaging photolithography developing and so on, which will be in the next class.