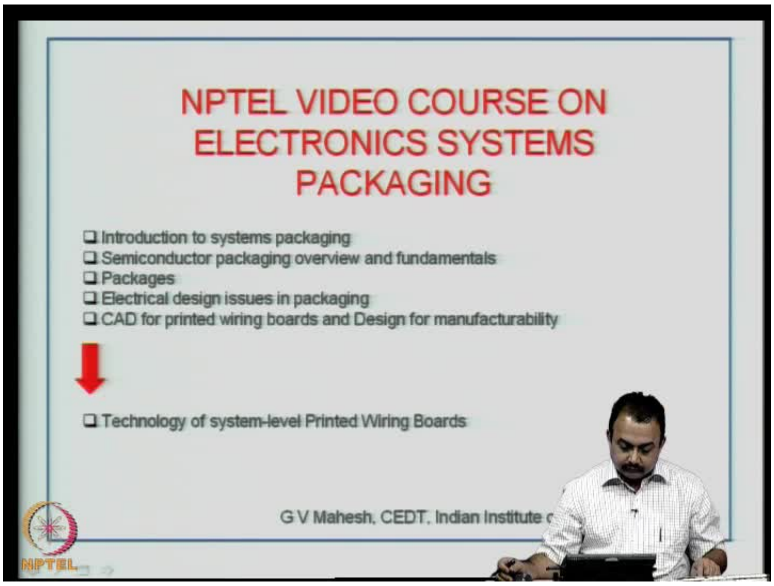
An Introduction to Electronics Systems Packaging
Prof. G.V.Mahesh
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 06
Lecture No. # 23
Review of CAD output files for PCB fabrication
Photoplotting and generation

(Refer Slide Time: 01:16)

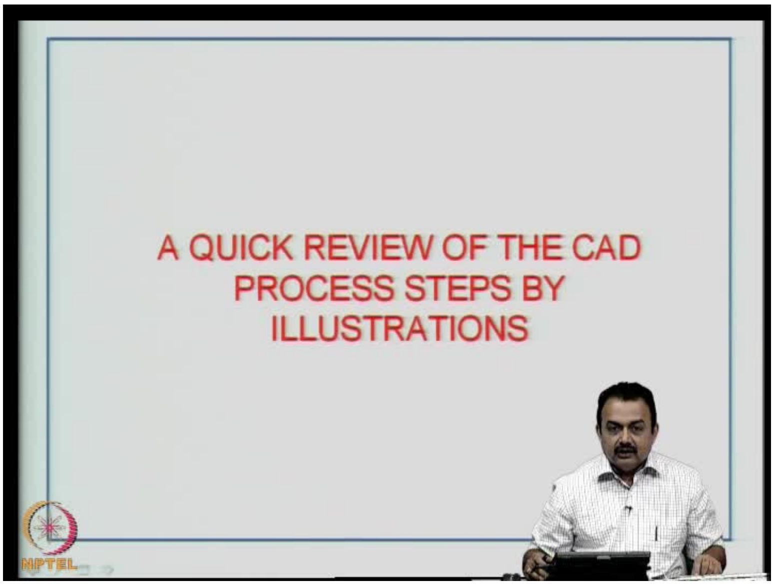


So we will continue with this video course on Electronics Systems Packaging as you can recollect what we have seen in the following chapters.

Introduction to systems packaging, semiconductor packaging overview and fundamentals, and all about packages overview: different types of packages, advanced packages and the salient features of various types of packages we have seen. We have seen electrical design issues in packaging and we have also seen computer rated design for Printed Wiring Boards. Typically, system level Printed Wiring Boards and concepts like design for manufacturability, design for reliability, design for testability and so on.

