An Introduction to Electronics Systems Packaging

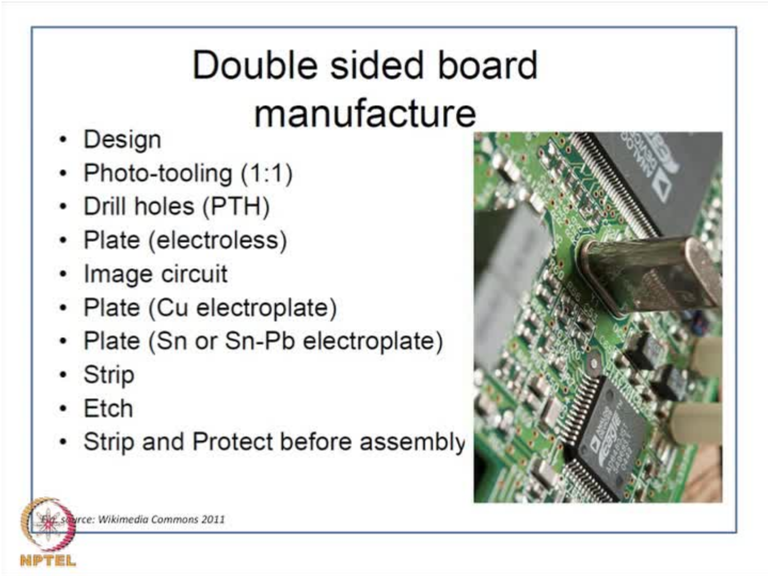
Prof. G. V. Mahesh
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 06
Lecture No. # 27
PWB etching
Resist stripping
Screen printing technology

We will continue with the Printed Wiring Board Technologies module. We have finished four hours in this particular chapter and if you have observed, what we have basically done is, taken you through the basic process steps of fabricating and understanding the technologies involved in the manufacture of a printed wiring board. It can be a simple single-sided board or a double-sided board plated through hole board or a multilayer board or a high density interconnects structure.

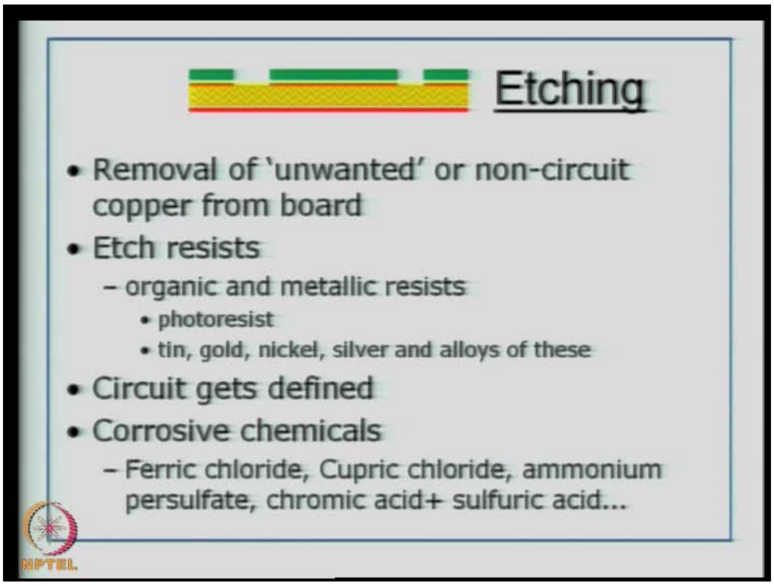
(Refer Slide Time: 00:27)

(Refer Slide Time: 00:41)



Basically, what we have seen now is, the individual processes have been described. So, that these basic processes recur at every layer manufacture. It is going to be repeated, if you are going to do a multilayer structure and therefore, it is very essential to know the basics of the key processes that we are now discussing.

(Refer Slide Time: 01:51)



Now in the previous stock, we have seen about the laminate selection. What kind of laminates you have to choose, the quality of the laminates and then, we spend a lot of time on imaging. Before that, we talked about cleaning and preparing the substrate for the manufacture or the imaging process. Then the imaging and image transfer including photolithography was discussed. Now we are going to see, how a circuit is established on the copper surface after the developing process is completed in the image transfer process. If you recollect, we have used dry film or a wet film photoresist; we have applied UV light and then we had cured the board. We have developed the board and now, the board is ready for the etching process.

Now, etching as the term indicates is the removal of unwanted or non-circuit copper areas from the board. You can see in this figure at the top, what you see here, the red lines are the copper areas; that means, these are a typical double-sided copper structure (Refer Slide Time: 02:42). On the top, imaging has been done using a dry film photoresist material and then it has been developed, opened and then you can see some areas of copper have been etched out and some copper areas are remaining. Typically, in this particular cross section, what you should understand is that the area behind the mask or the resist is the circuit areas the non-circuit areas have been removed. This is typically depicting the etching process in the printed wiring board sequence.

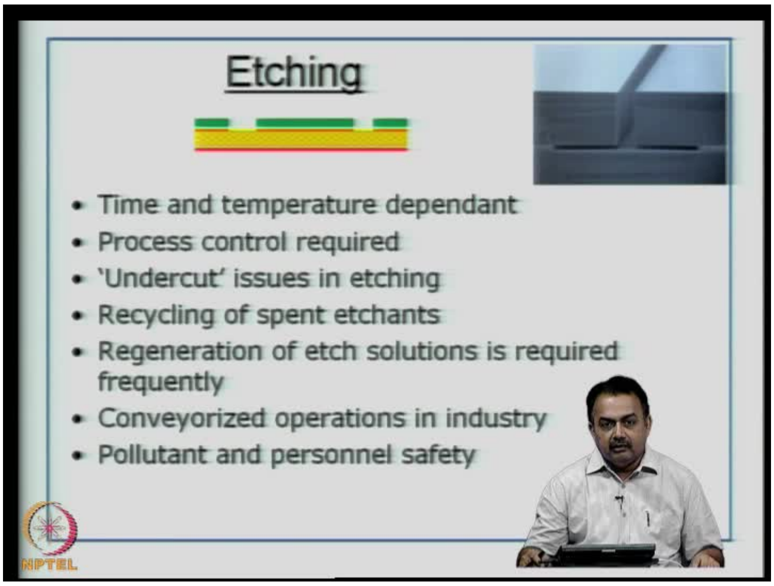
Now the term etch resist, you should understand because, you can have organic and metallic resist. As the term indicates, resist is the term for resisting the etching process. The resist material should not be eaten away or react with the etchant chemical. So that is why the term etch resist comes into usage. You can have an organic resist like a dry film material or a wet film material. Typically, a photoresist material can behave as a etch resist or you can have metallic resist. In the case of metallic resist, typical examples can be tin, which is plated on copper then you can have gold; gold is usually plated after an under coat of nickel is plated, so you can have nickel, gold plating which can act as a etch resist.

Then you can have other alloys typically, based on silver and so on, but in general practice, more than 90 percent of the etch resist - in the classification of metallic resist - is tin or nickel gold. The reason why we use nickel as an under coat of a gold plating is that, if you directly plate gold on to copper, after some time or during its operation and based on the environmental factors and so on and also due to the inherent crystal structure and properties of copper and gold; gold can dissolve into copper interstices and therefore, after sometime, you will see that the thickness of the gold that you have plated is gradually reduced.

