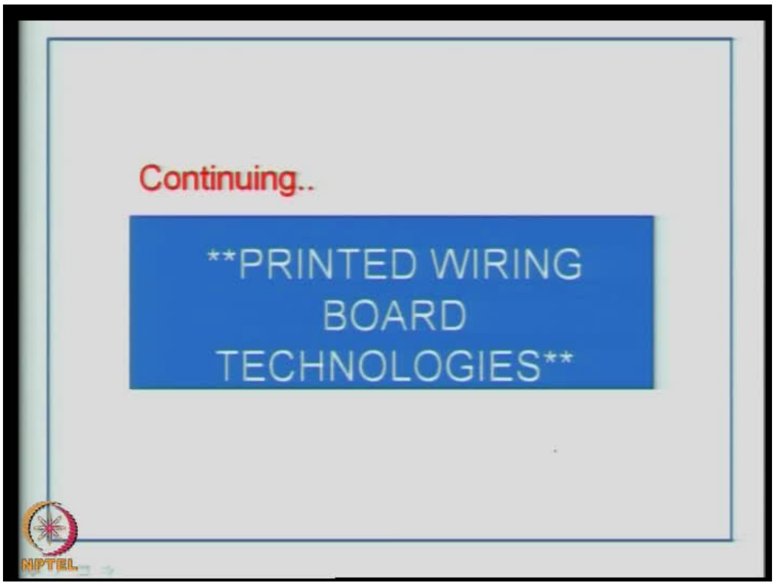
An Introduction to Electronics Systems Packaging

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

Module No. # 06
Lecture No. # 28
Through-hole manufacture process steps
Panel and pattern plating methods

We will continue with this topic; the module on printed wiring board technologies.

(Refer Slide Time: 00:18)



This is the sixth hour in this particular topic, we have gone a long way in looking at the basic technologies that are required or that are needed to be understood by a designer, so that the design from the manufacturing term can be well understood. Because, we are also tagging along with this DFM and DFR, this means design for reliability and design for testability.

The goal of this particular chapter is to help you and get sensitized to the various process technologies. Here, we are covering the basic minimum process technology features; minimum chemistry is discussed and minimum material information is given. After this chapter, if you got some kind of an insight and if you feel that you need to learn more then there are various books that can help you go through this.

This chapter will end up looking at high density interconnects; it is the sort of ultimate in the current state of art manufacturing technologies using microvias and new chemistries. Before that at this juncture, what I would like to do is to give you a recap of various samples that we have gone through in this particular chapter and that can probably refresh your memories (Refer Slide Time: 02:00).

The first thing that I would like to show you is basically a material here, as you can see this is a prepreg material and all of you know this is a dielectric material. There can be various organic resins that can be used to manufacture a prepreg material. A prepreg material is very important in the sense it gives the electrical characteristics to your system level printed wiring board. It can have two components, essentially one is the organic resin and the other is the filler; storage conditions are very important.

The next sample that you need to have a look at; probably you have seen copper foil; you can see here, this is the copper foil which gets bonded or laminated with the prepreg material in the manufacture or during the manufacture of a printed wiring board - substrate. When you combine these two materials you get a sample like this. You can see here, this is the copper clad laminator and this comes in various thicknesses right from 0.5 mm to 3.2 mm.

The thickness of the copper and the dielectric can vary accordingly; according to the required thickness. This can be used as a system level printed wiring board – substrate, it can be also used as an inner layer substrate in the manufacture of multilayer printed wiring boards. Now, typically you can manufacture a single sided board. As you can see here, in this case the printing or the circuit areas will be on one side of the substrate and then you can have a double sided board as you can see here, this is odd sized PCB,

I am showing you here - typically an irregular form of PCB and a circular printed circuit board. The reason why this is circular is the product requirement demand for this kind of design and shape of the PCB (Refer Slide Time: 03:55). Normally PCBs are rectangular, but you can also have this kind of designs depending up on the product. You can see that this is the double sided board, the holes are plated and you can see a lot of through holes here. These through holes are plated, which means interconnections have been provided between one side and the other.

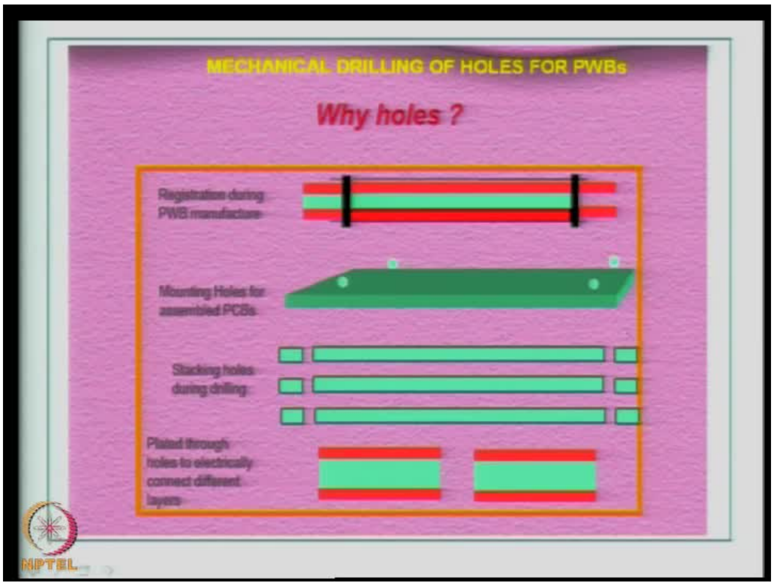
You can also have advanced PCBs like this, which I am showing here; you can see that this surface is gold plated. There can be a few system level printed wiring boards, which require special surface finishes like this (Refer Slide Time: 04:47). This is a gold plated printed wiring board; in fact if you look at it very closely this has got mechanically drilled through holes of the order of 0.15 mm that is, 150 microns. These through holes have been made electrically conductive that means, they are plated through hole in these PCBs and they are highly reliable. So, these are used for specific applications.

This is a high density interconnect structure, we can also have another kind of high density interconnect structures. As you can see here, this is a eight layer sequential build up high density interconnect board (Refer Slide Time: 05:47). You can see advanced packages like BGA, flip chip, lot of QFPs and surface mount resistors that have been mounted; both sides have been utilized. Here, obviously from this vision you cannot look at the cross section, but I can tell you that this contains microvias of the order of 19 microns. You can also make boards like a flex circuit; you can see here, this is a rigged flex board and you can also make a completely flex board.

Now, I have been taking you through the various process steps in the manufacture of printed wiring boards. I will also show you some of the samples that have come out through this process. You can see here, this is a photoresist coted copper clad laminate; the red material dry film that you see here is a photoresist material (Refer Slide Time: 06:41).

Now, this is imaged as you know and then it is developed. So you can see here, after the photoresist process is over this has been imaged, then you can see alongside samples where you have specifically plated on the circuit areas (Refer Slide Time: 07:10). Finally, the material is etched and then you will get a complete printed wiring board. So, these samples will take you through the various process steps of the printed wiring board.

