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. # 29
Video highlights on manufacturing
Solder mask for PWBs
Multilayer PWBs
Introduction to microvias

We will continue with the printed wiring board technologies. This is the seventh hour in this particular module where we are discussing the basic printed wiring board technologies.

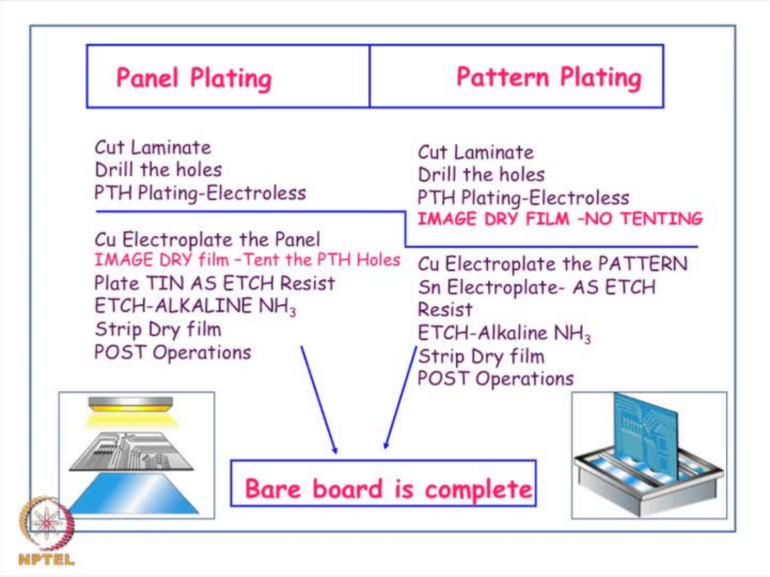
(Refer Slide Time: 00:16)

Continuing..

PRINTED WIRING BOARD TECHNOLOGIES

Remember, I have used the term system; so this is a module that we have exhaustively looked at, in terms of how a printed wiring board can be used as a system level entity.

(Refer Slide Time: 00:50)



Panel Plating | **Pattern Plating**

Cut Laminate
Drill the holes
PTH Plating-Electroless

Cu Electroplate the Panel
IMAGE DRY film -Tent the PTH Holes
Plate TIN AS ETCH Resist
ETCH-ALKALINE NH_3
Strip Dry film
POST Operations

Cut Laminate
Drill the holes
PTH Plating-Electroless
IMAGE DRY FILM -NO TENTING

Cu Electroplate the PATTERN
Sn Electroplate- AS ETCH Resist
ETCH-Alkaline NH_3
Strip Dry film
POST Operations

Bare board is complete

Now, we have also gone in to the various sequence steps for plated through hole process, as part of the double sided and multilayer buildup. So, we continue with the plated through hole sequence.

What do you mean by a plated through hole? There is a dielectric material and then you have the copper at the top and at the bottom; now we are drilling a mechanical hole. These holes are meant for providing interconnects - electrical contact between the top copper and the bottom copper - when you drill a hole the inside of the hole-wall is a dielectric material, it is nonconductive. So, you have to make it conductive. The plated through hole process sequence will let you know, how a conductive hole wall is established from using a combination of electroless copper and electro plating copper.

So, we look at the sequence once again and this is highlighted through videos. Now, if you look at the first step (Refer Slide Time: 02:09), as we have discussed last time it is the image transfer process. So I let you watch the image transfer, because this is going to be a very important step, whether we are going to do pattern plating or a panel plating for a double sided printed wiring board.

(Refer Slide Time: 02:24)

You can see here, the board has been laminated with the dry film photo resist material. Now, remember all the properties that are required to be established before a dry film operation. The suitable masks for imaging the electrical layers are subsequently taken. They are aligned with the through holes that you have drilled and then they are introduced into the UV exposure unit. You can see this is a UV exposure unit, where sufficient UV light is exposed to the dry film through the mask. Now, this will help establish an image (Refer Slide Time: 03:05).

This image is further stabilized by the developing process, so the resist will be removed in the unwanted areas. The resist will be remained on the substrate where the circuit is to be protected or in some cases, where the circuit pattern requires additional copper to be added in terms of increasing the thickness of copper on the circuit areas. This is the first step - that is the image transfer.

Now, we get into the second step of the plated through hole sequence which is called the electroless copper plating. So once again, I emphasis we are not using an external DC current here, we are using a chemical reduction process from a copper rich solution and using some reducing agents to deposit copper on to the surface of the board, as well as the through holes.

Now before that, if you look at the video what you are basically doing is microetch. This will remove some amount of copper from the surface of the board to the extent of about 1 to 2 microns. This will provide more roughness, more surface area for imaging process to happen in a good manner. Then the next step after the microetch is a water rinse; typically you can use a cascade rinse or a spray rinse in some cases, as you have seen here for 30 seconds. Then it under goes an acid treatment in a 10 percent sulphuric acid dip to activate the surface with the acid, then it is neutralized with water. So, these are all the cleaning processes; what are you seeing is basically - a Microetch is a cleaning process, the Sulphuric acid is a cleaning process, to remove oxides and other debris that may be on the surface.

We are introducing in to a pre dip; remember the next step is going to be a catalyst activator which is Chloride base. So, we now wet the board with a Chloride - Hydrochloric acid; now it is introduced into a tray containing Palladium Chloride which is our catalyst. You remember the equation that I have put up in the last class, where we describe, how the Palladium is absorbed on the surface of the copper and then to the surface of Palladium your copper gets attached; it is the strong physical bond and this again is very key to the reliability of the entire board. Now after the recommended time in the catalyst, the board is now wash well, water rinsed for about 30seconds. Remember the water that we use here is deionised water, which is free from all contamination like Chlorides Carbonates and so on.

Now, the board is introduced into a post activator: you remember the post activator will actually break the embryo containing the palladium colloidal gel into the individual palladium atoms and they will be evenly distributed on the surface of copper as well as the hole walls. So these are key processes in the manufacturing sequence. It could be 8 layer - multilayer board or a 16 layer - multilayer board, they have to undergo these critical process steps.

Now comes, the water rinse after the post activator; now the board is ready for activation in the electroless copper bath. If you recollect, this is the electroless copper bath, the blue solution which basically contains a copper rich aqueous solution. At the time of usage, the reducing agent like a sodium hypophosphite or a hydrazine based reducing agent can be added. There are several reducing agents that can be used based on the recommendation of the manufacturer of the catalyst, because there has to be some compatibility. You can see that the board is plated with copper, this is real nascent copper it can get oxidized; so the subsequent process has to be immediately carried out otherwise you have to protect the copper with some kind of anti tarnish solution or a coating.

The next step in the PTH sequence is the panel plating process, which means the copper panel or the laminate - the entire panel gets plated, where as in the pattern plating as the name indicates you will specifically open up those areas to receive plating.

In the plating process, what you see here is basically a jig is attached to the panel for easy handling. For the copper plating bath as you know, there is an electrolyte, there is an anode and the cathode; your PCB acts as the cathode. Here you see that the technician is now introducing the jig containing the panel, the printed wiring board which is a cathode into the copper plating bath, then the current density about 20 amperes per square feet is now activated for the plating bath. So, you have 2 anodes on either side here, as well as here, you have the anodes in a bag so that, you get high purity anode getting in to the electrolyte.