The reason why we plate gold is, because of the good contact resistance, good conductivity and you need to plate only a very thin film of gold compared to larger thicknesses of tin, for example if you want to protect copper. So, nickel is known as a under coat or very basic essential requirement. If you want to plate gold on copper, the thickness of nickel can be around 3 to 5 microns and gold can be about a micron or less than 1 micron.

Now, after the etching process is over the circuit gets defined, you are removing all the unwanted copper. Now, the etchant chemicals are corrosive in nature, typical examples of etchant chemical, as you can see at the bottom of the slide are ferric chloride, cupric chloride, ammonium persulfate - which is a micro etchant, chromic acid and sulfuric acid combination and so on. For the through whole processes, we also use ammoniacal - alkaline ammonia solution. These of the basic resist etchants that you will see and also the etch resist given here typically organic and metallic.

(Refer Slide Time: 06:59)



Now, etching is the process which is dependent on time and temperature. If you take an etchant; let us say cupric chloride, you have to maintain suitable working temperatures for this cupric chloride. Then, if you are using machine based conveyorized etching, then there are other factors that come into the picture that are based on the equipment - the machine. Therefore, those controls have to be taken care of, maintenance is a crucial issue, if you want to get very high performance etching and especially, if you are doing fine lines of the order of 4 mills and so on. Your etching process has to be well controlled because, time and temperature are going to be closely monitored. If you want to etch away a few microns of copper perfectly with very clean edges, but etching normally, whatever be the best controls that you do, etching is not a perfect process, there will always be defects because, it is a very fast process and it is only the resist that protects your copper, nothing else. Process control is required therefore.

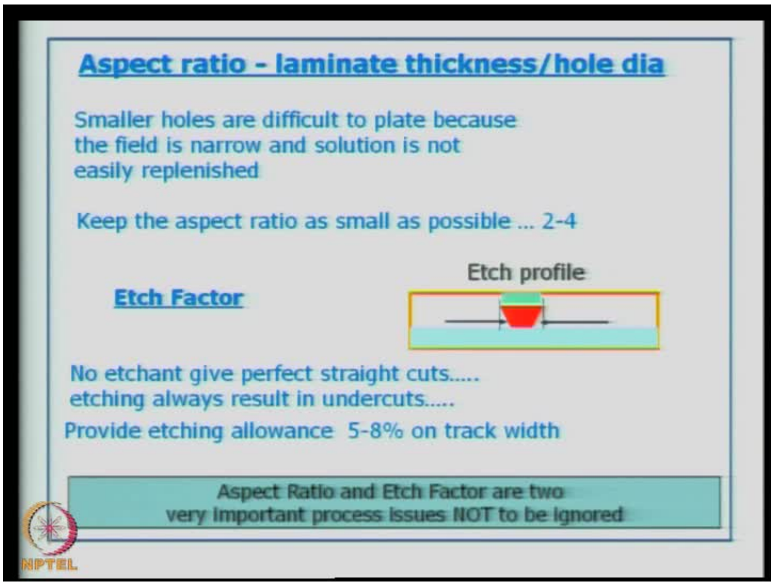
There is a term called undercut, so undercut issues in etching are very important. We are going to discuss, what undercut is. Basically, as the name indicates, if the etchant is going to sweep under the photoresist material here and remove copper in those areas which are supposed to be protected then, it is known as an undercut. Undercut can be fairly small in number or it can be large, if you are etching controls or not well monitored. So the idea is, you should have minimum undercut on the lines and the pads that you have defined.

Now, the environmental question of a recycling of spent etchants is not a normally well considered, but if you look at the current legislations of the usage of certain chemicals and also the local environmental regulatory body, they will issues the guidelines, how you have to recycle yours spent etchant? But, today it is somewhat straightforward; that means, if you buy an etchant from a company A; the company A is willing to take back your spent etchant at a fairly nominal price. So in that sense, you do not have to do the recycling. The manufacturers themselves are willing to take back the spent etchant; spent etchant means that you cannot regenerate that etchant any further.

The next point is, regeneration of etch solutions is required frequently, because when you makeup an etch solution, the concentration of the basic etchant ions or the etchant free radicals or whatever is generated based on the etchant, it has to be regenerated. Now, the rate of etching comes down slowly, as you progress using the same etchant. Therefore, you need to monitor regularly the concentration of the copper that goes into the solution, because the basic reaction of an etchant with the copper surface on the PCB is that the copper has to dissolve into the etchant.

Therefore, as the copper concentration increases in the etchant your reaction slows down; the rate of etching goes down. Therefore your etch time, which is actually a very key factor in large volume manufacturing, so the time will increase which will again indirectly show up in the undercut. Regeneration of etch solution is considered at most necessary in large volume manufacturing and typically, you are going to make an extent in the etchant life, but after sometime, you will understand that it has been classified as a spent etchant and you have to make up a new solution.

(Refer Slide Time: 12:11)



Typically, in large volume manufacturing, you will use conveyorized operations because etchant chemical is corrosive. You have to take care of the pollution factor and also the safety of the personal working in the lab or the industry. We have seen in our previous class, a very important factor that determines the design from manufacturing aspect; that is aspect ratio, which is defined as laminate thickness by whole dia. We have seen, how a designer needs to know whether a particular aspect ratio in a design can be manufacture or not, because smaller holes are difficult to plate. Therefore, the plating solution will not wet the small holes therefore, the reliability is a concerned. We have said that to keep the aspect ratio as small as possible in the range of 2 to 4.

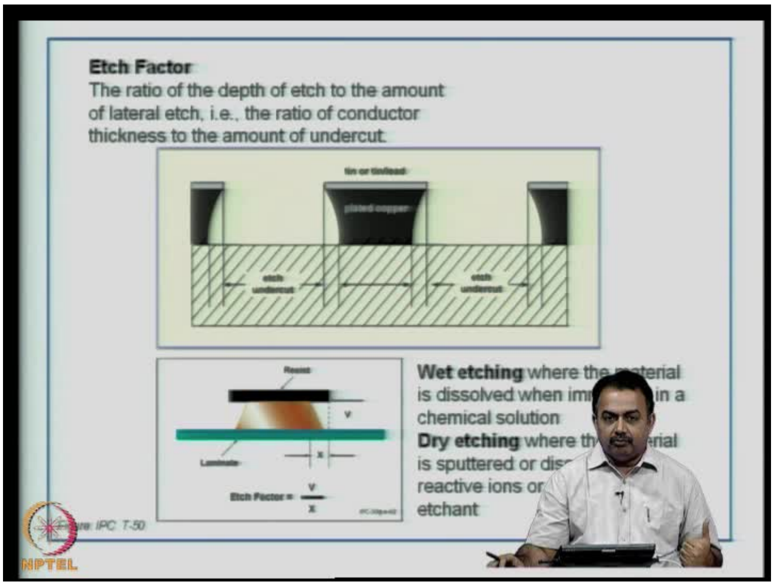
Similarly, another key parameter that you have to consider with more is concerned with the manufacturer - here not really with designer, is that etch factor is a key issue because no etchant gives perfect straight cuts. Etching always will have difficulties, you will always observe undercuts. If you carefully look into the microscope of the side wall of the copper track that you have etched, you will always see defects.