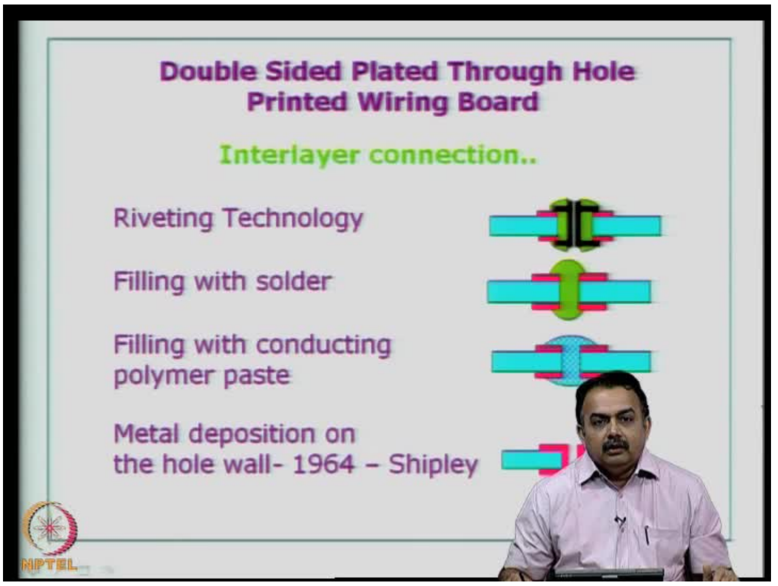
(Refer Slide Time: 07:47)



Today, we are going to focus on the completely plated through hole sequence, through videos and slides. Let us begin with the topic on mechanical drilling very briefly; we have touched upon this in the last class. Why do we require holes? We require holes because we need to register the printed wiring boards during the manufacturing, because when you build up a multilayer substrate; for example, when you build up a four layer multilayer board you need to register each of the layers perfectly using the photo mask and the laminate structure. Therefore, you require registration holes or tool holes for manufacturing. The next requirement is that you may require mounting holes on the PCB, which will eventually go through the mounting of the PCB on the chassis of a product. So, holes are required for mounting holes and for assembling of PCBs.

Stacking holes during drilling; even during the drilling step, when you are stacking, let us say 3 to 4 PCBs at that time, you require all the laminates structures to be stacked perfectly so that there is no miss registration or vibration or movement during the drilling process, because if that happens you are going to one-break the drill bit, you are going to get miss registration and you will end up with the wasted laminate structure. Finally, the important thing is you drill holes, so that you can create plated through holes. The post drilling process would be to make sure how well you can prepare the hole wall for plating.

(Refer Slide Time: 09:33)



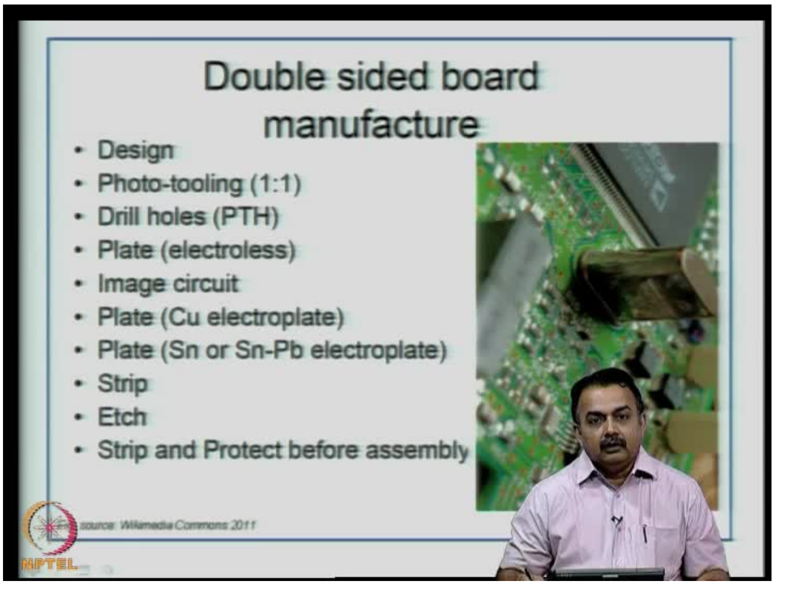
If you look at the double sided plated through hole printed wiring board structure, in the interlayer a connection can be established; one by riveting technology, which is very old. Basically, in the area where the hole is drilled you insert a rivet and then seal the structure, but you cannot repair it because you are introducing or adding more weight to the board. The next thing is filling with the solder material, so you have to pour molten solder and fill the hole. These kinds of technologies are not being practiced today.

The third one listed here is filling with the conducting polymer paste that is being practiced in some sectors. Once you drill the hole, you use a conductive paste; typically a paste which contains metallic powders, metallic particles like gold, silver or a combination of silver copper and so on. These are dispersed in epoxy medium, organic medium; they are printed by stencil printing or siring dispensing into the hole wall area and they are cured. Finally, they are polished; or basically the extra material is removed, the surface is treated in such a way that the surface is plainer after the process is over.

The technology that we are using today is called electroless copper, which was put forth in a paper by Shipley during 1960s, 1964 to 1967. You can see a number of papers and this became a revolutionary idea in the electronics industry, because you can deposit a very thin layer of copper by electroless further by electroplating; which means, to get a very reliable hole wall.

Modifications and improvements in these technologies until today have made this process essential. You cannot think of any other alternative today of making a hole wall conductive by such a reliable method. Therefore, you can see only a lot of research activities going on in this area to probably look at reliability issues, to improve the process and so on. So, electroless came about later followed by electroplating to increase the hole wall thickness of copper and today it is a very standard technology.

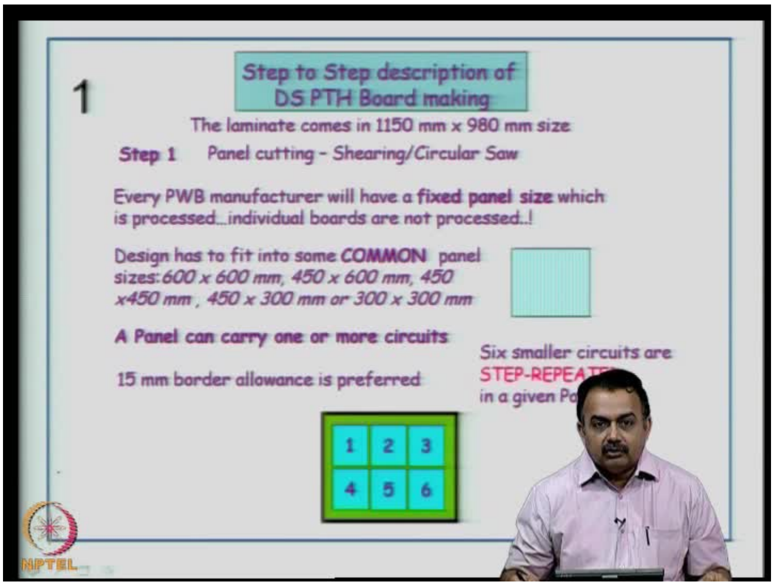
(Refer Slide Time: 12:36)



To recap double sided board manufacture process steps; the first thing is the design, then we talked about photo tooling. Then drilling the hole and then making the hole wall ready enough to carry the electroless plated copper, because you have to prepare the hole wall enough well so that there are no debris inside and the structure is perfectly cylindrical. It undergoes various cleaning processes. After that it goes through electroless copper and then you can image the circuit. Add copper on to those areas where you want specifically more copper thickness to be built so that is the circuit pattern that you want to add up.