Now after sufficient time, here in copper plating you have to calculate the area to be plated; there should be constant agitation, there should be good air circulation and the distance between the anode and the cathodes have to be measured and accordingly you have to set the current density. Typical guidelines is 20 amperes per square feet and it depends on what thickness you want to plate your final thickness, if it is around 32 microns or 35 microns accordingly; based on that, you can see that this panel is very shiny, a good coverage of copper has taken place on the entire surface of the panel.

If you go deeply and carefully into the microscope, at this stage to look into the through holes they will be well plated, you can see from a side shot here that the holes are well plated with the pure copper. Now, again this is going to be a key step. As the name indicates in panel plating, the entire board is plated; now you can do an image transfer on the printed wiring board and then remove the unwanted coppering.

So, you are now going to protect the areas that you require and then leave out the other areas for etching. Now, we will go in to pattern plating; so here the difference is that you will - after the electroless process - do image transfer on to the panel and then mask the areas that do not require additional copper plating. So for this particular step, you do acid cleaning first, because the copper that is going to be plated on the circuit areas the base area needs to be clean for good addition of electro copper.

Now you can see here, in this particular shot that the image transfer has been done by means of a photoresist material. The blue material that you see here is a photoresist material, that has to withstand all the chemical reactions in the plated through hole lines and in some cases, as we have seen earlier they also act as a etch resist.

During the plated through hole process sequence, you cannot ignore any step which includes a water rinse or an acid cleaning or a microetch process as the process step indicates, because if you miss out one of the steps or ignore one of the steps you are going to contaminate one chemical into another tank and then you will have lot of process variation and that will affect the final yield.

(Refer Slide Time: 12:42)

You can see here now, that the exposed copper which is going to receive plating is micro-etched for about 60 seconds using Ammonium per sulphate micro-etchant. This is basically, again to improve the addition and to make sure that the electro plated copper adheres well to the pattern area. The dry film as I said is going to protect the non circuit areas from receiving copper. We can also see in this particular shot that the panels have a buzz bar around the sides; that is basically for a jigs to hold the panel and also to receive the current through the jig, so that plating or deposition can take place at a inform rate.

Another aspect that you have to see here is, as we watch this video this is the electrolyte, this is the acid copper bath (Refer Slide Time:13:39), typical timings are indicated here about 30 to 40 minutes for 20 amps per square feet you will get about 20 microns deposited; again you see here, after every process there is a water rinse to avoid contamination.

Now, these area that is given around panel is basically for the jig to be held and to provide even plating, you have to make sure that your electro plating bath provides even plating from end to end along the entire panel. If there is uneven plating then you will have electrical issues finally, because it is not a complete translation of the electrical design.

There is a 10 percent sulphuric acid dip for about a minute or so and then it is this copper that is plated exclusively on the pattern areas. It is now further protected using acid, tin pattern plating using an acid; tin plating bath which contains the tin rich source and then typically the current density about 20 ampere per square feet for about 20 minutes. Here essentially, we are immediately protecting the copper from atmospheric oxidation by tin plating. You can see here the entire pattern areas, as well as the side bar areas are plated with tin.

Now, the tin can act as a etch resist. In this case, you can see now that there is a tin plated area and on the contrast you see the dry film area which as protected the non-circuit areas so far. So you have now seen, what is the pattern plating and panel plating. We go into final step of the PTH sequence which is basically removing the dry film as well as performing the etching operation.

So, the panels are introduced in to alkaline sodium hydroxide caustic solution, typically about 5 to 10 percent of hydroxide solution and you can see that the dry film is lifted of it is basically striped of the panel. What you see here is basically the pattern areas which are protected with tin or plated with tin and then you will see here the brown areas which are the copper which is going to be removed by the next step, which is etching.

So, the next step is basically, in this case, conveyorised alkaline etching using alkaline ammonia. We have seen earlier the different types of etching; alkaline ammonia basically contains ammonia hydroxide, ammonia chloride and liquor ammonia in addition to copper as a catalyst to start with.

(Refer Slide Time: 15:00)

You can see that, there are spray nozzle from the top and as well as from the bottom going through a conveyorised machines and there is a water rinse at the end of the conveyorised line. In some cases, as you can see in this particular example that after the first pass the board is not completely etched; you have to set the right temperature, there has to be complete control of the elements in the alkaline etching bath and now the board is reintroduced into the conveyorised etching machine, further to remove the existing copper.

So in some cases, if it is long line and if it is a large voluminous tank then you can probably complete the etching process in one go or one pass but, being a proto type this now happening in the second pass. You can see here that the board is completely etched out, you can see the base laminate and you can see the copper pattern areas which are intact. So, have a microscopic examination at this particular point, to see if the etching process has been completed thoroughly without any undercut; that is a very crucial examination, quality examination that one needs to do.

(Refer Slide Time: 18:18)

Now, the particular board has been solder masked, you can use a dry film or a liquid solder mask and then complete the operation. You can also see that the legend printing has been done on the right side; you can see legend print using white ink have been marked.

(Refer Slide Time: 18:31)

Post operations

1. Quality Inspection (at all stages)
2. Solder Masking
3. Legend printing
4. Scoring and Routing
5. Edge chamfering
6. Bare board Testing
7. Packaging/Shipping

So this completes basically, the plated through hole process sequence for doing a double sided electrical interconnection. Now you can repeat these steps for multilayer board, but at the same time you have to look at final measure thickness; therefore, you will use thin laminates for inner layers. What are the post operations that one does after the board is completed? Quality inspection is a must at all stages, so I would mark this as very important because in the volume manufacturing you have to incept the board at every process step.

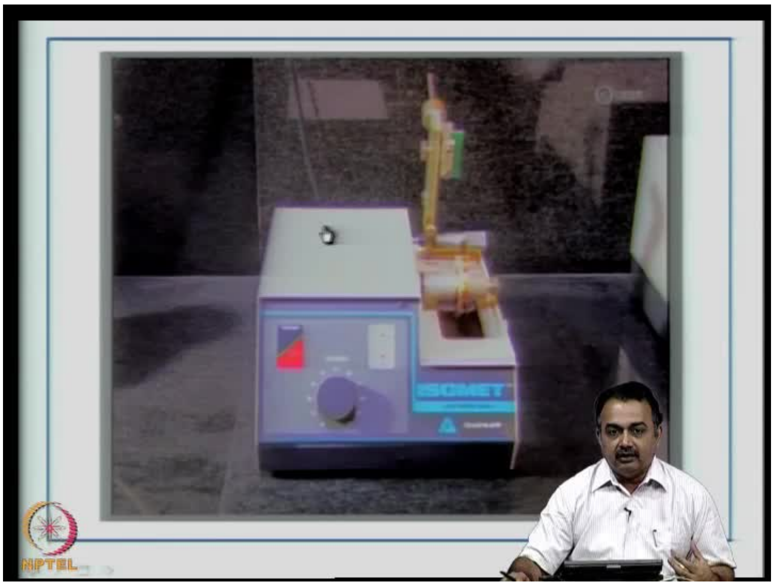
Then solder masking, legend printing using white inks, scoring and routing because in some cases the panels may have a large number of boards; so you have to isolate this boards. In some cases, in the individual boards you may have to perform some kind of a routing or opening a slot; that is basically for introducing an electromechanical device or a chassis part of it into the board. You can do edge chamfering, typically making the board edges smooth enough, typically the board edges will be sharp after the manufacturing process is over, but then you can smoothen it out using edge chamfering.