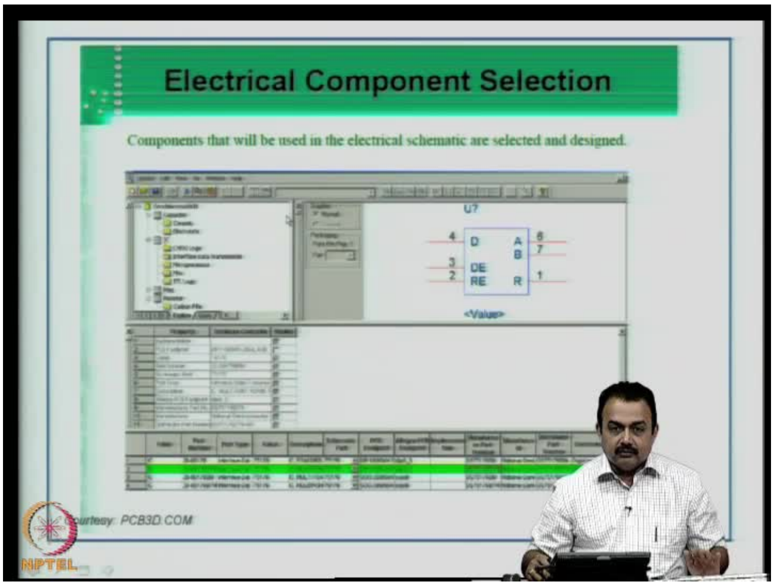
Now having completed these chapters, it is now time for us to go into the technology aspects of system level Printed Wiring Boards. I emphasis here that we talking about system level printing wiring boards and we are not talking about Printed Wiring Boards that are really low dense, but along the way we are going to see the manufacture and technological aspects of all types of single, double and multilayer boards including high density interconnect organic substrates that are used for advanced packages as well as for mounting these advanced packages, multichip modules etcetera on to a basic substrate typically, in organic substrate. When we talk about Printed Wiring Boards, we think about organic substrates there are of course, ceramic boards which can be used to mount packages and interconnect them.

(Refer Slide Time: 02:27)



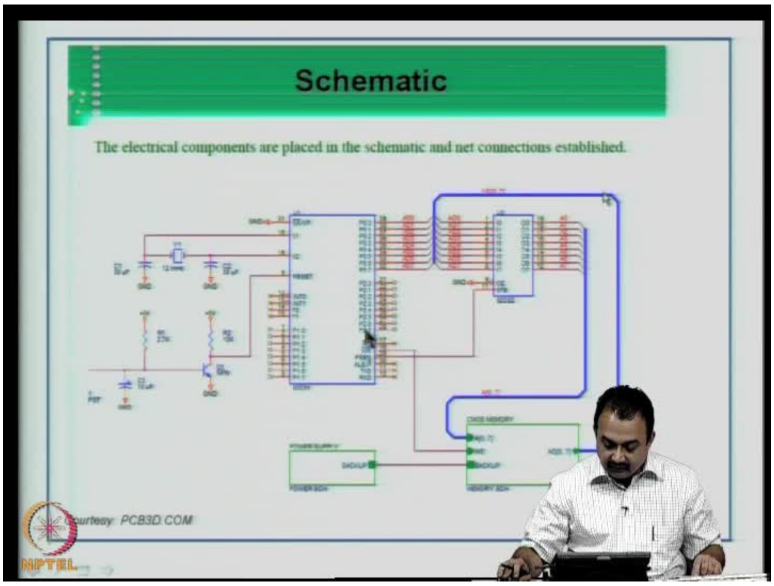
Before we go into the technology aspects of Printed Wiring Boards as a last review, we will look at the CAD process steps only by illustrations so this is something like a quick review not exactly a tutorial, but a quick recollect of what we have seen in the CAD. I will reduce the explanations here as much as possible, but we can have a look at the basic illustrations and this will explain to you the complete process steps that normally one encounters in any CAD program. So I hope this will help all of you into the confirmation of what exactly needs to be done, if you work with the CAD package: in terms of process steps, in terms of the deliverables, in terms of outputs that you can generate for a manufacturing process. So, that is the aim of this review illustration so we will start this review

(Refer Slide Time: 03:40)