You can protect the copper with tin or tin lead, then you can strip out the unwanted photoresist material and you can etch out the unwanted copper. Then do suitable protection by using solder mask; you can also use solder mask over bare copper which you will see shortly today. Protect the board with the conformal coting or otherwise send it for assembly.

(Refer Slide Time: 13:54)



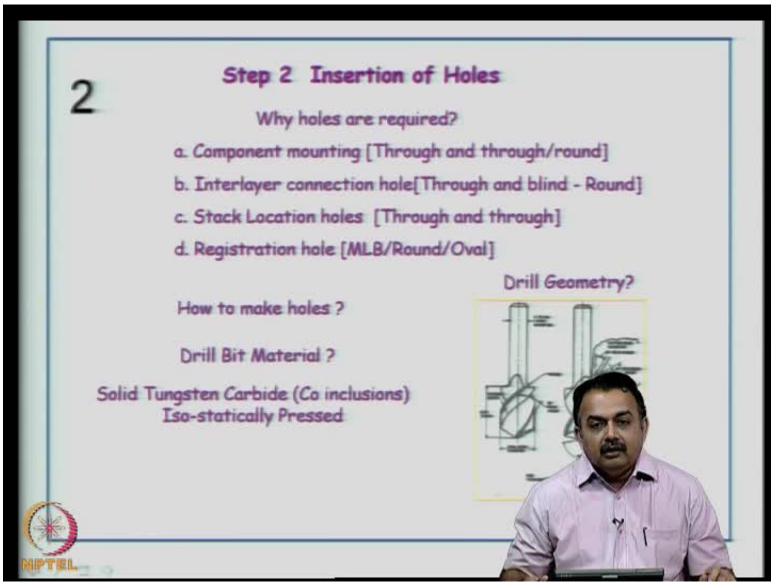
Now, I will take you through the process description in much detail; step by step for a plated though hole double sided board, which is again can be replicated for multilayer boards. The first step in the manufacturing is the panel cutting process followed by shearing using a circular saw. The laminate comes in these kinds of large panels 1150 by 980 mm panel size, you have to handle them carefully and you have to keep it flat so that there is no warpage. Storage becomes a key issue, no dust should be collected and they have to be protected, because as you know, we have seen during the earlier classes that any kind of extraneous materials has to be removed later.

Every PWB manufacturer will have a fixed panel size in this process house, accordingly the plating bath is fixed and all other chemical baths are fixed based on panel size - maximum panel size that a company can do; it can vary from manufacturer to manufacturer. Individual boards, if they are very small they have to be fitted in this panel. If this is the panel area, then you have to fit the individual board designs in to these areas, so that you can maximize the FR-4 laminate materials.

You can reduce the wastage and you need to give clearance. You can reduce the wastage based on very good float planning for your panel process. The common panel sizes are 600 by 600, 450 by 600, 450 by 450, 300 by 300, 300 by 450 and so on. Obviously, if the circuits are small it can carry more replicated designs on the single panel.

You have to spend some time as the manufacturer to fit in the entire panel circuits that you have on job. Typically a 15 mm boarder allowance is given as you can see here (Refer Slide Time: 16:20). Once this planning is ready, it will go through the entire sequence, right up to the etching process is over including probably the solder mask printing and legend printing. A lot of time has to be spent in the initial step.

(Refer Slide Time: 16:37)



Now drilling; the board now goes for drilling as we have seen why holes are required. It is required for component mounting, interlayer connection by through hole technology, stack location holes through and through during the entire process, and for registration. How do you make holes? You have to use a drilling machine, typically for prototyping you can use a manual drilling machine, but in a large volume production you have to automate it, so you have to use CNC drilling machines.

In some cases, companies invest a lot of money on having multiple CNC drilling machines, because as you will understand drilling can be a bottle neck. If you look at their entire process flow from drilling to the finishing processes, lot of time is spent in the CNC drilling because it is a sequential drilling. Therefore, you have to examine for quality at the end of each panel drilling, so lot of time will be spend because the subsequent processes are fairly fast, they are all parallel processes including imaging, etching, solder mask printing and so on. So, you have to use CNC drilling machines.

Today in the Indian context, you can buy a single spindle CNC machine for around 30 to 40 lacs depending on the make, but you can have multi spindle machines that can easily go up to about 1.5 crore; multi spindle machines. So, instead of having one spindle and do the job of panel drilling you can have in the same machine something like 3 to 4 spindles working simultaneously and sharing the job of drilling the entire panel details.

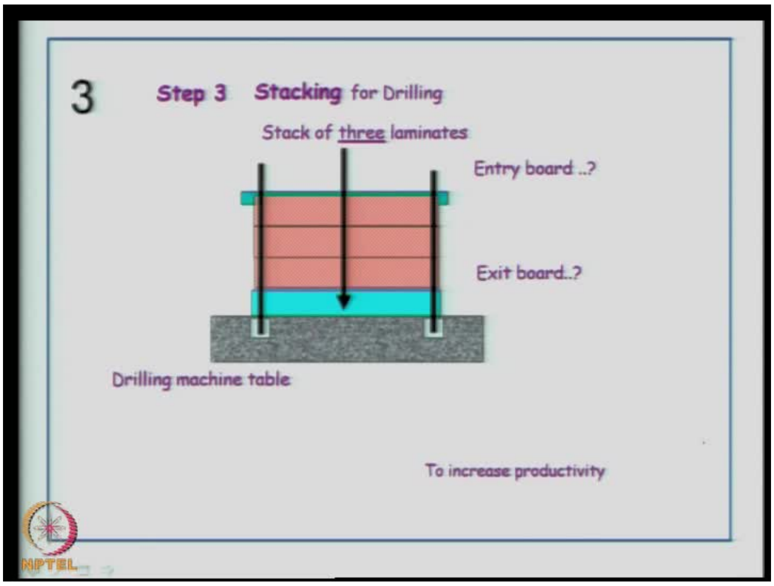
Now we are talking about a laminate which is very tuff; copper is soft; the dielectric material, the resin and the filler are fairly tuff. For example, if you use a FR-4 you have the glass material which is very tuff. The drill bit has to pears through all of these materials and come back. When you drill obviously there is going to be a lot of heat generated, the idea is use of good material that can withstand these kinds of changes. During drilling there is a heat produced, when it comes out it is cooled. Again, these are very fast process and lots of holes have to be drilled in a short time.

So, the drill bit material has to withstand these kinds of environment and the process. Therefore the most suggestive material is solid tungsten carbide with cobalt inclusions, it is a very tuff material and it is isostatically pressed when it is manufactured. In fact you can repair these kinds of drill bit materials for at least two or three usages; that means, if a drill bit material breaks you can re sharpen it and then reuse it.

It is expensive as the drill bit sizes goes smaller and smaller and the cost of drill bit increases. The drill bit material as you can see from this picture here, it has got a shank - turbo shank, you have to operate it at higher speed. As the drill bit size goes down and down you have to operate it at higher speeds. We are talking about maximum of 1,00,000 to 1,20,000 rpm, if you go down the drill bit sizes.

I would like to show the figure in this slide, here you can see there are various formats of drill bits available, but for CNC drilling machine, on such a tuff substrate you will use a turbo shank. The sharp edges you can see here, this is a tungsten carbide drill bit material and there will be a lot of length available for this to be mounted. You can re sharpen it for at least 1 or 2 times, so that you can use the material.