Then the board will be done or sent through for electrical testing, in this case it is known as bare board testing. Here, you will do an electrical test that is basically aimed at looking at shorts or open, whether the there is a problem in the manufacturing that as resulted in a short or open. So these kinds of indicators will be given to the manufacturer for every single board. You have to qualify every single board for all of these quality inspections including electrical testing and finally, the board is packaged according to the recommended packaging material and methodology and then shift.

Now, I mention about the plated through hole process sequence. What you will notice is that, the board reliability rest on the through hole quality. So, how do I measure the quality of plating and the thickness of the copper plated inside the through hole wall of the printed wiring board? Obviously, you cannot examine every hole, so you have to prepare some coupons from probably 1 in 10 panels that will give you an indicator of how your plating bar and how your processes work.

PWB Micro sectioning is one such example were you prepare coupons for testing the quality and testing the nature of your plating process because that is a key to the yield part of it in manufacturing. So, I will take you through this video where we show you how to prepare a sample of a plated through hole, which will be used for analysis and also measuring the copper thickness.

(Refer Slide Time: 22:10)



Basically, we prepare some test coupons using some epoxy material and then we will zero in on the site, where the sample needs to be measured and then microscopically look at whether there are cracks, whether there is a de-lamination from the base substrate and the nature of deposit, whether it is pores or hard and whether it is having the required thickness. You see in this particular instrument which is basically a sawing machine, where you can see here the PCB a small sample which contains the through hole has been clamped (Refer Slide Time: 22:44).

Then you can see here (Refer Slide Time: 22:53), that there is a saw typically a diamond saw that is activated. There is a liquid here, typically water to remove the heat while the cutting process is on; the board is introduced to the saw. Now, cutting takes place at slow speed because you do not want to damage the sample and the cutting is now taking place typically at the right, near the through hole.

So, you would not do not want to damage the through hole (Refer Slide Time: 23:32), you are going to cut this very close to the through hole and then using polishing methods, you are going to approach the site. Here, you can see that the technician is introducing the PCB which contains the through holes (Refer Slide Time: 23:42); you can see the through holes into a template.

Now, you can see here this is a template, which is now introduced into a container and here you can see (Refer Slide Time: 24:02), we are now mixing an epoxy material and on the right you will see a hardener is being used. So, based on the recommendation of the manufacturer epoxy and a suitable hardener is mixed thoroughly without air bubbles and slowly introduced into the plastic container, which contains the printed circuit board coupon.

Now here, you have to make sure that you do this process very slowly lot of heat will be generated (Refer Slide Time: 24:32), this is exothermic reaction and you have to introduce the epoxy material slowly into the container to make sure that there are no air bubbles. Typically, you can keep the epoxy material to stabilize enough after it is mixed for about 10 to 15 minutes and then introduced into the container.

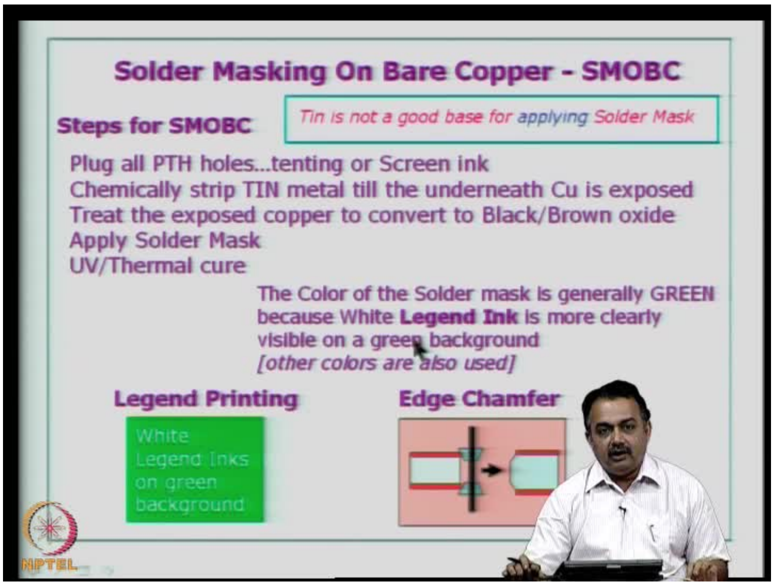
Now, this is a quick set polymeric material (Refer Slide Time: 25:00); it will set quickly in about an hour, now you can see here that the coupon has been completely set; you can now release it from the container. Now this sample, which is their embedded in this epoxy is now you can see here, that the PCB is now embedded into the epoxy coupon. This is called a plotting process using an epoxy and you can say this is a cross section sample. We are now at the intermediate stage of looking at the site where you want to measure the copper thickness.

After this process is over, we now had to polish the sample to get close to the area; we are looking at few micron thicknesses. Therefore, the polishing process from that raw coupon has to be very slow. So here, you can see there are various materials used typically emery paper of various grades that we normally use for polishing (Refer Slide Time: 26:03), but here we have to go from course to less courser and more fine emery paper material. You can also use some kind of slurry - alumina silica gel; for example, to aid you in the polishing process. This container which contains the emery paper also holds the sample (Refer Slide Time: 26:35), that we have just prepared and then using some kind of a slurry; typically water and you can see here alumina silica gel is now introduced and then continuously polished at very slow rates or at high rates initially and then slowing down the rate of polishing. Here we can see, (Refer Slide Time: 27:01) the surface of the coupon was done after the coupon was released from the plastic container.

Now, after polishing you will slowly approach the PTH area, you can see here (Refer Slide Time: 27:18), very carefully the two plated through holes and the copper very clearly visible to the naked eye; you can now introduce it into the various microscopes that you might have and also measure the dia as well as the thickness.

Typically, this is important and a must process in the volume manufacturing also. When the drilling process starts for the certain panels they will drill this coupon of various drill sizes. Then the quality team will have to do samples for each of these coupons or earmarked areas and then they have to report back to the manufacturing line about the quality of the deposit and how much title control they have in the deposition rate; because here the electrical performance is going to depend on the thickness of the copper deposit that your manufacturer can provide you, because you cannot plate too much of thickness of copper in more than necessary and also you cannot have lesser thickness other than the specified.