Therefore, you have to provide an etching allowance of 5 two 8 percent on the track width. That is where I say designers needs to understand, what an etch factor is or the manufacture can suggest to designer that if you have kept track at 6 mills, you can give an etching allowance of 5 to eight 8 on the track width because incase, the manufacture has observed that consistently in his manufacturing, he is getting an etch profile like this. What you see here in this picture the red one is the copper ,the top blue one is the mask or the resist. It can be an organic resist or it can be a metallic resist (Refer Slide Time: 14:22).

After the etching process is over normally, we would be delighted to see a perfect cut but, if you look into the microscope, you will see that there is a deviation. You can see that the area steepering down from the top surface of the copper and extending to the complete width of the copper that you have taken; you will see progressively copper has been eaten away. This distance that you see here, can play a major part in defining the electrical performance of that particular track. Therefore, if you get an etch profile like this; the extent of undercut, so the copper that is eaten away here, it is known as the undercut, you have to minimize the undercut.

If you minimize the undercut then, that directly reflect on the etch factor. Etch factor is a number that defines the quality of etching of a particular etchant. For example, material like cupric chloride will have an etch factor, alkaline ammonia will have an etch factor number; then ferric chloride will have a number and so on. So do not forget to discuss with the manufacturer about the aspect ratio and etch factor.

(Refer Slide Time: 15:56)



They are two very important processes not to be ignored. Let us try to define what an etch factor is? The ratios of depth of etch - so what you see here (Refer Slide Time: 59:24). This is the depth of etch to the amount of lateral etch; that you see, there is a ratio of conductor thickness to the amount of undercut. Basically, it gives an idea about the extent of undercut that has taken place despite having a metallic resist or an organic resist, which is a fairly resistant to the etchant that you have used.

If you look at this picture here (Refer Slide Time: 59:24). You can understand, this is a cross section of the base core structure or the FR-4. For example, this is the plated copper and you have protected on top with tin or tin lead; this is the undercut. You have to measure it using a microscopes, how do you look at etch factor? You are not going look at etch factor every day in manufacturing. If you prepare a new etchant or if you have working consistently with the same etchant, you can do it once in a while but, if you are moving to some new etchant then you have prepare a sample, test board to perform the etching and then do a cross section and then into the microscope, you go to the exact area where you have observed the undercut, measure the distant and then you know the thickness; you know the starting thickness of copper. So, the ratio of the conductor thickness to the amount of undercut will give you the etch factor. The etch factor has to be very high, which means the undercut is going to be very very low. So that is what you actually desire.

As a designer, you are aware of this problem and you can give a certain amount of allowance on the conductor thickness. You can increase the conductor thickness by 2 to 5 percent and then really not worry about the undercut. Now, we talking about wet etching where all the problem comes. Wet etching is where the material that is the copper; this case, the PCB is immersed into the chemical solution - that is the etchant - where it dissolves completely in the etchant material. Now, the etch factor is V by X as I mentioned here (Refer Slide Time: 18:32). If you want an equation for etch factor, you know the conductor thickness and you know the extent of undercut which is X, in this case. So, V by X will give you the etch factor. The etch factor is again different for different chemicals.

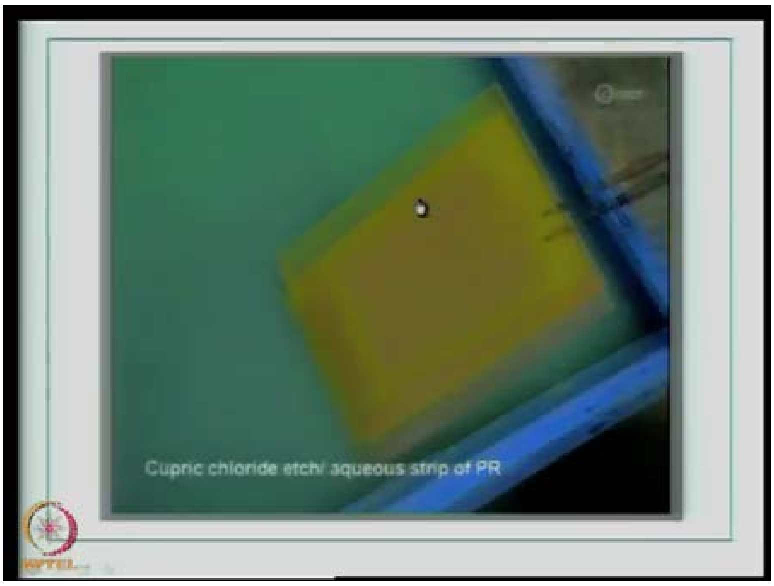
Coming to wet etching and now to dry etching, where the material will be sputtered or dissolved using reactive ions that are generated in an equipment typically, in the vapour phase etchant chamber. There are lots of differences between using a dry etchant, you can also use a dry plasma; that means, in a reactive ion chamber, you can create reactive ions which form plasma and then they react with a copper surface and remove the copper from the surface of the PCB, so that a dry etching process. There is going to be a lot of difference between a dry etching and wet etchings. Wet etching is typically very useful advantages for large area removal of copper, whereas dry etching, typically does not work well for large volume manufacturing and you can use a dry etching for prototyping.

Now, we are going to look at a video of the etching process and also final step of removal of photoresist which has behaved as a etch resist material in this case, the photoresist is behaving as a etch resist. This video as been shot at CEDT, you can see here (Refer Slide Time: 20:17). We have equipment which uses cupric chloride as the etchant and typically this is an aqueous based solution - cupric chloride dissolved in water and other chemicals added. Typically an acid cupric chloride; so use hydrochloric acid to the required percentages. So, basically this is an acid etchant. You also have alkaline etchants, which we will also discuss later.

After etching, the important process is to remove the photoresist which acted as an etch resist. You can see here, etchant is a very corrosive chemical even the equipment that you have used in this case, it is actually a PVC material, even then you can see that the leakages from the spray of etchant and the fumes that comes out from the conveyorized etching machine or the batch etching machine will create a lot of corrosion. So, the personal safety is very important and also we have to look at how you can save the equipment for extended life.

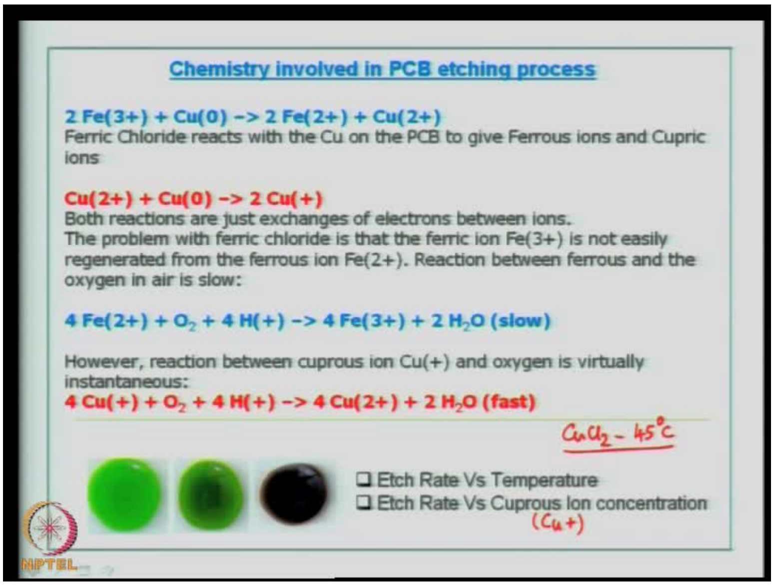
Now, typically in etchant equipment you start with the power supply; then you have the conveyorized equipment in this case, for example, you can see that this particular board is now imaged and developed. You can see here the areas that need to be etched are exposed; that is, the copper areas here are exposed to the etchant (Refer Slide Time: 22:00). The pink area that you see here are the protected areas pertaining to the circuitry. Now, the board enters into the conveyorized chamber and then typically the equipment will have spray nozzles that spray the etchant. Then finally, you will remove the protective photoresist and then you will get the completed circuit.