(Refer Slide Time: 21:28)



A requirement of CNC drilling machine is that in order to increase productivity you have to stack drill the laminates. Typically you will use three laminates stacks for CNC drilling machine. If you look at this picture there is 1, 2 and 3 laminates that have been stacked, they have been secured using a tooling hole. This is the bed of the CNC drilling machine (Refer Slide Time: 21:57).

You can see here there is an exit board and here is an entry board. This is the drill entry (Refer Slide Time: 22:10). Assume there is going to be a drill at this particular area. As I said before, because of the complexity of the substrate heat can be generated, so you use an entry board typically made out of aluminum. This aluminum acts as a heat sink, by the time the drill bit comes out, most of the heat is removed and the entire panel is not affected.

Otherwise, you will see that the heat affecting the hole wall structure, because there is an organic resin which can melt and it can be quickly cooled and solidify. Finally, you have to make sure that the hole walls are clean. In order to present a very good hole wall structure you use aluminum entry boards and also at the exit, because you do not want your drill bit to heat your garnet structure on the x-y table of the CNC drilling machine.

To avoid brakeage, you use an exit board typically made out of Bakelite laminate. You can adjust the depth of drilling so that you can prevent the misfortune of having your drill bit broken due to touching the x-y table. These are some subtleties that an operator… all though it can be completely programmed but, initial settings and procedures has to be followed. Now, we will look at the video highlight of a CNC drilling machine.

(Refer Slide Time: 23:43)

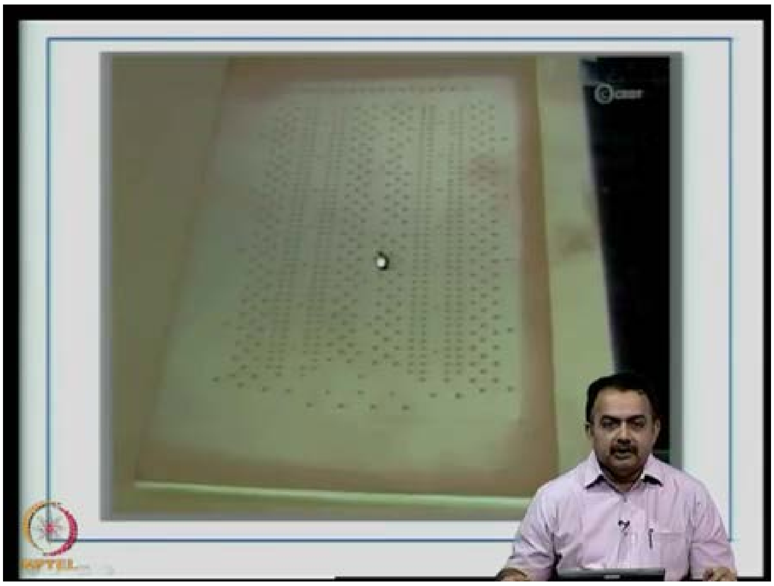
First thing we look at is a manual drilling machine, it is just to give you an idea how prototyping can be done using a simple manual drilling machine for printed wiring boards. You can see there is a view finder, here is the speed setter for the rpm of the drill bits, here the PCB is placed on the x-y table, there is a placeholder for the PCB which holds the PCB during the manual drill (Refer Slide Time: 24:07).

The operator is showing you how to insert a drill bit material into the drilling machine. Sufficient height is given during the fixing of the drill bit material, here again we are using tungsten carbide. As you can see here the drill bit is protruding, this height is very important because that defines the PCB board thickness for a complete through and through drilling.

It cannot be a through hole that is partially drilled through the depth of the laminate. You have the view finder; you can look at the pad that requires the drilling at this point of time. The pads are completely covered with copper; you are now centering the pad and then drilling at the center, because it is a manual drilling machine and you can have error.

Now, you can see a drill operation is been carried out. Manually the boards are moved to the next pad; the annular ring is also looked at and centered manually. The lever is pressed down to lift the drill bit up at the rpm set for this particular board and then the process is completed sequentially. Obviously it is very slow process, but typically it is well utilized for prototyping.

(Refer Slide Time: 26:03)



Now we will see a very simple single spindle CNC drilling machine, from this you can extend your imaginations to huge CNC drilling machines, which have multiple spindles. Here, you can see an emergency button is there for all CNC drilling machines and all high manufacturing machines. You have a CCTV; you have a camera which is now loaded to the drill area and close to the drill spindle. There is a huge x-y table; laminates are fixed at the base of the x-y table.

You can also do programming using your photo mask to create your own database. If you are going to do some kind of volume manufacturing of a particular design for which if CAD data is not available then you can manually program it, then store the data and offloaded your CNC drilling machine for subsequent drilling processes (Refer Slide Time: 27:10). The data is loaded from your database and then your CNC machine can actually move to the x-y coordinates designated for a particular board.

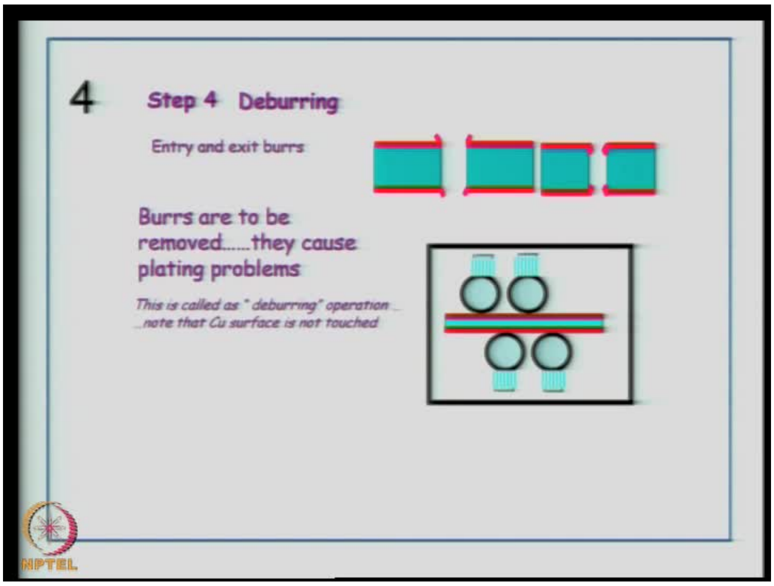
In CAD system you will generate XL files for CNC drilling machines; they will be just off loaded onto your CNC drilling machine. So, make sure that you have stacked the required number of PCB laminates. Here, what we are showing you is the spindle, where you are mounting the drill bit. There are certain set procedures especially with regarding to the drill bit material protruding out of the spindle.

Here, the technician is showing you what kind of distance or height is required for a stack of three laminates to be drilled through and through. In some cases, after a multilayer board is finished, you want to do controlled depth drilling; in that case you have to readjust the drill bit to the required depth that you want, because in those cases you do not want through and through drilling to be done.

Now the rpm has to be set based on the drill bit size and the thickness. As I said, as we go down the drill bit size more rpm is required, otherwise the drill bit will break. Once these parameters are set then the securing of the laminate is done firmly on the CNC x-y table and then the drilling process starts. In any CNC drilling machine you will see that the debris are coming out or dispersed on the surface of the laminate, but they will be vacuum pulled in to a collector close to the CNC drilling machine, so that the dust coming from the glass fabric and the organic resin does not spread throughout the machine and your joining areas.