(Refer Slide Time: 28:42)



Solder Masking On Bare Copper - SMOBC

Steps for SMOBC

Tin is not a good base for applying Solder Mask

Plug all PTH holes...tenting or Screen ink
Chemically strip TIN metal till the underneath Cu is exposed
Treat the exposed copper to convert to Black/Brown oxide
Apply Solder Mask
UV/Thermal cure

The Color of the Solder mask is generally GREEN because White **Legend Ink** is more clearly visible on a green background *[other colors are also used]*

Legend Printing

White Legend Inks on green background

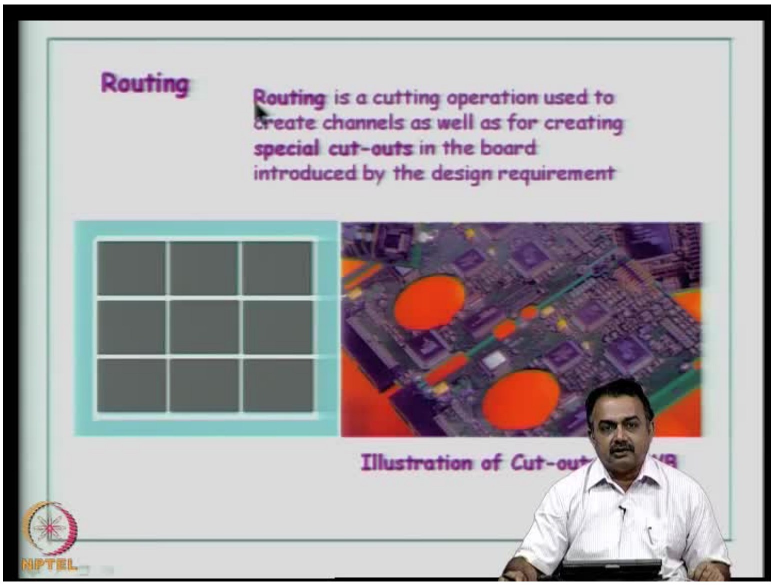
Edge Chamfer

So, these videos have given you highlights about the stricter quality control that establishes in today’s manufacturing process, which the designer should be aware of. Now, we look at the post process for example, solder masking on bare copper it is called SMOBC. Now you have protected copper with tin, but tin is not good base for applying solder mask, it can peal of because the surface of tin is very smooth, it does not provide good addition. Typically, some people opt for applying solder mask directly on to the bare copper, after treating the copper sufficient enough to hold the solder mask.

What are the steps for a SMOBC? Plug all the through holes by either tenting using a dry film process or ink printing using a stencil or a screen printing process; chemically strip of the tin metal until the copper is exposed. This as to be carefully done removal process, otherwise you will remove the copper also. So, based on the information on the manufacturing line about the thickness of tin that you have coated, you can safely remove the tin plating. Treat the exposed copper to convert to a black or brown oxide, this is basically a surfacial treatment that you can give to convert the exposed copper on the surface to its oxide and then you do not have to convert the entire copper. So, once the top atomic layer of copper is converted to its black or brown oxide different compositions, you can now apply the solder mask using stencil printing or a dry film, or a liquid solder mask using curtain coating. Then you cure it using thermal process or UV curing process; after that you can do the imaging is part of the process and then you can remove the unwanted solder mask, so this how SMOBC is done.

Now, the color of the solder mask is generally green and white legend ink is typically used on the screen background; so that there is a very good visibility there can be other colors, but this are general standards. The picture on the right shows you how edge chamfering can be done to the edges of the PCB.

(Refer Slide Time: 31:21)



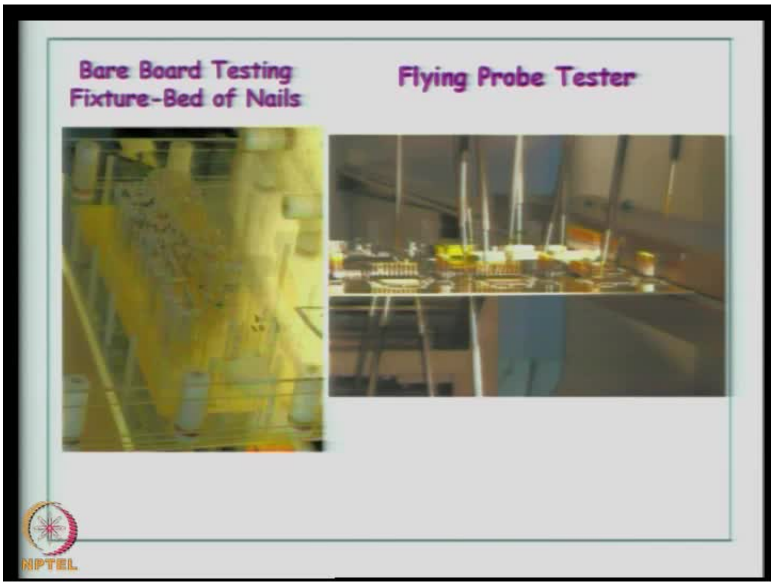
Routing

Routing is a cutting operation used to create channels as well as for creating special cut-outs in the board introduced by the design requirement

Illustration of Cut-out

Now, routing is another process that is done on completed or finished boards. It is a cutting operation used to create channels. You can see in this particular picture, right at the center of the PCB a particular channel has been cut open. There are no circuits in these areas but, typically this are used for holdings some kind of a mechanical device or introducing the PCB in a chassis, which contains some kind of a protruding mechanical device and so on. These are special cutouts introduced basically based on the product design requirement.

(Refer Slide Time: 31:58).


Bare Board Testing
Fixture-Bed of Nails

Flying Probe Tester

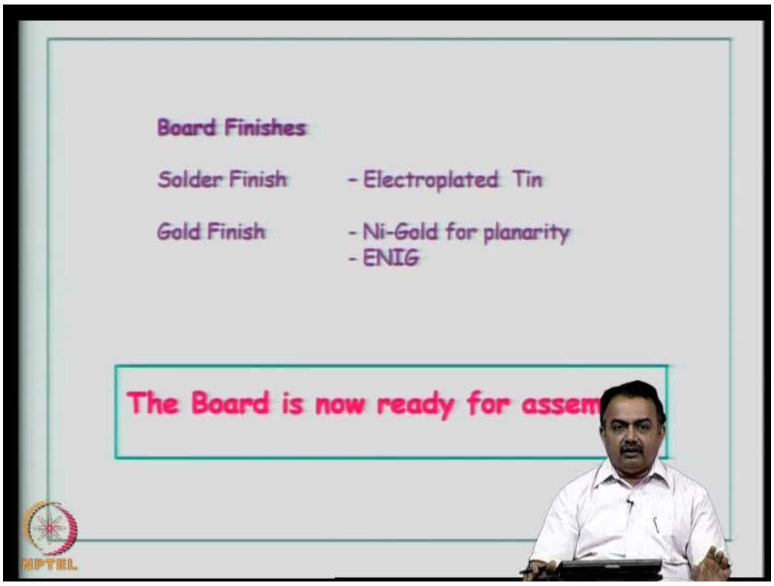
I talked about bare board testing; on the left you see here, there is a bed of nails and these are used by all manufacturers as a compulsory test for qualifying every single board. What does this mean? There is a circuit pattern for your board and that is dominated by your net list. So based on the nets, they will produce a bed of nails; you can see the long protrusions these are stainless steel pins or nails that typically depicts your circuit pattern and this is done on some kind of a plastic mold a typically polycarbonate and so on.