(Refer Slide Time: 22:51)



So, I will switch on this video once again for your attention. Now, this is the stripling chemical which is basically a hydroxide - potassium hydroxide which removes the photoresist material that completes the etching process.

(Refer Slide Time: 23:31)



Now, let us briefly look into the chemistry involved in PCB etching that is copper etching. If you take ion ferry chloride, Fe is present as Fe 3 plus and it reacts with the copper on the surface of the PCB and then you get a ferrous ion Fe 2 plus and Cu goes into cupric 2 plus. So, ferric chloride reacts with the copper at the PCB to give ferrous ions and cupric ions.

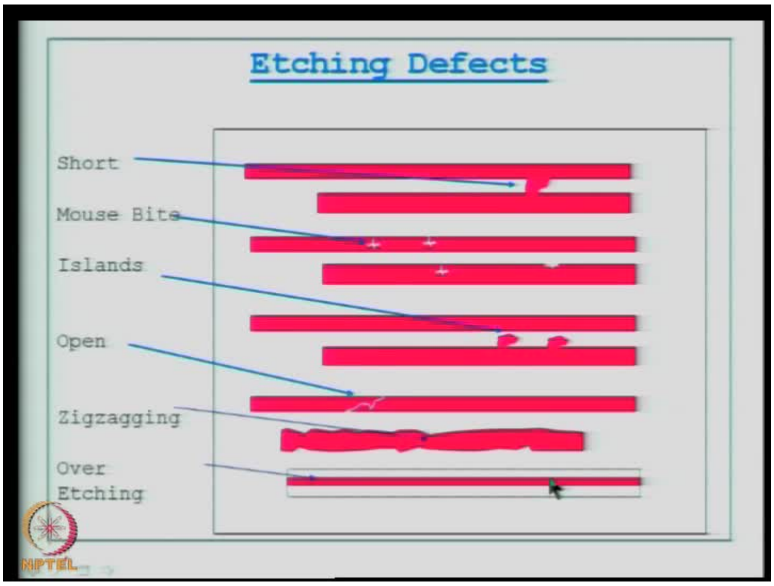
Now, this concentration will keep on increasing and you will see that the ferric ion concentration will keep on reducing that is why the rate of etching gradually decreases. So, regeneration is required in this typical case. So, what you do is, you add regenerate chemical which is basically a source of oxygen; let us say hydrogen peroxide, now to convert to the Fe 2 plus to Fe 3 plus. So that the reaction can progress further to react with copper on the PCB and then produce the required rate of etching. In the case of cupric chloride where copper is in 2 plus and it react with copper on the PCB copper 0. You get cuprous ion cu plus. Therefore here again, as I mentioned earlier, you have to regenerate Cu plus to Cu 2 plus to start the reaction or increase the self-life or extend the bath concentration of this particular cupric chloride. Now, the problem with ferric chloride in the first case is that Fe 3 plus is not easily regenerated from the ferrous ion.

Therefore, the reaction between ferrous and the oxygen air is very slow. Therefore, you have to again use hydrogen peroxide based supplier of oxygen, so that you can get Fe 3 plus and then water. This is a very slow reaction that is why ferric chloride is very difficult to regenerate; whereas, in the case of cupric chloride the reaction between Cu plus and oxygen is virtually instantaneous and you get back Cu 2 plus very fast. So, the regeneration of cupric chloride is well understood compared to the complex $Fecl_3$ that is ferric chloride. If you look at the recycling process for ferric chloride and cupric chloride in a spent etchant of ferric chloride, you have Fe2plus and Cu 2 plus. Therefore, the removal of Cu 2 plus which is a commodity that is worth removing from such as spent etching is going to be very complex; whereas, in this case, where you have only copper it is going to be recovered fairly straight forward.

Now at the bottom, what you see here is the progress observed in the cupric chloride etchant (Refer Slide Time: 26:53). For example, you see here at the extreme left; this is a very new solution of cupric chloride which is mild green in color. As the time progresses; as you start using the etchant material, you see it becomes dark and then finally, without regeneration, you see it becomes black in color, almost brownish black. This is the time when you have to regenerate the material. So at the bottom of the slide, I have put two very important issues that you need to always look at as a manufacture, etch rate verses temperature and etch rate verses cuprous ion concentration. What you mean by this? The etch rate will be dependent on the temperature of the etchant.

Typically, for cupric chloride the temperature normally is set at 45 degree centigrade so try to maintain this temperature to get a good rate of etching; whereas, in ferric chloride it can be 40 or 39 and so on. So, look at the manufacturer’s recommendation or look at some of the industry practices that produce best results for using cupric chloride or ferric chloride. The other important thing is Cu plus concentration so you need to monitor the cupric chloride for example, in the case of a cupric chloride etchant you need to monitor the copper ion concentration. Therefore, try to regenerate as early as possible into cupric Cu 2 plus, so that your etchant life is extended.

(Refer Slide Time: 28:57)

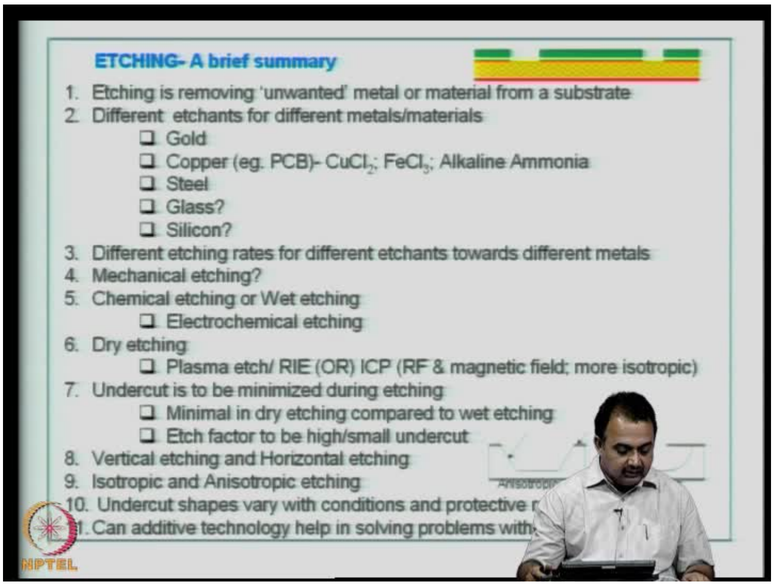


This slide will give you some simple illustration of what kind of etching defects that you can see normally in a finished PCB after etching. The first one indicates for example, here in this case, this is a typical electrical short; there are two lines and you can see a short that is a copper not properly etched. One of the reasons could be that strip of mask or photoresist is present there which you have not noticed. There is a problem with a developing process during the photoresist application or image transfer or it could be not undergone the right exposure in that particular area or there could have been a bubble which basically as effected the exposure and it has got translated finally into copper.