In any volume manufacturing, the company can spend lot of time and quality control on CNC drilling; therefore, people have invested in having multiple CNC drilling machines to increase the productivity in the units. Once the drilling is completed, whatever is the best process you will see burrs, so you have to remove the burrs. Burr is a defect that is protruding out of the drill hole wall. The process used to remove the burrs is known as deburring.

(Refer Slide Time: 30:21)

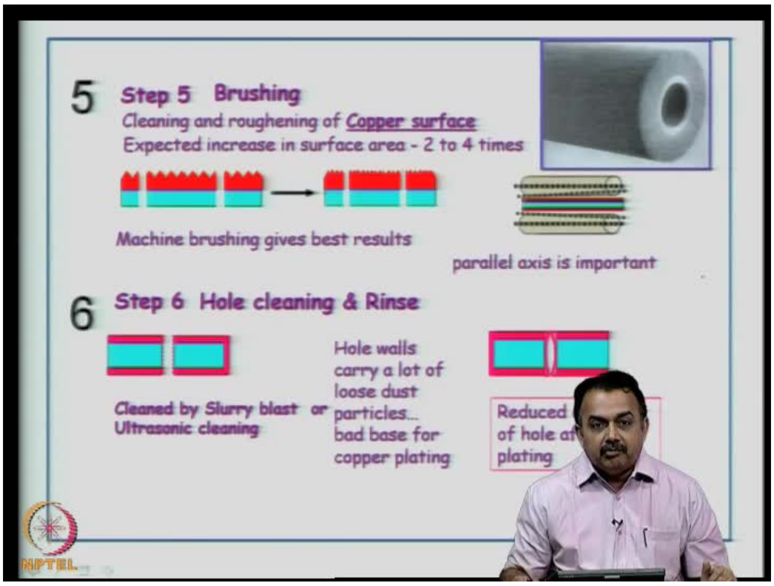


This is the fourth step, as you can see here in the slide; step four is deburring. You remove the entry and exit burrs that have been formed after the drilling process is over. If you do not remove this, these are going to act as high density areas where your plating can get a kind of un-uniformed, uneven thickness compared to the hole wall and you cannot insert your component through this, so you have to get a very clear hole wall.

Now, what are the things that you can do is, push the burr inside the hole wall. You can use brushing machines that can go through the surface of the panel and push the burrs inside the hole wall. Later, you can clean the hole wall inside by chemical and mechanical cleaning. One thing is sure that burrs have to be removed, they cause plating problems.

You can use conveyrised brushing machines like as you can see here, this is a conveyrised machine, you have two sets of brushes that operate and the board is moved across. It can be a dry process or it can be a wet process (Refer Slide Time: 31:40). Wet in the sense, you can have water sprayed on to the roller brushes and you can also have some kind of slurry like pumice slurry that can be use to roughen up the surface and also remove the burrs.

(Refer Slide Time: 32:08)

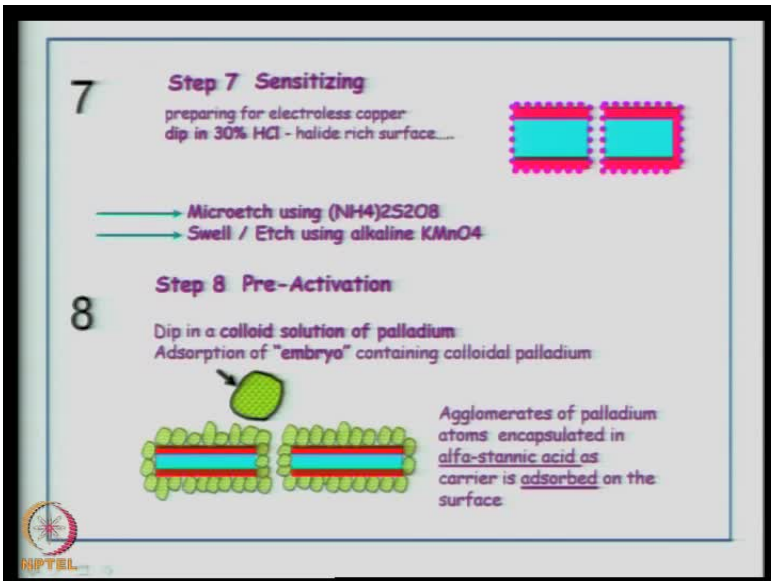


The fifth step is using brushing; here we are cleaning and roughening the copper surface and removing the copper to the extent of about 2 to 3 microns, in some cases much less. The idea is to provide a slight ruff surface so that your photoresist can adhere, we have seen this earlier, for this also you can use conveyorised machines. This is a photograph of a roller brush, which is used to increase the surface area on the copper's top and bottom surface in the case of a double sided board.

The next step is the hole has to be cleaned to take plating. So as I said, you can use ultrasonic cleaning, where you can use a mild acid or you can combine both mechanical brushing and chemical cleaning to make sure that the hole wall is perfectly cylindrical, it divide of all debris glass fiber, epoxy smear and so on. So, what you will see after drilling is an epoxy smear or a glass fiber has been removed.

Smear is basically coming from the melting of the resin during the drilling process and quick solidification. They can also carry a lot of loose dust particles that comes from the drilling; therefore, if you do not clean up as I said, you will get a reduced diameter of the hole after plating. Because for example, if you have drilled, let us say a 0.6 mm hole and if you have not cleaned these debris and fibers etc, after plating it will reduce the thickness to let us say 0.5 mm. Later, when you insert a component here, you will see that the component is not fitting into the through hole wall, so what you will do is you will try to force its entry into the through wall and as a result you will break the plated copper, which will result in an open.

(Refer Slide Time: 34:30)



This seventh step in the PTH sequence is sensitizing the surface. As I said, we are preparing the hole wall and the copper surface for electroless copper plating followed by electro plating. When you prepare for a electroless copper, dip in 30 percent HCL and make sure it is chloride rich surface. You are trying to make sure that surface is wet with chloride ions.

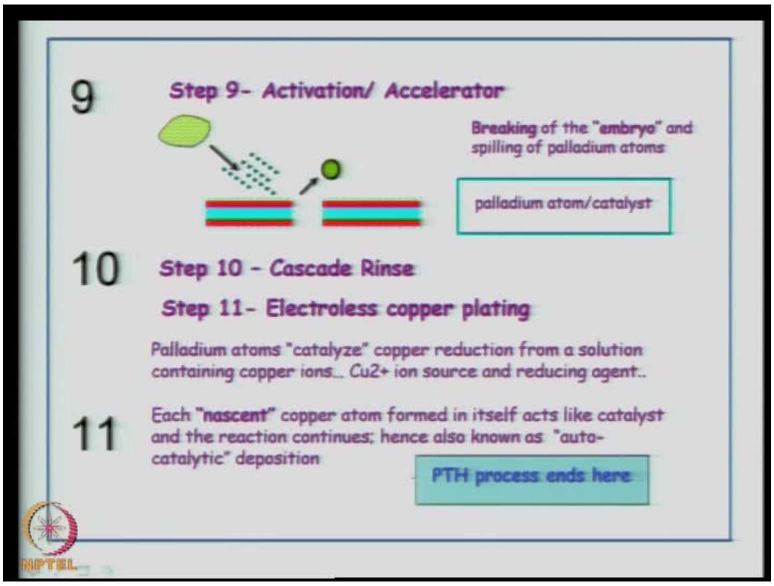
You do a micro etch using ammonium per sulfate, which is a micro etchant used to remove small amounts of copper. You can also use a swell edge chemistry using alkaline $KmNo_4$ to slightly swell up the glass surface inside the through hole wall, but these are the additional steps.