Then your board will come and sit on this bed of nails, based on the through hole coordinates they will perfectly stud fit into the bed of nails. Then you will now look at the parameters like shorts or open on your test board by applying a small voltage. You will now activate the board to get some information about which nodes or which part of the nets is open or shorting.

So, you can mark those areas and if this is going to be a consistent or persistent defect, then there is some process defect which can be identified and the information can be passed on to the particular section of the manufacturing for identifying the defect and rectifying it. Otherwise some kind of repair can be done, if you have very small defects. For example, it could be a lift off or it could be some kind of a broken line due to improper handling during the manufacturing process and so on.

Now, for assembled boards what you see here on the right is the picture showing you what a flying probe tester can do for assembled boards. So, in a completed bare board you have to do a bed of nails test, but in the finished board with components typically you will use a flying probe tester. What is a flying probe tester? Basically there will be different probes and there will be the CAD data that will be loaded to this equipment, based on the net list. Now, one of the probes can sit on one end of the net and the different probes can go at the different nodal points of the net and say tell you which segment are not working or which segments are perfect because this now dominated by the soldering process that you have done.

(Refer Slide Time: 35:45)



Board Finishes

Solder Finish - Electroplated Tin

Gold Finish - Ni-Gold for planarity
- ENIG

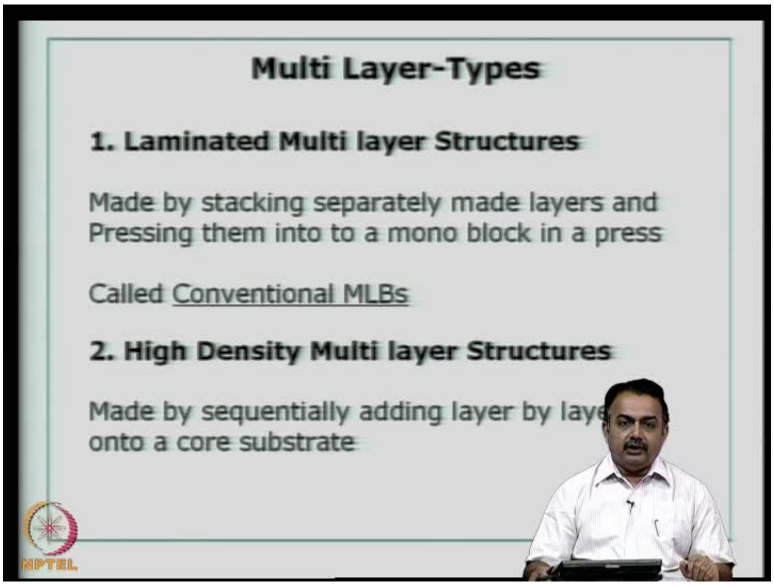
The Board is now ready for assem

This has to be very slow process and again there is a hindrance in terms of the components mounted, accessibility is a problem. You can do it simultaneously at both sides of the board as you can see here. So, you can use different probe points simultaneously to check a particular net and see the failure at different places. These are equipments that are used, one on the left at the manufacturing site of the PCB and on the right in the assembly segment.

Now in general, board finishes will be either a solder finish. So, you can expect electro plating of tin to take place because tin lead and lead is not been used today or not recommend to be used; we expect that solder finish using electro plated tin is an accepted process or you can have a finish with gold.

I mentioned earlier, when you plate gold you have to plate an undercoat of nickel and then have gold plated on the nickel to prevent gold diffusion into copper. So, today electroless nickel-gold is a popular process in many countries and the thickness is well controlled by electroless nickel-gold. The board is now ready for assembly at this point of time.

(Refer Slide Time: 36:35)



Multi Layer-Types

1. Laminated Multi layer Structures

Made by stacking separately made layers and
Pressing them into to a mono block in a press

Called Conventional MLBs

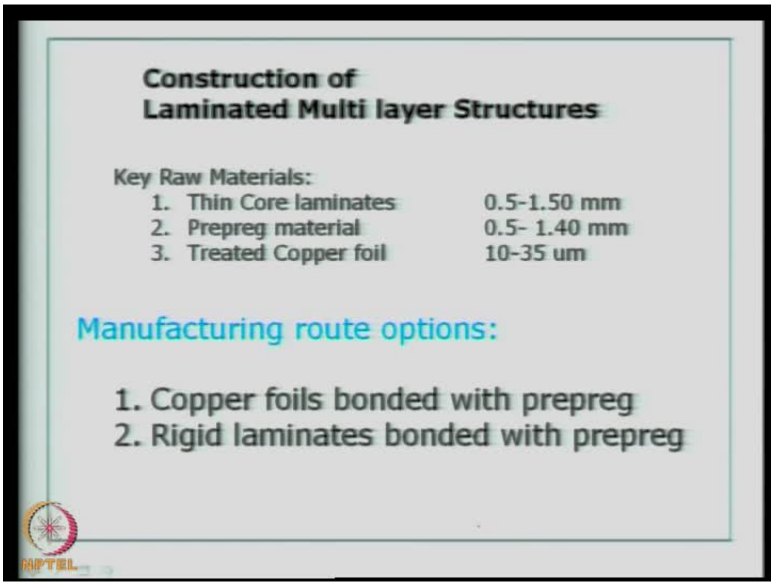
2. High Density Multi layer Structures

Made by sequentially adding layer by laye
onto a core substrate

Now, we look into some aspect about multilayer boards, so far we have seen double sided boards. Multilayer boards laminated multilayer structures are made by staking separately made layers and pressing them into a mono block in a multi layer press. So, you can have a double sided board and another double sided board you can introduce them into a multilayer press, separate them by a dielectric material called the prepreg, as you are aware and then they are pressed in the multi layers press to get a 4-layer structure and to the required thickness.

These are all called conventional multi layer boards, because today you have seen what is known as a high density multi layer structure made by sequentially adding layer by layer copper and dielectric on to a existing core substrate. So, this basically is to what is known as high density interconnects structure. You are aware of the term high density interconnect structure, where we are typically talking about existing core layer; it can be a 2-layer board and then you add one layer at the top and may be another layer at the top then have a dielectric and another copper layer here.