Then the next one is this kind of mouse bite kind of structures that you see where copper has been eaten away. This is again a process problem; then you have islands like this, fortunately it is not shorting but definitely not an accepted standard or an accepted quality after the etching process. I told you before that at every step whether it is image transfer or etching or developing; you have to have visual inspection.

Then you have zigzagging open areas, you can see typically, these can be due to debris - external debris sitting on your board. So one clear point that images from this kind of defect is that you have to work in a clean room atmosphere or a clean room and your materials have to be handled very carefully and stored in perfect clean conditions. Then you see a typical example here is, zigzagging again indicates a poor control on your etching process. Then the last one is over etching; that means, you are although it is somewhat over emphasized or exaggerated here, what it basically you see here is, the timing is not ok. Temperature could be above normal set temperature. Therefore, you can see the lines have been tin down compared to your design requirements.

(Refer Slide Time: 31:31)



So finally, a brief summary on the etching process: Etching is removing unwanted metal or material from a substrate. Different etching is used for different metals or materials. You can even etch gold, for example the etching for gold is aqua regia - a mixture of hydrochloric acid and nitric acid in a ratio of 1 is to 3. Then you can etch copper, typically in PCB using cupric chloride, ferric chloride, alkaline ammonia. You have different etchant for steel. You can even etch glass using hydrochloric acid for example.

Therefore, etchant can be employed for various materials and metals. Here, we are interested in copper; you have different etchants for silicon, in the semiconducting manufacturing that is typically potassium hydroxide is used. Different etching rates for different etchants towards different material; so constant agitation, continuous air supply, time, temperature are the key factor for good etching process.

Mechanical etching is basically a mechanical milling using a milling machine. Chemical etching or wet etching is always a question of debate but, people are more inclined to wet etching in printed circuit board industry, because the volumes are very large and they require at faster rate. Whereas, in the case of a dry etching; let us say using a dry plasma, you require expensive equipment and typically they are use for small area surfaces. You can also use electrochemical etching; in the case of chemical etching or wet etching, so that you can speed up the process. So, it is somewhat the reverse of a normal electrochemical cell or electrolysis process; whereas in this case, the PCB will be the anode instead of the cathode; instead of depositing copper on to the PCB, you are going to remove copper from the PCB.

So dry etching, as I mentioned, it can be a reactive ion etching using plasma or inductively coupled plasma using RF magnetic field. You can get more isotropic etching using dry etching process compared to a wet etching process, but the key issue here is undercut has to be minimized during etching. We have seen that it is minimal in dry etching compared to wet etching but, there are the ways to get a better quality control in wet etching.

Etch factor should be high between small undercut; vertical etching and horizontal etching are the two processes. In horizontal etching, the board is kept horizontal and it is moving through conveyorized equipment, whereas in a vertical etching a board is kept vertical and it is rotated as it is typically a batch process and you have sprays from both sides attacking the copper. Isotropic and anisotropic etching, you see in this particular figure here at the right; example of an anisotropic etching and isotropic etching (Refer Slide Time: 35:09).

So, isotropic etching is more has a good control in terms of the smaller undercut that you can observe; whereas, an anisotropic etching the rates are different at different areas compare to isotropic etching. Undercut shapes vary with conditions and protective resist, so you have to be a very clean observer if you want to look at the undercut. As you can see here in this figure (Refer Slide Time: 35:24).This is an isotropic etching and this is anisotropic etching. The rates are almost the same if use an isotropic etching, but that depends on the material also to be etched in terms of the structure of the material and the etchant that you will use for this.

This is anisotropic etching; on the top surface is the photoresist or the mask. You will observe more isotropic etching in the case of plasma etching, also in the case of a silicon KOH semiconductor etching, because there are different structures available and you can use suitable etchant for that. The undercut shapes in the case of PCB etching can vary with the condition and these kinds of resist that you are using. Now, the question is if etching is going to pose so much of problem, can additive technology help in solving problems with wet etching.

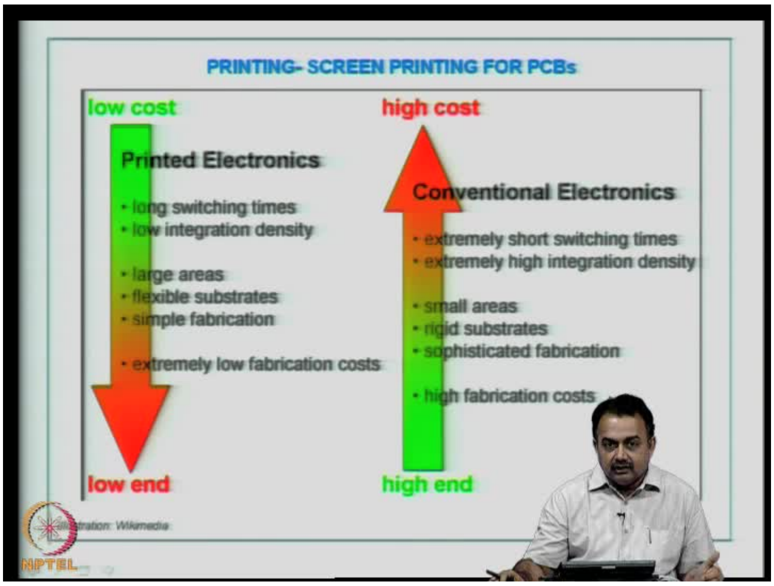
In the case of PCB instead of removing copper from the board or more thicknesses of copper from the board, you can add copper using electroplating to the exact requirement in terms of thicknesses and so on. That is why additive technology is becoming very popular, you can mask certain areas on the PCB that is a non-circuit areas with a metallic resist or a photoresist and then plate copper onto the pattern areas that you require to the required thicknesses.

(Refer Slide Time: 37:28)

Final step in the board manufacturing in general terms would be to strip the photoresist, because the photoresist which acted as a etch resist has done its job and now you are going to strip it. You can use aqueous solution - hydroxide based solution and then you can protect the copper with anti-tarnish treatment. It is a basically a simple dip of a board into a anti-tarnish chemical a thin layer will be protective, will be used as a protective agent for the copper surface and that can be removed later, if the board is going to be processed into the next stage.

Quality assurance a very key issue at every stage; visual examination at every step - in all the steps - that we have discussed so far is very important. You have to measure thicknesses, if you are doing plating for every batch, including thicknesses of the plating of the through hole walls. Microscopic examination for residues which can affect the board after the etching and before the etching. We have seen etch defects, so those can be avoided if you look at some microscopic examination, visual examination on the board. Typically, before etching every panel has to be examined carefully before it goes for etching, because at that point of time you can still rectify the board. Test coupons for every batch of boards will usually made for quality check, because a quality check will define how well your various chemicals performing at every stage and that can go for as a feed back to the manufacturing process. So, that completes the etching process.