Now we will do what is known as pre activation; dip in a colloidal solution of palladium. Remember, we are introducing new catalyst, this is basically a catalyst material, since electroless is a process where we do not use external dc compared to an electrochemical cell, where we apply external dc and then there is a catholic and anodic reaction. Here, we are going to catalyze the surface with palladium. We use the absorption property of palladium to attract copper ions from copper solution and copper two plus solution, thereby making the surface conductive. This is the essential process of electroless copper plating.

Now, what we are doing is there is a colloid palladium, typically represented by this structure here, this has to be absorbed by the entire surface of copper as well as the through hole wall surfaces. So imagine a picture like this, what I have shown here is agglomerates of palladium; atoms encapsulated in alpha stannic acid as a carrier for the palladium, remember here it is adsorbed on the surface.

Can you see this picture here, agglomerates palladium atoms encapsulated in alpha stannic acid as a carrier, it is absorbed on the surface (Refer Slide Time: 37:13). This is a key process in the entire plated through hole sequence, it is not absorbed, it is adsorbed. This catalyst will provide a good base for electroless plating.

(Refer Slide Time: 37:27)



The next step is activation; these embryos now have to be broken so that the palladium atoms can be spilled over the entire area. In the earlier process, you can see there were embryos – embedded; now you have to spilt them individually using an activator material and then they are spread in a mono atomic layer fashion here, over the entire copper area as well as the hole walls. This is an atomic thickness, so palladium atom and catalyst is now very well distributed and absorbed.

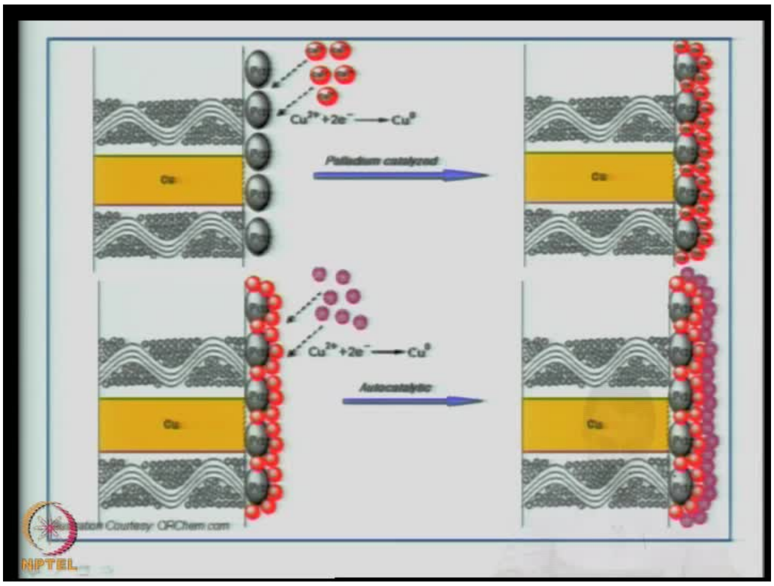
Now, you do a cascade rinse; does not use a forced spray rinse, it has to be a cascaded rinse. These palladium atoms will catalyze a copper reduction process from a solution containing copper two plus ions like a copper sulfate solution; this can be a copper sulfate solution. You have a reducing agent; a reducing agent will provide electrons, typically something like hydrogen or sodium hypophosphite and so on. These can be used to produce copper insitive which can be absorbed, which can be attracted by the palladium atoms. They sit on the surface of the copper, so you get a good physical bond, but very strong bond that cannot be easily disturbed by chemical reactions except for dissolution of copper.

So, you get what is known as a nascent copper atom that is formed during this electroless process; remember this process is very important because this is how you can make nonconductive surfaces to be conductive. In PCB application, it become very essential because your hole wall is nonconductive, your copper surface top and bottom is conductive. Now, you are going to conduct the hole walls and establish a connection between the top layer and the bottom layer.

Once this process starts, it is auto catalytic; you need not stop the reaction unless you remove the board from the solution. You can buildup the thickness fairly well to about 3 microns or even up to 5 microns, so there is initial layer formation of copper, the addition provided between the copper and a palladium is a key issue in determining the reliability of your though hole wall.

In some sense this electroless copper process which determines the life of the PCB itself, because if there is failure, it can happen at this electroless copper. The technologies available today are high ended; you can look at various technologies offered by various companies, which offer highly reliable electroless copper solutions. At this point, the PTH process ends and I have given minimum information here regarding the electroless copper. You can look at various books for more description on electroless copper process.

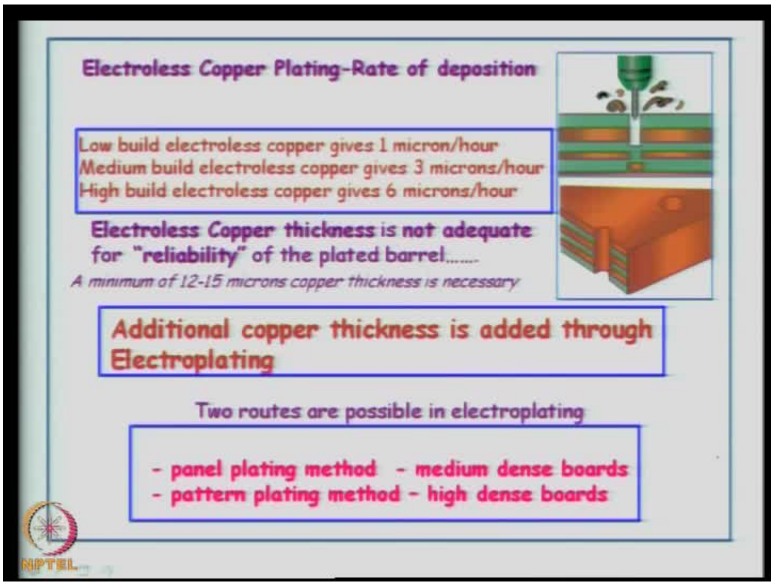
(Refer Slide Time: 41:06)



If you look at this diagram, it will give a more clarity. Here, you can see the cross section of copper and the dielectric; you can see the palladium is coated here from the colloidal palladium solution. Then, you have what is known as the copper solution in the presence of a reducing agent, you get copper 2 plus, plus 2 e giving copper zero. This copper will now be absorbed and attach itself to the palladium surface. Now you can see, here this bond is very crucial in determining the reliability of the board itself.

This is autocatalytic in the sense that you can immerse this electroless copper plated board in to electro copper. Now you can increase the thickness of the copper, so what we can see basically here is the red ball indicating that it is electroless copper and the lighter one here shows the thickness is improved by electro plating copper solution.