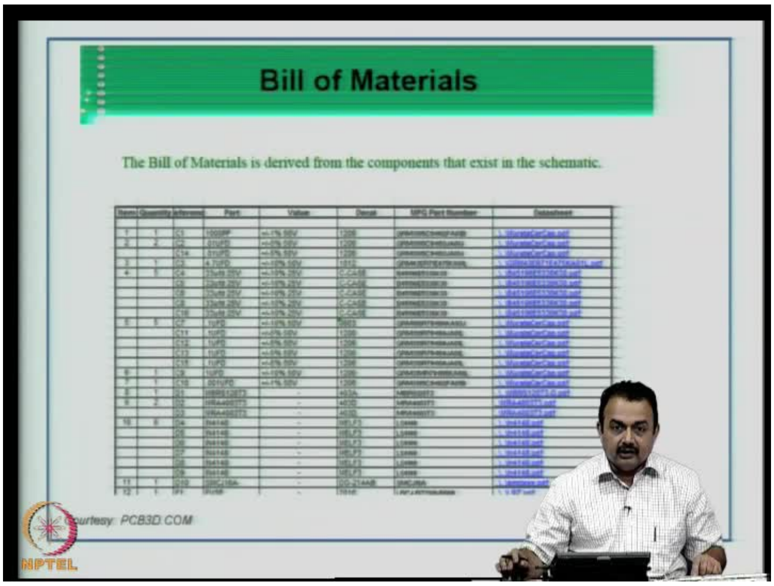


The first thing in any CAD process is the electrical component selection. Components that will be used in the electrical schematic are selected and designed. So as you can see here, you can create a symbol and you can create an equivalent foot print because this is going to be used in the electrical schematic page.

(Refer Slide Time: 04:06)

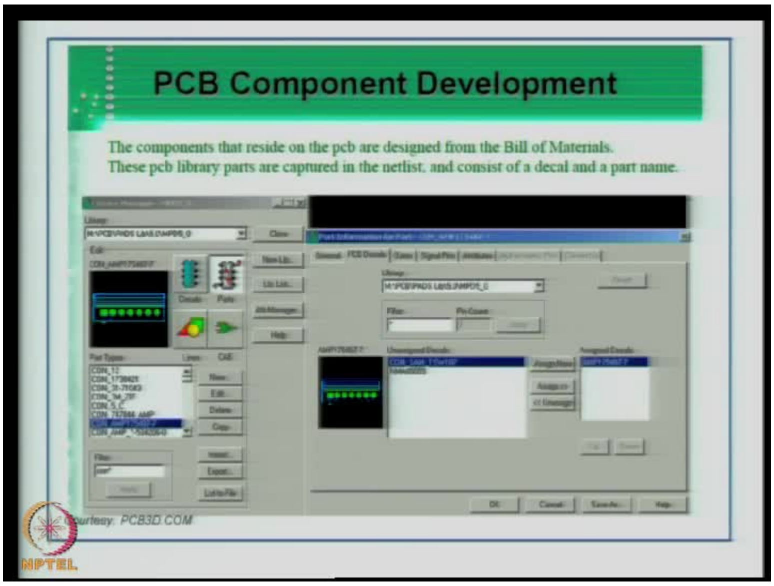


(Refer Slide Time: 04:43)