(Refer Slide Time: 39:19)

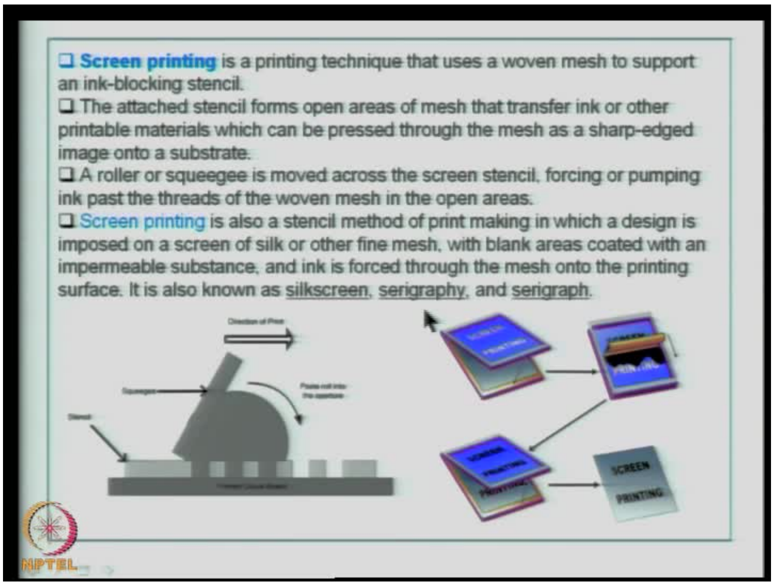


Now, we will go into the new topic called screen printing for printed circuit boards. Now screen printing is a very commercial activity that most of you would have seen screen printing of various inks on to various surfaces like your t-shirts or a simple printing like your invitation cards or simple printing inks on to walls or any surfaces to provide a legend. The same technology can be used for printing your tracks and pads on to the surface of the laminate structure, which contains a copper and the dielectric material and you clear the ink and then expose the unpatterned areas to copper for etching.

So, it is simple print and etches the process if you adopt screen printing for printed wiring boards. Obviously, you cannot do fine line circuitry with screen printing, this is normally used for commercial electronics applications; boards which have larger line widths, which obviously it cannot used for high density interconnects. Although screen printing can be used for solder mass printing and legend printing as a finishing process in the high density interconnect or any kind of PCB structure or any system level printed wiring board.

This picture shows you two types of situation: one is the printed electronics and other is the conventional electronics. The printed electronics are using printing - screen printing as a step in the manufacturer of PCBs it will be lower cost, because it involves a simple screen printing process. Long switching time, low integration density, large areas, flexible substrate can be used, simple fabrication process as I said, you simply print and then etch. Extremely low fabrication costs, but the important thing here is it is low end - in terms of performance - it is low end. If you go to high end performance, high end electronics, which is obviously high cost. Here, you have extremely short switching times, extremely high integration density small areas can be done, rigid substrate can be used, sophisticated fabrication technologies, new chemistries, new materials, high fabrication costs because, you are talking about fine line circuitry.

(Refer Slide Time: 42:27)

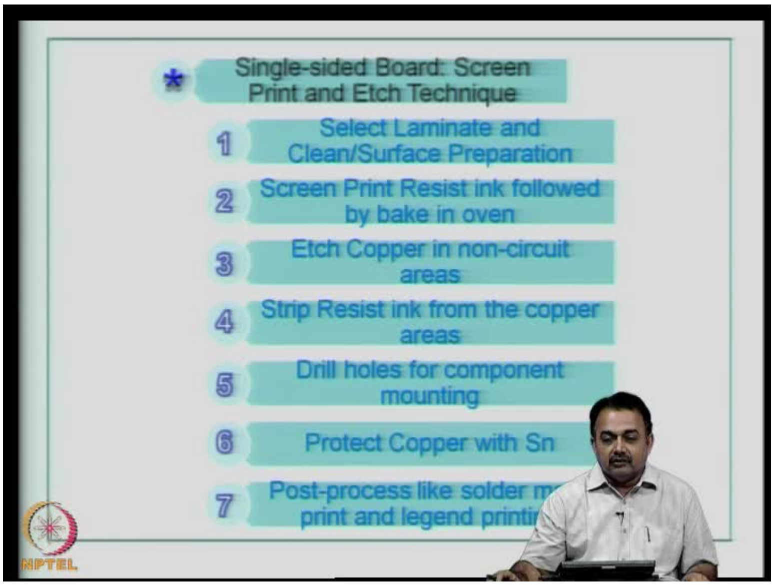


Screen printing is a technique that uses a kind of woven mesh a nylon or a polyester woven mesh to support an ink-blocking stencil. So basically, you are preparing a stencil using a screen plastic or a polymeric screen and using this stencil, you are going to do a printing process and this mesh is called a screen, therefore this process is known as a screen printing process. Obviously, the surface has to be very clean, the copper surface has to be clean or any surface for any form of printing. This stencil forms open areas in the mesh; that means, in the mesh, you are creating an image and that image will have open and block areas. The open areas in the PCB can be your non-circuit areas; the ink areas which go or flow through the mesh can pertain to your circuit, because the next step you are going to clear the ink therefore, they will act as a etch resist.

In this process, you are going to use a mesh and you are going to image it on to the substrate by printing, how do you do that? You use a roller or a squeegee and it is moved across this screen stencil. As you can see here, there is a printed circuit board before that I will explain to you what a screen is? You can see that there is frame of aluminum which contains the polymeric mesh. Now you create a stencil, here the word screen printing is depicted here (Refer Slide Time: 44:22). The areas where the letters are - are the open areas and the blue areas are closed. Now, if you run an ink from top to bottom using this squeegee here; the ink will flow into the open areas were screen and printing is written here; and it will print or the ink will flow through the mesh onto the base substrate.

Now, what you see here after the printing is done or after moving this squeegee, your lifting the stencil and you can see on the base substrate the words 'screen' and 'printing' have been printed. You can use this as an analogy for printed circuit board. Here, you have a printed circuit board; you have the stencil, which depicts the copper and the non-copper areas of your design which needs to be translated on to the printed circuit board. You take the ink and you squeegee it through this stencil so, this is known as the squeegee (Refer Slide Time: 45:24). It can be a silicon rubber, which will have a very sharp edge at 45 degrees to the stencil here, moving the ink through the stencil areas - open areas in the mesh. So, the ink will be transferred on to the base substrate which is the printed circuit board, you will move it only one direction, so assume this is the direction of print, you will get the print on to the PCB surface.