(Refer Slide Time: 42:31)

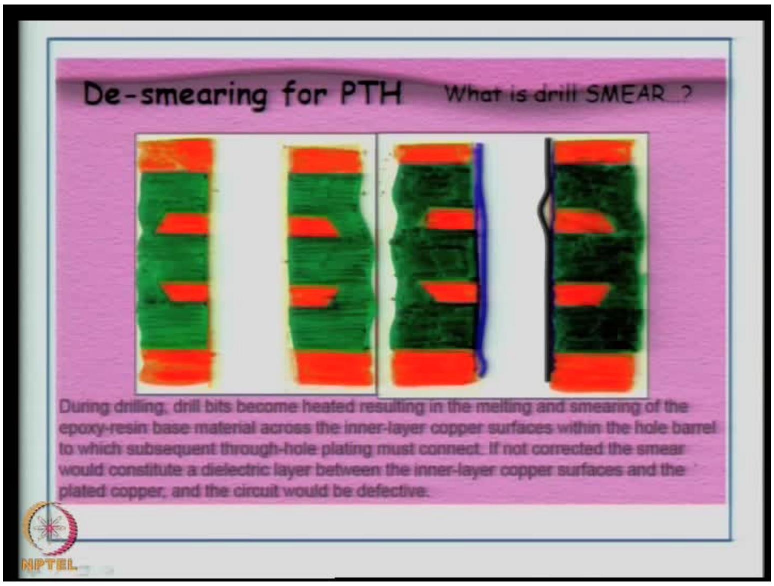


In electroless copper plating we look at the rate of deposition, what is known as a low build electroless copper, which will give you about 1 micron per hour, it is very slow plating process rate. Medium build gives you about 3 microns per hour rate of deposition; high build electroless copper gives you up to 5 to 6 microns of copper. So, it is a very slow process, you need to have good care control over process condition; for example, good agitation, good temperature maintenance, shelf life, concentration analysis of the solution and so on.

Electroless copper thickness is not adequate for reliability of the plated barrel. If you want something like 12 to 5 microns beaning built up in the addictive process technology that we talked about, then you have to go for additional plating by means of electro plating. Here, the entire board is now conductive, so it is now suited for electro plating process, you can add as much as copper as you want through the electro plating process.

So, the copper thickness final decision is based on the electrical design that you have specified including the hole wall thickness as well as the ground plane and the top signal layer thicknesses, and so on. Two routes are possible in electro plating; one is the panel plating method, other is the pattern plating method. The panel plating method is used for medium dense board buildup and the pattern plating is done for high density boards.

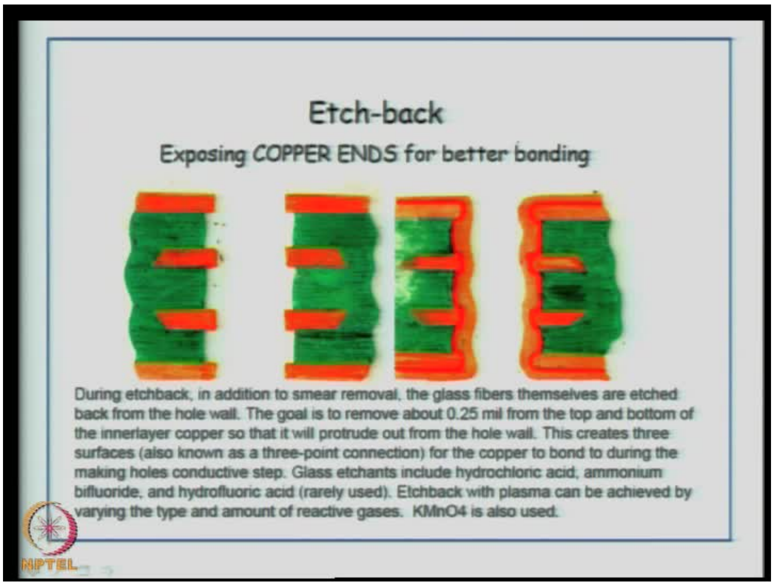
(Refer Slide Time: 44:16)



Now, if you look at a multilayer board, it has individually prepared inner layers, you have stacked all the six or eight layers in that you have now drilled a through hole. Any how the smear is going to be there; epoxy smear. You have to remove the epoxy smear, therefore you do a de smearing process. Smear as I said is because the drill bits becomes heated which results in the melting and smearing of the epoxy resin based on the material across the inner layer of copper surfaces within the hole wall.

If you do not remove it your through hole plating will not connect, therefore you will use certain chemicals to remove the smear. As I said, if this is the kind of structure you get after the platting process is over, then you cannot insert your component through this (Refer Slide Time: 45:20).

(Refer Slide Time: 45:25)



In another situation, where in a multilayer board, you want to make the hole walls perfect for a better reliability. Remember, you are having inner layers, these are stacked together and there can be misregistration during the final stack up. If you drill a hole wall in some cases, your inner layer copper can be protruding, in some cases it can be inside; it can also be a situation where it can be inside and it is not a protruding.

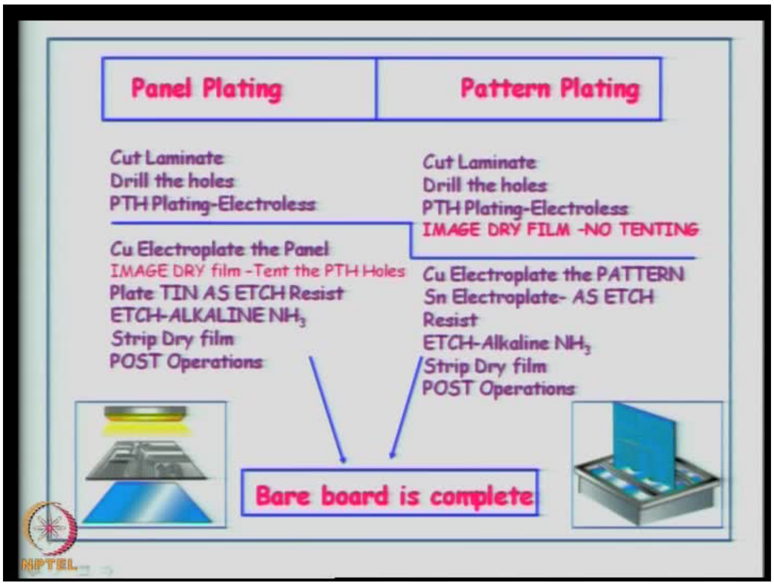
If you do plating in this area, then how do you establish perfect connection? Therefore, we do compulsorily what is known as etch back process. It will expose the copper ends of all the inner layers for better bonding. Once you see the copper ends of the inner layers you can do electroless copper followed by electro plating copper and you can get better bonding between the inner layers; how do you do that?

You do epoxy de smearing, which means you remove these kinds of epoxy smear material or even the base and the prepreg material. You remove to an extent that you see protruding area of all the inner layers of the copper including the top. Now, you do electroless plating as shown here in a dark red, then followed by electro plating. You see at these areas - the junctions, the three type of copper; that is the base copper, electroless copper and electro plating copper are well bonded.

In which case, you will get very reliable hole wall compare to areas where if you have the epoxy smear or the epoxy base itself, it is sort of over shadowing for the copper in inner layer areas, where you will not get better bonding. So, you get these kinds of pillars, it is called as a three-point connection; this will ensure copper two bond, copper two dielectric bond as well as three types of copper existing at this point to be reliable. It can undergo thermal cycling very efficiently without any breakage and it can with stand all mechanical failures vibration effects.