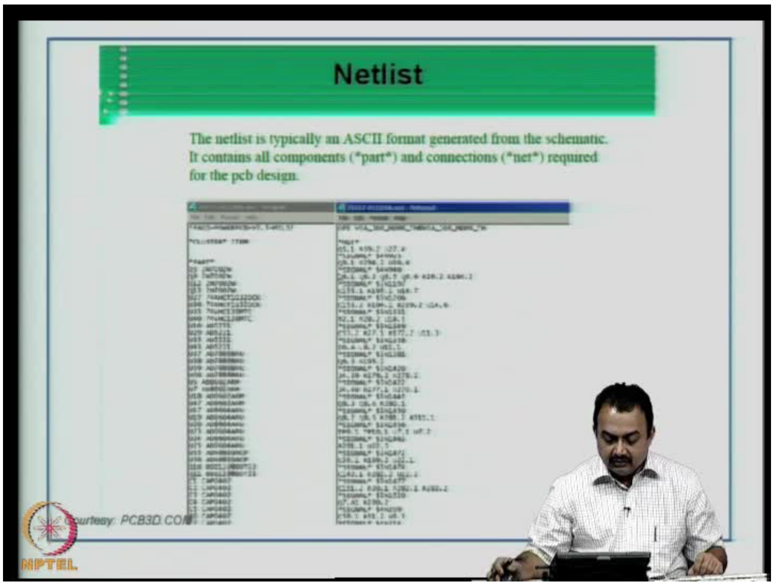


We have seen what is a schematic, the electrical components are placed in the schematic screen and the net connections are established. So you can have about hundred components placed, actives and passives and they are interconnected. So that will complete the schematic part of the CAD program. As you can see IEEE symbols are used, various passive devices are used and then they are interconnected. Then the bill of materials is automatically generated from the library, because the library will contained details of the components that exist in the schematic in terms of: what is the part number, what is the reference designation and what is the format or form size, then the part number and the company from where it has been purchased and we can also link it to the data sheet of that particular device for instructions on mechanical foot prints and so on. So, bill of materials is a very important thing in the documentation part of a CAD work.

(Refer Slide Time: 05:32)



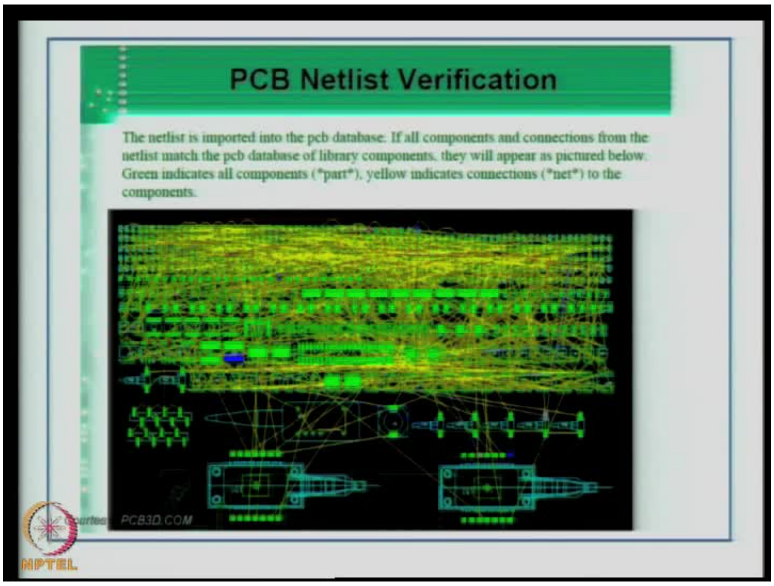
(Refer Slide Time: 06:10)



PCB component development: the components that reside on the PCB are designed from the bill of materials. These PCB library parts are captured in the netlist. As you know from the schematic you are generating a netlist and this netlist is used with all the information about the component foot print and so on and is used in the layout as well as the rooting stage. So, this figure that you see here includes the component development aspect, foot print development aspect and that is embedded in the netlist. So the netlist will look like this and we have seen in the earlier class what typically a netlist will do. Netlist is very important if you want to export your work to some other CAD program.

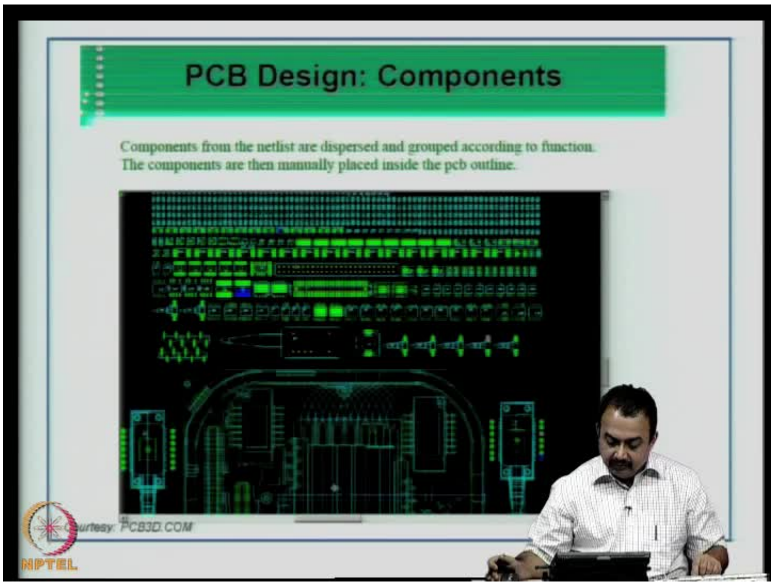
So, netlist is typically an ASCII format file generated from the schematic and it contains all component and connection details, that is the part and the net information required for the PCB design