(Refer Slide Time: 38:10)



Construction of
Laminated Multi layer Structures

Key Raw Materials:

1.	Thin Core laminates	0.5-1.50 mm
2.	Prepreg material	0.5- 1.40 mm
3.	Treated Copper foil	10-35 um

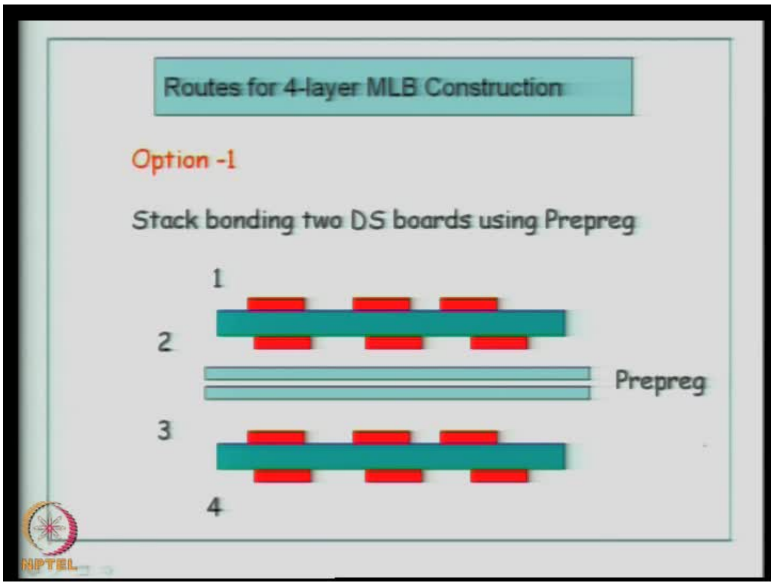
Manufacturing route options:

1. Copper foils bonded with prepreg
2. Rigid laminates bonded with prepreg

So, you can have five layers built by sequentially adding copper and dielectric material. The construction of laminated multi layer structures the key raw materials are you should have thin core laminates; typical dimensions could be from 0.5 mm to 1.5 mm, prepreg material 0.5 mm or even downwards 200 microns 0.2 mm to 1.4 mm. These are all based on the kind of final dimensions that you require as per your design, treated copper foil because as you built the multilayer structures you are introducing copper foils in some cases or you can also add copper by plating.

So typical thicknesses are from 10 to 35 microns, again these are dependent on the electrical design specifications. The manufacturing option will be first, copper foils will be bonded with the prepreg material. This is the conductor and here you have the insulator, this will be introduced in to the multilayer press along with the core and then, structures can be, layers can be increased.

(Refer Slide Time: 39:45)



Routes for 4-layer MLB Construction

Option -1

Stack bonding two DS boards using Prepreg

The second point using rigid laminates bonded with prepreg material. We are going to see both of these options. So, in the second case rigid laminates; let us see a 4-layer board is now introduced with two prepreg layers, one at the top and bottom and then you can have copper foil on both sides to get a 6-layer board. Let us look at the case study or an example for a 4-layer multilayer board construction.

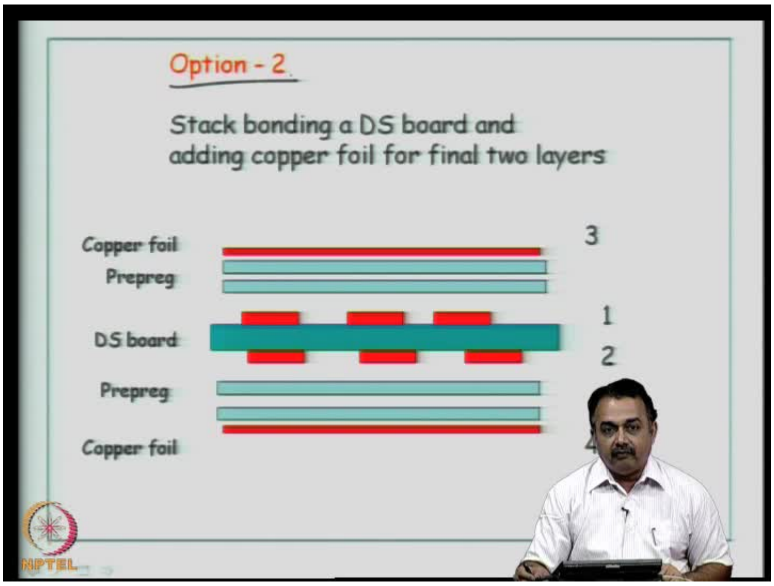
Option-1: The first thing here is stack bonding two double sided boards using a prepreg material. Now assume that you have done this part - this is a double sided board: you can see here, there is copper; you can see here, there is copper and this is a two sided printed wiring board. Similarly, here this is a two sided printed wiring board, which you have done as per the process that we have just described; this is a copper, here you have the copper.

Now, the design has been in such a way that if you introduce these 2-layer boards together using a set of prepreg materials. Here in this particular example, I have shown you there are two prepreg sheets, the thickness can be varying from 200 microns to let say 500 microns, for each of these introduce them and then stack them together and send it to a multilayer press.

Now the boards will become, the epoxy will melt and then they will nicely sandwich between the two double sided boards. Later, you can introduce a through hole from top to the bottom; let us say interconnect layer one to layer four. You can also build copper on one side by adding the dielectric material or you can do the imaging process at the top layer and the bottom layer subsequently after the drilling process is over.

So for example, if you want interconnect layer one to layer four, you can drill a through hole and then you can make it conductive. You can access from here to here by using a plated through hole process sequence; so, these are the design variations which can be very clearly explained for the manufacturer as to what you require.

(Refer Slide Time: 42:10)



Option - 2

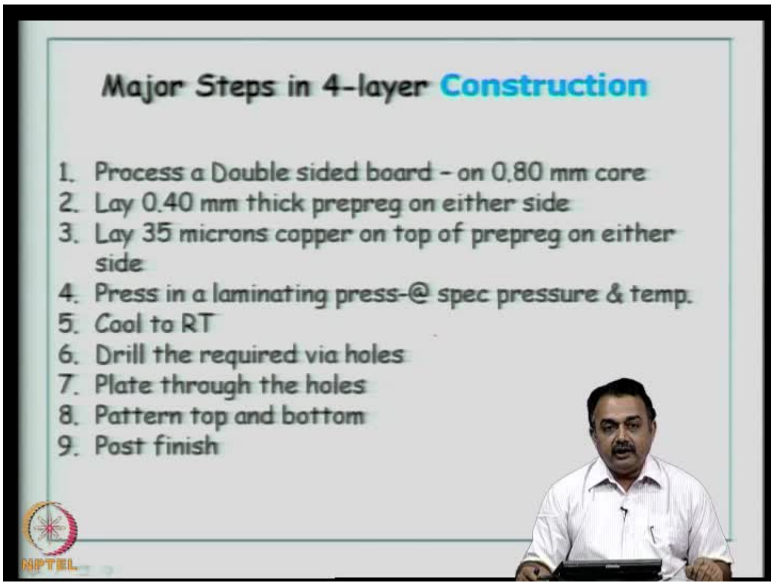
Stack bonding a DS board and
adding copper foil for final two layers

The second option is that, you can do stack bonding of a double sided board and adding copper foil for the final 2-layers. In the first case all your imaging is complete and now here, we are talking about a double sided board. In this particular case, a double sided board with the copper and the dielectric material imaging is complete. Now, we introducing the prepreg material here and then this is the copper foil, again another copper foil, it is now sent to a multilayer press; so at the required or specified temperature and pressures there is good bonding.

Now, you can again - introduce a through hole; you can do imaging here, image your copper areas at the top and bottom and then establish connection between layer three and four, in this case as notified (Refer Slide Time: 43:06).