The etch back is done using plasma material; plasma etch back using CF_4 plus oxygen in plasma chamber or you can use $Kmno_4$- potassium permanganate to remove these kinds of this epoxy glass material to the required depth. This is the key process in the reliability of the multi-layer printer warring board.

(Refer Slide Time: 48:39)



Coming back, panel plating and pattern plating are two important routes to the board manufacturing. Panel as the name indicates the entire panel will be plated including the hole walls, where as in pattern plating only the selected circuit pattern areas will be plated, not all the areas will be the plated. So, when do you use it depends on your process step that is finalized and the thickness that is required for the final circuit pattern.

Let us look at panel plating first; you cut the laminate, drill the hole do a PTH plating including electroless. Since, it is panel plating you plate the entire panel. This is the important steps that distinguish between panel and plating. Now you do imaging of the dry film closed through holes, plate tin as etch resist, etch using alkaline ammonia and etching strip of the dry film and do post operations. Here you are plating the entire area, then imaging and then using tin as a etch resist. Then you are the removing the unwanted copper from the board and then you are stripping out the dry film.

At this point, you have to watch how it is done in the pattern plating process. In the pattern platting process you cut the laminate, drill the holes, electroless platting is done. Now, you do imaging of the dry film and no tenting is done; compare it to the previous process. You build up the copper by electro plating method on the pattern areas that you required and add more copper to the required thickness, it could be from 8 microns, you are going to build a let us say 22 microns.

The tin eletroplate act as the etch resist on the pattern areas only, then remove the photo resist or before that you etching using alkaline ammonia, so the non-pattern areas will be removed. You strip up the dry film and do post operation like solder mask, legend printing, etch amfering, conformal coating and so on. To understand when to use panel and pattern plating you need to look at your board design process steps, in some cases additive technology is required. Therefore, you will do pattern plating in a subtractive process, where you are very strong in your electroless copper process followed by buildup technology like pattern plating, then you can go ahead with pattern plating.

But, the key difference between the two is that what kind of copper thickness you can build in each of these. In the pattern plating process, obviously you have to remove more copper so your etch end should be well controlled. Where as in the other case, your plating should have more control in the pattern plating process to get the thickness. You have to calculate the area on the board, the current density and the final thickness followed by controlling the parameter in your plating solutions.

We will look at the process steps; the first thing in plated through hole sequence process what we are doing is the image transfer. This will get very good idea about the entire process step that I described in detail. As you can see here, we are doing dry film lamination, the board is ready, it is drilled and dry film laminated, both size have dry film laminated here, there is no dust and it is in well controlled condition (Refer Slide Time: 52:55). This is the photo mask, it has been used for this image process and alignment is done here manually as you can see. The hole wall have been drilled and then it is sent for UV exposition. The exposition machine; you see can it has a vacuum, it is drawn in inside, light is on, the board is exposed, the dry film is exposed, latent image is now built on the board.

Now, the board is sent for developing after the poly olefin layer is removed from the surface of the dry film laminated structure, you develop the image using 1 to 2 percent sodium carbonate solution. At the end of this conveyorised machine you will have a water rings to remove all the excess sodium carbonate solution, now you can see that the board is developed.

Now, if you want to do electroless copper this is very arsenical step, whether you go with pattern plating or panel plating you will have to do electroless copper. What are the process steps; watch this video carefully to understand electroless plating process in much detail (Refer Slide Time: 54:21).

You have the drilled board, it is now micro etched using ammonium persulfate solution for about 60 seconds and this will remove few microns of copper, it is mild etchant. Agitation is very important in large ply in large workshop, you have conveyorised automated system. What we are showing is a simple proto typing process and then it under goes water rinse for 30 seconds to remove all the micro agents that we have used.

This is followed by a acid dip, 10 percent sulfuric acid solution for about 60 second to neutralize all the agent, if any of that is present on the surface as well as in the though hole. This is followed by a water rinse, so you can see after any chemical dip we do water rinse, because these should be not transfer of one chemical from a tank to the adjacent tank, which can be a different chemical.

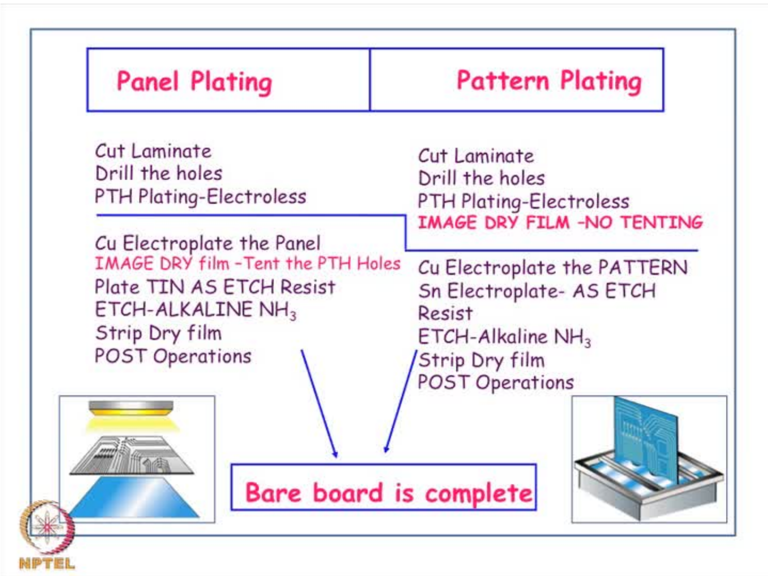
Now, you are activating it as you see in the illustration in the slides, using 10 percent hydrochloric acid solution, this is for about 60 seconds. Then, this is now introduced into the activator or the catalyst palladium chloride, this is fairly large time. Now, what we show here in simple lab type prototyping, but in practice you will see huge tanks in which this panel will be loaded with gigs and then introduce into the palladium chloride solutions.

So here, time is very important, agitation is very important, wetting of the surface is very important. Now, you follow it with a cascade rings typically, but what we showing here is a spray rings; very soft spray. Finally, after the activation is done we now introduce it into the sequence of chemical that is post activator. So here as you can see, this will be palladium atoms will split, the embryos will split, they will be disposed from the embryo forming a very uniform layer on the surface of copper as well as the through hole walls.

Remind you, we are activating surfaces as well the through hole walls, we cannot selectively do for the hole walls. Any process here will cover the hole walls as well as the surface that will ensure good binding and provide better connective from top copper surface to the bottom copper surface. After the water rings is done ,this is now introduce in to the electroless copper plating bath, what is actually shown here in a tray a bath. But again in practice, for large volume manufacturing self life is very important in prototyping units, because this has very low shelf life, only small quantities are used as and when required.

So, this contains copper sulphate solutions and a reducing agents, which will reduces the copper at the site of the palladium in to copper atoms. So you can see the color has changed, copper has been deposited on the surface as well as the hole wall surface. This is the very key process for all through hole boards, so this is now a complete electroless copper plated panel.

(Refer Slide Time: 58:25)



We will stop here at this point and we discuss panel plating, pattern plating, how the boards are etched get a completely top to bottom interconnection between two electrical layers in a system level printed warring board, in the next classes.