(Refer Slide Time: 06:39)



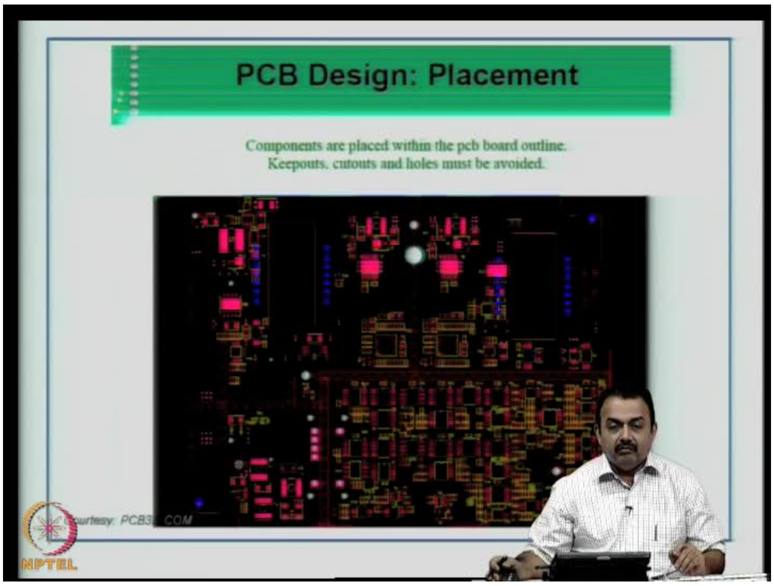
Then you do a netlist verification. So, as you can see in this figure there is, what is known as the rats nest. There the components are dumped on the screen in the layout module of your package with the connections. The yellow lines that you see here are the netlist connections and so this is typically a medium dense or high dense package. I mean design that you see here: you can see various parts and the crisscross connections between the components.

(Refer Slide Time: 07:18)

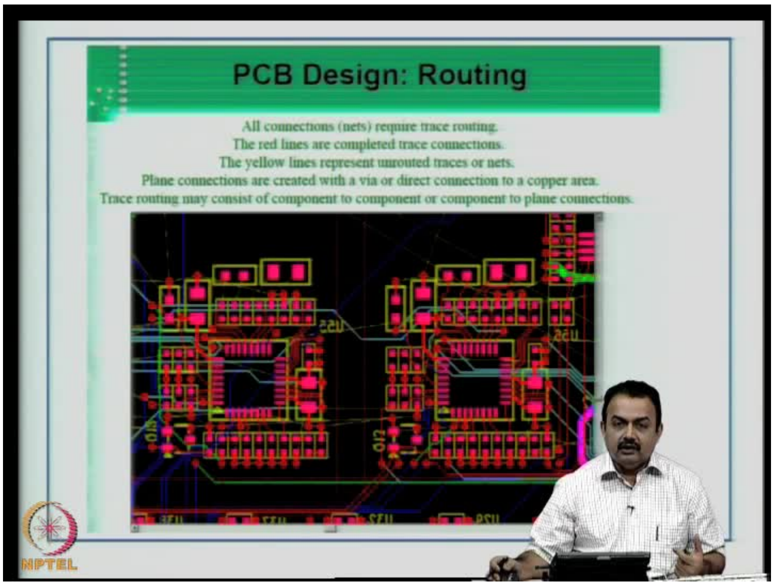


So this is the output of a netlist. Now you can see that you can switch off the net in your software package and look at the total number of components that have been used, that means the foot print details that have been used in your design. So, components from the netlist are dispersed and grouped here. As I mention before, you have to use your skills to manually place the components inside the PCB outline at least 10 to 20 percent of them and then probably you can going for an assisted placement.