Here again, you will do a dry film process; you can do a pattern plating if you require: it depends on what kind of copper foil, thickness you are taking. Typically, it will be thin then you can add enough copper to create your required copper thickness top and bottom. You can also establish connection between the layer one to the inner layer of copper, here again you can do that and you can make it conductive. This hole wall can also made conductive; so, this is the through hole, and this is typically a blind via and is again a blind via.

(Refer Slide Time: 43:57)



Major Steps in 4-layer Construction

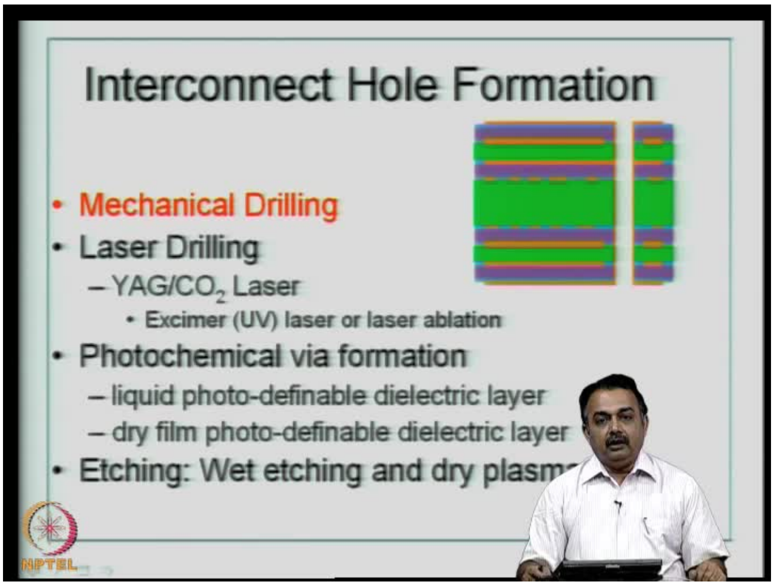
1. Process a Double sided board - on 0.80 mm core
2. Lay 0.40 mm thick prepreg on either side
3. Lay 35 microns copper on top of prepreg on either side
4. Press in a laminating press-@ spec pressure & temp.
5. Cool to RT
6. Drill the required via holes
7. Plate through the holes
8. Pattern top and bottom
9. Post finish

Here you may probably have through holes, which will now become a buried via. So these are the various schemes. In the nut shell, what are the major steps in creating a 4-layer board process? A double sided board or let us say 0.8 mm core, you are starting double sided board will be 0.8 mm.

Now, you will lay 0.4 mm thick prepreg on either side. So it is almost 1.6 mm thickness now, then you add 35 microns of copper on top of prepreg on either side so that completes the copper construction; introduce it in to lamination press at specified pressure and temperature, it is a very slow process.

Now cool it to room temperature, drill the required via holes to connect the top and the bottom, then plate the through holes through the sequences that I mentioned earlier pattern plate top and bottom and then post finish using tin or you can use nickel-gold then it goes for solder mask then it goes for legend printing.

(Refer Slide Time: 45:18)



Interconnect Hole Formation

- Mechanical Drilling
- Laser Drilling
  - YAG/CO$_2$ Laser
    - Excimer (UV) laser or laser ablation
- Photochemical via formation
  - liquid photo-definable dielectric layer
  - dry film photo-definable dielectric layer
- Etching: Wet etching and dry plasm

This is the kind of steps that you will look at, as a designer how your 4-layer board is going to be manufactured. So, having seen the basics about double sided board and multilayer boards will now slowly get in to the topic of high density interconnects.

The interconnect hole formation as you have seen here, basically is by through hole buried or blind via. They are typically done by mechanical drilling, but here in mechanical drilling there are certain limitations. What are the limitations? They are basically the size of the drill bit, the size of via that you can drill.

Today’s modern equipment can drill very safely, let us say up to 0.3 mm or 300 micron but, below that let us say if you want to do 0.2 mm or 200 microns it is expensive because drill bit is expensive, the equipment also becomes expensive, because of the high rpm that you have to use for lower drill bit sizes.

But today, we have mechanical drill equipments that can drill up to 0.1 to 5 mm or 125 micron. So, lot of quality control needs to be there especially, if you want to plate a through hole which is having a high depth, when you are using lower drill sizes you have to make sure that the thickness is small, so that you can maintain the aspect ratio.

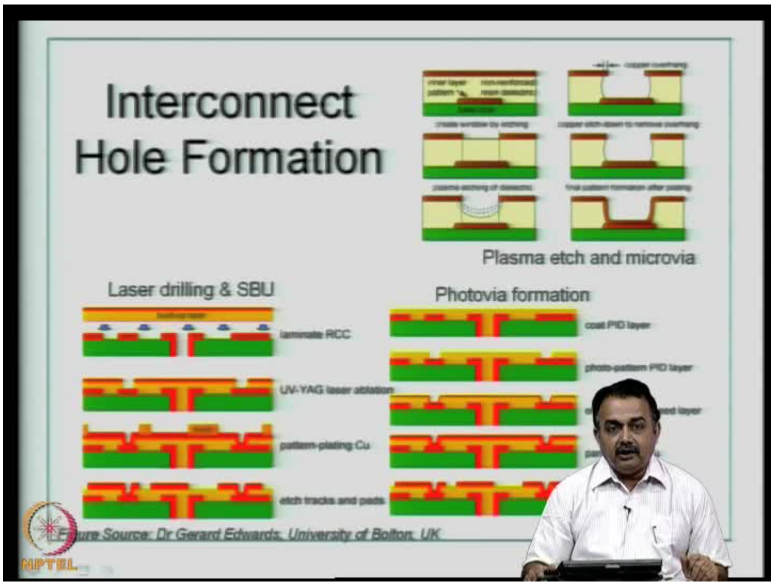
The other option today that is available for you building high density interconnect is to use laser drilling and do away with mechanical drilling. What are the lasers that you can use? You can use carbon dioxide laser or YAG. YAG stands for Yttrium aluminum garnet. Basically, you are going to use excimer laser or you are going to ablate or burn away the material in order to produce a hole. So, if there is a substrate you are going to burn away this material here and then remove the dielectric material, so that you get the hole.

Now, the process looks simple on description, but again we are talking about the quality of the hole that is required, if you are going to use laser drilling for a through hole with a moderate aspect ratio. Then follows the adaptively or the condition of the hole wall for subsequent electroless plating, because if you drill the hole wall which has got a lot of debris and defects then it is going to be reflected in your electroless copper plating. Therefore one has to be careful about what kinds of a methodology you are going to use but, laser drilling equipments are becoming very common and they are affordable. They have indicated that, you are going to get very high through put and yield in terms of number of PCB’s produced using these laser drilling machines.

The other step is using a photo chemical via, where we use a liquid photo-definable dielectric layer and you are going to use photolithography to open up the via, from this very thin dielectric layer material. So, it can be either liquid or a dry film but, the dry film photo definable dielectric layer is not available in abundance. So, most people tend to use liquid dielectric layer material to produce photo definable microvias.