(Refer Slide Time: 46:22)



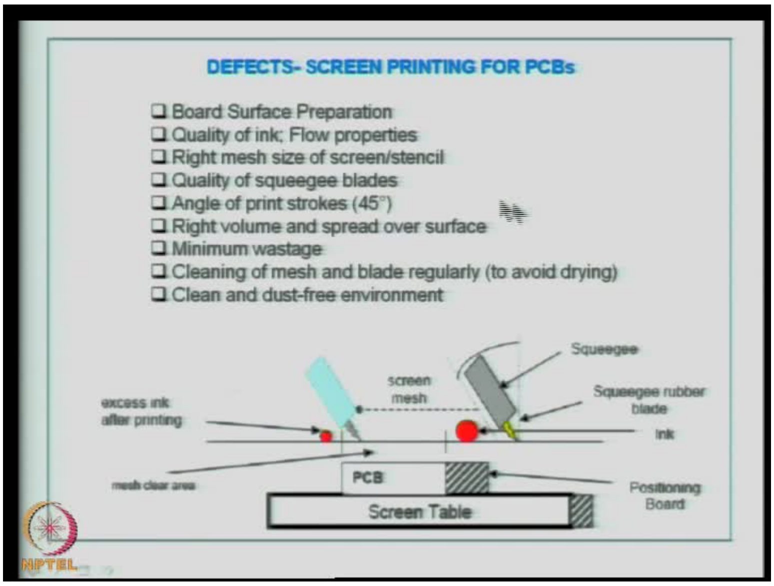
Screen printing is a stencil method of print making in which a design is imposed on a screen of silk or other fine mesh material with blank area coated with an impermeable substance and ink is forced through the mesh on to the surface or the substrate and also known as silkscreen serigraphy or serigraph. I will describe here the complete process steps for a screen print and etch technique. You select the laminate, clean the surface in the normal methods which you have described. Screen print the resist ink followed by baking in an oven.

So you use a suitable stencil, image it and then create a mask for that and then keep the stencil ready, print it. Now, after the baking is over, you introduce it into the copper etchant; this will etch copper in non-circuit areas. Now you can strip the resist ink from the copper areas which protected your circuitry. Now, you can drill the holes for component mounting. Protect copper with tin, because as you know copper react with environmental oxygen or moisture. Therefore, immediately you have to protect copper with tin or tin lead in the earlier case, but today, we are discouraging people to use lead, so tin lead. You can do plating is also here at this stage, but you can also use a simple tin coating - molten tin on top of the copper surface.

Post-process is like a solder mass, printing and legend printing can be done later to complete the single-sided board. So note here, I have emphasized that screen print and etch technique should be used for manufacture of PCBs only if it is a single-sided board.

Do not use this technique for double-sided board and other processes because is not going to work; we have to focus on additive technology there. This is an example of a subtractive process, where you simple or focusing your work on removal of copper.

(Refer Slide Time: 48:19)



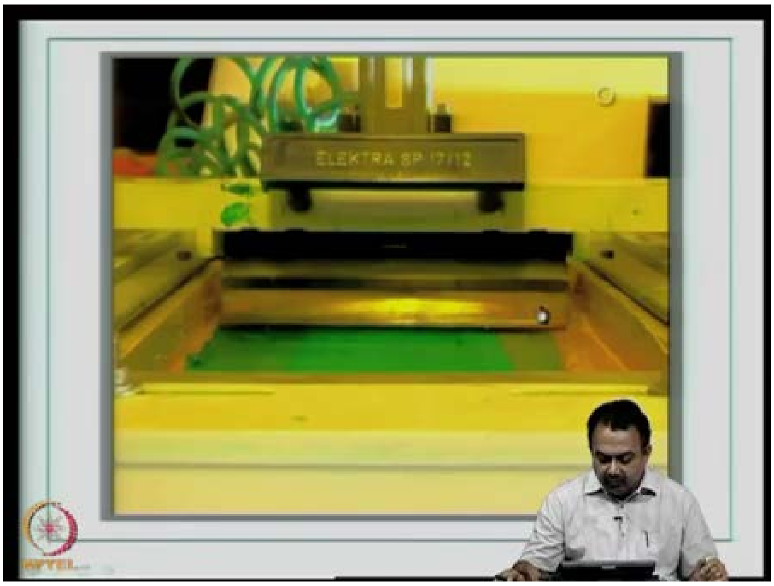
What are the defects that you will see in screen printing for printed circuit boards: board surface preparation is very important. Quality of ink, the flow properties it should not be too thick or too thin. Look at the viscosity; look at the type of structures that need to go on to the surface and the time taken for drying, what kind of solvent is used there? What temperature is need to be used for curing the material? Is there a shrinkage of the ink after curing? If you look at all of these things, you can fix the exact thickness of ink. Right mesh size of the screen of the stencil that you have used. Quality of the squeegee blades typically, this squeegee blades must have perfect edges. Otherwise, the pressure of transferring the ink to the surface of the PCB will not be perfect.

The angle should be at 45 degrees to the stencil that is holding your squeegee blade. Right volume and spread over the entire surface for a particular area that you are working with, ensure that you have minimum wastage of the ink, this will come with experience. Cleaning of mesh and blade regularly to avoid drying. Typically let us say, in volume production of single-sided PCBs, you will use some 500 boards and then you will have to remove mesh, remove the blade, clean them thoroughly and use them again or preferable use a new stencil or a new mesh. Clean and dust free environment as always very much required, so that these are not translated to etching defects that we have seen earlier.

This is again a figure showing you a screen table, a PCB is mounted which holds the PCB the cleared area between the stencil or the screen mesh and the PCB. So, introduce the PCB sorry introduce the stencil on to the surface of the PCB, load the ink and start the stencil the squeeze towards one direction. Using enough force to transfer the ink through the mesh openings and then lift the stencil, so that it detaches itself from the surface of the PCB. There should be no ink sticking between the surface of the stencil and the PCB surface; so that is where you have some experience on the quality of ink.

We will look at a video of a semi-automatic screen printing for printed wiring board applications. You will understand how important it is for the operator or the technician to understand this process? This is the mesh that we have seen and the blade - squeegee blade that you have seen and this is the semi-automatic printer, where you have the squeegee loaded or fitted on to the machine. Here, you can see that the ink in this case, it is a solder mask for example, it is loaded on to the screen printing machine.

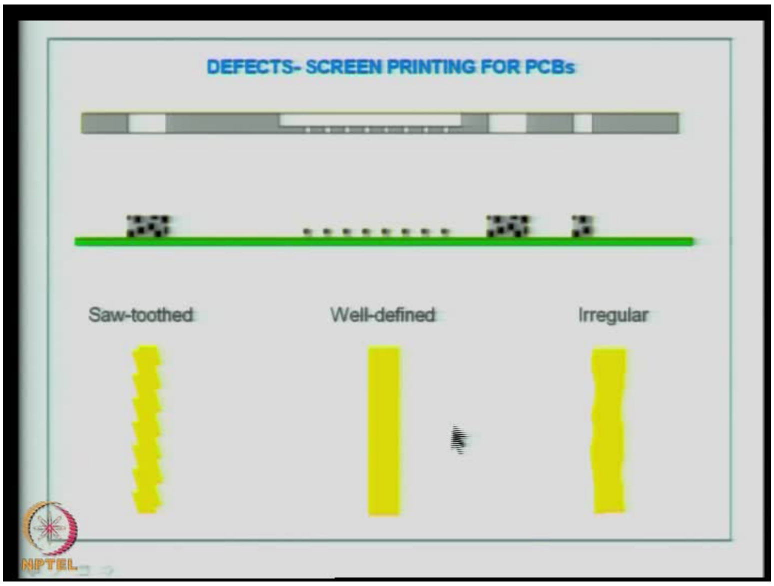
(Refer Slide Time: 52:02)



Now, basically you are trying to create a nice hold using vacuum between the stencil surface and the PCB. Here you can see, this is the squeegee blade mounted on to the screen printing machine. The ink is dread or pushed through the mesh openings on the stencil on to the surface of the PCB. If required, you can also do a second motion of the squeegee blade in the reverse direction to the original starting point and then finish the process. Now, switch off the vacuum and lift the holder - place holder of the stencil. Now, you can see at the bottom there is PCB surface which has been printed with the solder mask material.