(Refer Slide Time: 07:54)

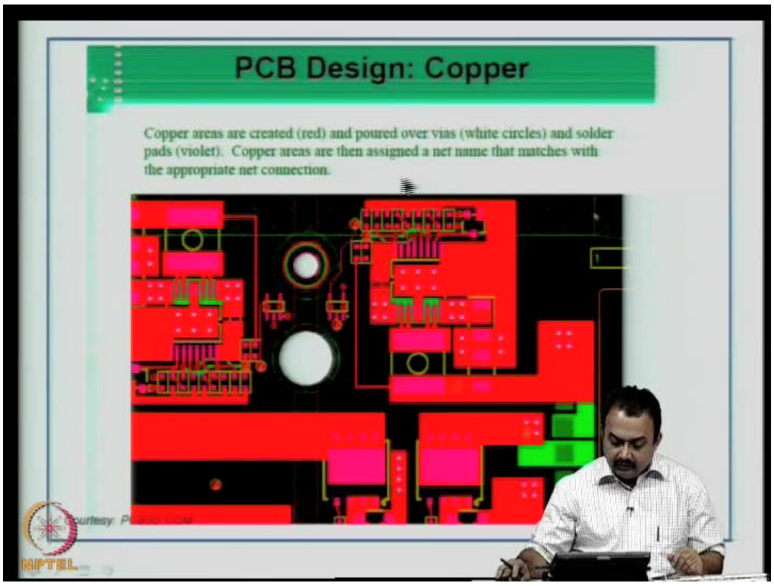


(Refer Slide Time: 08:36)



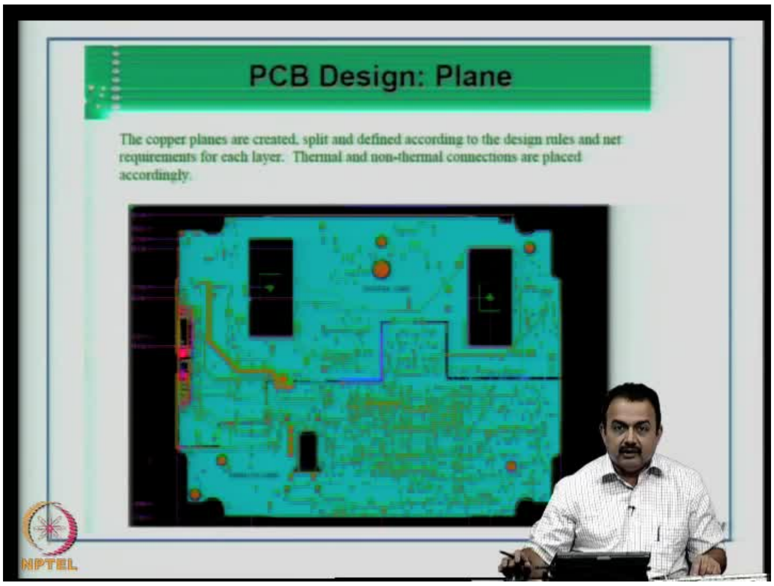
Now components are placed within the PCB board outline. So, the board outline definition is a key factor in deciding the product size. Keepouts, cutouts and holes, mechanical holes other than the electrical interconnect holes that you may use for your electrical circuit needs to be created. So normally you will have a mounting hole, sometimes you may have to create some kind of an opening for placing a mechanical component in the board outline itself, but these can be avoided by a better design. Then we go to the routing program, utilize the routing efficiency of your CAD program to interconnect the components based on the netlist information that we have generated. All connections are nets required to trace routing. The red lines, as you can see here in this particular figure are the completed trace connections. The yellow lines represent unrouted or incomplete traces or nets. Plane connections are created with a via or direct connections to a Copper area that you have generated for ground and so on. Trace routing may consist of component to component or component to plane connections. So unrouted nets can be done manually or you can do a better placement procedure or you can also modify the design rules and then complete the routing for your design.