When you use these kinds of methods other than mechanical drilling, we are talking about producing microvias not the vias that we normally use for component mounting, which is plated with copper. So, microvias by definition is typically those vias which have dimensions of 150 micron and less. Today, it is redefined as 125 microns and less. The methodology for microvias can either be laser drilling or photo chemical via formation.

(Refer Slide Time: 50:37)



Interconnect Hole Formation

Figure Source: Dr Gerard Edwards, University of Bolton, UK

There is another method called wet etching, but it does not produce the kind of results that we expect; for this kind of high density interconnect structures using a dry plasma etch has been tried in prototyping, it produces better results in terms of the hole wall definition and the kind of undercut. When you talk about etching, there is always a problem with undercut and wet etching produces high undercut compared to dry plasma etch but, it is a very slow process.

We look at the different interconnect hole formations from this figures. The photo via formation happen this way, as you can look at this picture on the right, here you can see that there is a photo imageable dielectric layer that is coated on already existing core substrate. You can see the yellow line that is the photo imageable dielectric material that is coated, it tends the through hole, and it also covers the non circuit areas on the board the red areas are the copper patterns that have been established already.

In the second step you can see that, an imaging process is carried out a photo patterning process is done. If you can show this picture clearly, completely, that the microvias are generated here by photolithography process.

The next step is doing electroless copper plating process. Now, you can see here a thin layer of copper has been found using electroless copper, it covers the entire depth of the microvias. So, the microvias becomes conductive and it also contacts the layer below.

So this microvias, after electroless becomes complete conductive touching the base copper immediately below the dielectric material. Then you can add the required amount of copper that you require by following it up with an electroplating process, so the thickness is increased.

Now, you can do further an imaging process, you can do another imaging process and then you do print and etch and then define the tracks and the pads. You can repeat this process a number of times to get the number of layers. So, this is the essence of a photo via formation.

This can be done at the top of the copper or top of the printed circuit board core and also it can be done at the bottom. You can do it separately or simultaneously, the only thing is that there should be very good registration between the microvias and the through holes or the patterns and the pads that you have defined. So, you have to generate masks for each of these layers. On the left if you see, we are going talk about laser drilling and how you can use laser drilling for sequential buildup circuitry.

So, you have a base core that is already generated and you are adding a laminate RCC that is Resin Coated Copper foil. You can see here(Refer Slide Time: 53:41), this is introduced and then there is lamination process that is done; so the dielectric will ooze out upon these temperature and pressure and that is applied. Now, you can drill on top of the copper and the dielectric using UV YAG laser Yttrium Aluminum Garnet. It is basically an ablation, it removes the copper and it removes the dielectric material; so, you can see here the opening as been created.

Now, you can do the copper addition by pattern plating by applying a resist or a photo mask. You can see that the microvias has been plated here and then you can define the pattern by subsequent etching process and defining the tracks and the pads. So this can be done for the bottom side also. This is a similar process compare to a photo via process only difference is that, here you use a laser source and you can do you can get very good definition using laser drilling process. On the top, you see here the plasma etch to create the microvias and how it is suitable for SBU process.

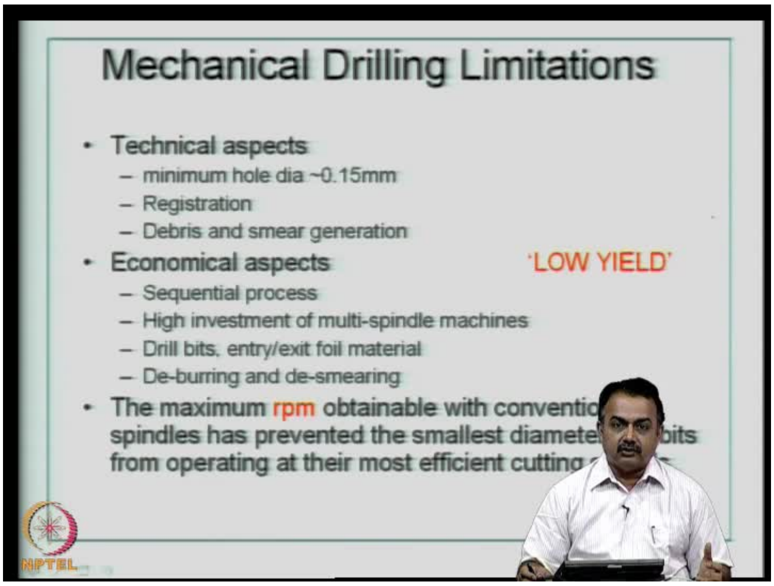
You have a base core here, inner layer pattern you can see and there is a non reinforced resin dielectric material. Then you create a window by etching and then introduce plasma etching. So basically, the plasma is generated in a chamber were for example, you have two gases. Let us say CF four and oxygen interacting together and you get plasma from the resultant free radicals or ions at higher frequencies.

Now, these resulting ions or free radicals will react with the dielectric material and basically it is a slow process, but you can see here the material is ablating. The dielectric material is removed here, you can see in this particular figure. Then the copper etching is done basically to remove the kind of protrusion that you can see here, that the plasma etch has not taken care of.

You can do a very quick copper etch, then you get a perfect hole wall and this is finally converted into a interconnect by plating; you can do a electroless plating followed by an electro plating to get well defined hole wall. So, this is a very good description about plasma etch and how plasma can be used, this is the dry plasma etch process to create microvias and following the creation as microvias it goes into the layer formation by addition of electroless copper and electro plating copper.

So, the summary here is the interconnect hole formation for high density interconnects can be done by mechanical drilling in one case, but for vias which have a smaller diameter people are going in for laser drilling process - plasma etch and photo via formation. As I said before, mechanical drilling to the extent of 125 microns is currently available although expensive and if you include that, these are the 4 major methods to create a microvias.

(Refer Slide Time: 57:50)



Mechanical Drilling Limitations

- Technical aspects
  - minimum hole dia ~0.15mm
  - Registration
  - Debris and smear generation
- Economical aspects 'LOW YIELD'
  - Sequential process
  - High investment of multi-spindle machines
  - Drill bits, entry/exit foil material
  - De-burring and de-smearing
- The maximum rpm obtainable with conventio spindles has prevented the smallest diamete bits from operating at their most efficient cutting

The basic reason is that mechanical drilling has its own limitations, because the minimum hole dia you can create is 0.15, registration is a problem and debris and smear generation during drilling can block the holes, you need proper hole wall preparation before electroless plating. The economical aspects is it has got low yield, because you are going to depend on sequential drilling process and it can take time; whereas the processes like laser drilling and photo via formation especially is a parallel process. Although laser drilling is the sequential process, photo via formation can be very economical and it can increase the yield because it is a parallel process.

High investment is required for multi-spindle drilling machines, drill bits are expensive; you have to use entry and exit foils. De-burring and de-smearing is very important operation to remove the debris and to prepare the hole wall conduced for a electroless plating. The maximum rpm obtainable with conventional spindles as prevented the smallest diameter drill bits from operating at their most efficient cutting speeds which results in breakage of drill bits and so on.

So, we will look in the next class, a comparison of all of these various methods to create microvias and look at how actually you build a high density interconnect structures from a given core substrate.