This is a photo imaginable liquid solder mask that has been used in this particular case. Now this covers the entire area; this is the circuit area (Refer Slide Time: 59:24). We are going to use a mask for the solder mask after this material has been cured in the oven at let us say, 100 degree centigrade for about 10 to 15 minutes. Now, you can place a solder mask photo tool and then open up the unwanted areas and remove it by developing. So, the areas that is the pads and the tracks or the pads will be open, other than the pads all other areas will be covered with this solder mask material. So, these screen printings is very useful for print and etch process, solder mask process or legend printing for printed wiring boards.

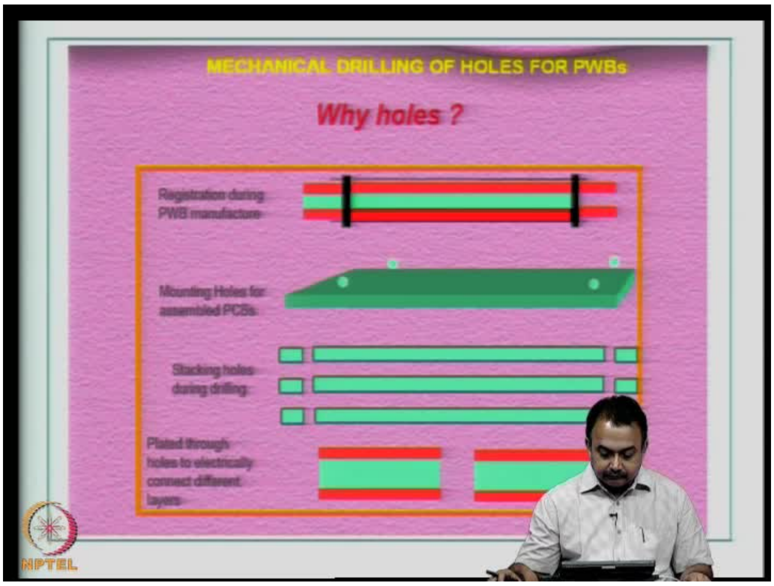
(Refer Slide Time: 53:44)



Now you can see here pictorially, defects that can be observed during screen printing process. This is the stencil; this is the PCB surface (Refer Slide Time: 54:24). You can see, if you have the controls right in your equipment and the process steps, you can get perfect dispensing of the material, it can be an ink, it can be a solder mask, it can be a white ink for legend printing or today screen printing is also used for solder paste dispensing during surface mount assembling. In this case, this picture typically depicts metal particles embedded in epoxy medium and what have basically shown here is a screen printing process for solder paste dispensing on to the lands that have been generated already on the printed wiring board.

At the bottom picture shows you typical defects. This is a well define structure of a line that has been screen printed; this is irregular, sometimes you get this, sometimes get perfect lines. Here what you do is, basically you have to check the mesh openings, you have to clean the mesh opening and the stencil regularly to avoid these kinds of defects and instead of getting these kinds of prints. This is again saw-toothed, you can see consistently you get these kinds of defects. So make sure that you get a well-defined line or an image after screen printed process.

(Refer Slide Time: 55:21)

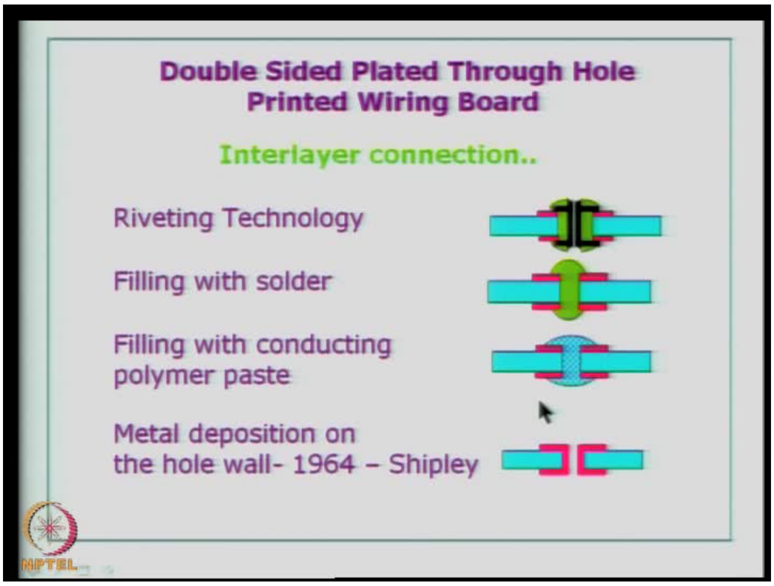


Now, we will look at another aspect of the PCB process. Remember, we have seen that mechanical drilling is a final step in single-sided board manufacture; whereas, mechanical drilling will be the first step in the double-sided board manufacture. Today, with all the advanced methods, we can also drill first for a single-sided board and then register your photoresist material and mask for single-sided PCB also. So mechanical drilling of holes for PCB, why do you require holes? You require holes for registration during manufacturing. If you are working on large volume production, you have to register you photo tool to the copper surface of the PCB at various times during let say a multi-layer PCB manufacture.

Therefore, you are not going to use manual registration, we will do an automated registration for that you require holes. Mounting holes for assembling the PCBs or in your PCB, you definitely have mounting holes which need not be plated but, these are going to be mounted to your chassis of your product. Stacking holes during drilling; in a CNC drilling machine, the panels have to be stacked before the drilling process. Therefore, you require stacking holes and holes are required basically to inter connect two copper layers and make them conductive by plating the whole walls of a mechanically drilled hole.

In our case we are more focused on how to make a whole wall conductive from a mechanically drilled hole of a structure like this; where you have copper on both sides, a dielectric in-between sandwiched, you are doing a mechanical drilled hole and you are going to copper plate this. We are going to look at, how well a mechanically drilled hole can be made suitable in all aspects for electro plating.

(Refer Slide Time: 57:44)



Double sided plated through hole printed wiring board. In this case, the inter layer inter connection is very important. The earliest technology was riveting the holes and making it conductive; in the olden days, people also tried filling it with solder, although it tripped – very immature for us today, but earlier days, this was simply an easy method of filling holes with molten solder and then providing some kind of an interconnection, but never reliable one.

Filling with conductive polymer paste; this is again well utilized today. If you are making hide density interconnects, where you are making thin inner layers, you can fill the hole with conductive paste and then polish off the top and bottom area and then get a very reliable conductive hole, but the more reliable one is metal deposition on the hole wall by electro plating methods. Electroless and electroplating methods - this was done or brought into the focus by Shipley in 1964 and we are still working on these technologies of electroless and electro plating combined, to get a very reliable hole. So, we will discuss more on these aspects in the next hour.