(Refer Slide Time: 09:38)



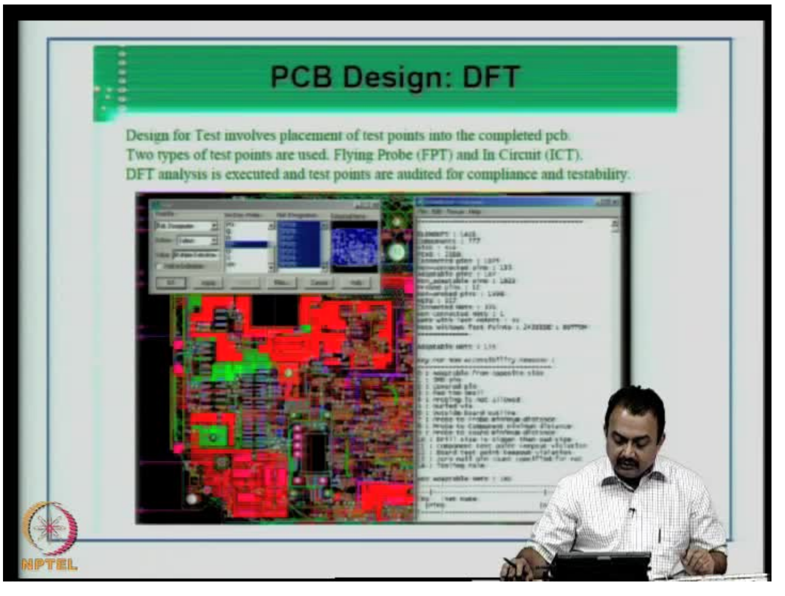
There is always possibility that you may have to pour Copper in your design. When is Copper used? Copper areas are created, the red areas that you see here and poured over vias, the white circles that you see here in this figure and solder pads, these are the solder pads. Copper areas are then assigned at net name that matches with the appropriate net connection. So this is a very important period in your electrical design, where you assign the area or Copper pour area that you design for electrical circuit performance and sometimes the traces end with the Copper. So, this will act as ground areas and these are very essential in a multilayer board.

(Refer Slide Time: 10:38)



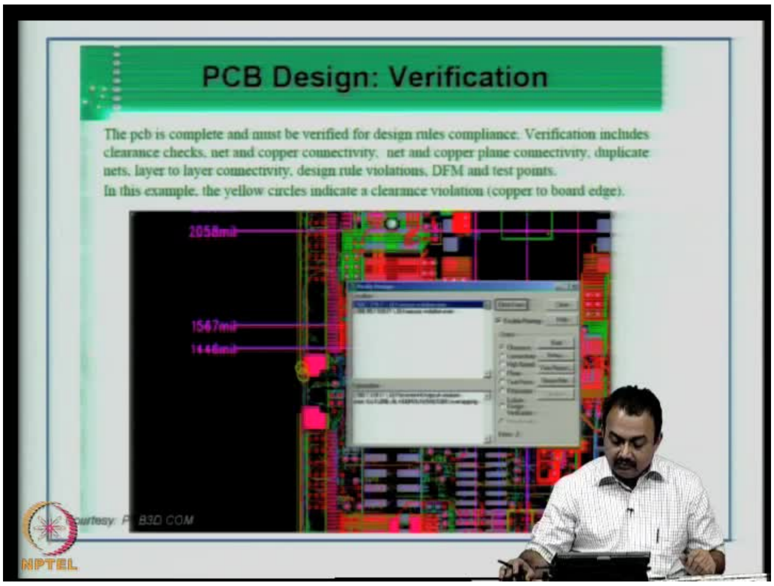
This is the plane of a PCB design. The Copper planes are created, split and defined according to the design rules and net requirements for each layer. Thermal and non thermal connections are placed accordingly, sometimes the Copper can act as a heat sink and therefore vias can end in a plane. So it requires the creativity of a designer to put these Copper plane areas in the inner layer of multilayer board and assign the tracks or lead the tracks to the plane. So the via connections need to be decided in terms of the via size and on the plating for the vias, whether it should be an anti pad connection or it should be a thermal relief connection for thermal purposes and so on.

(Refer Slide Time: 11:33)



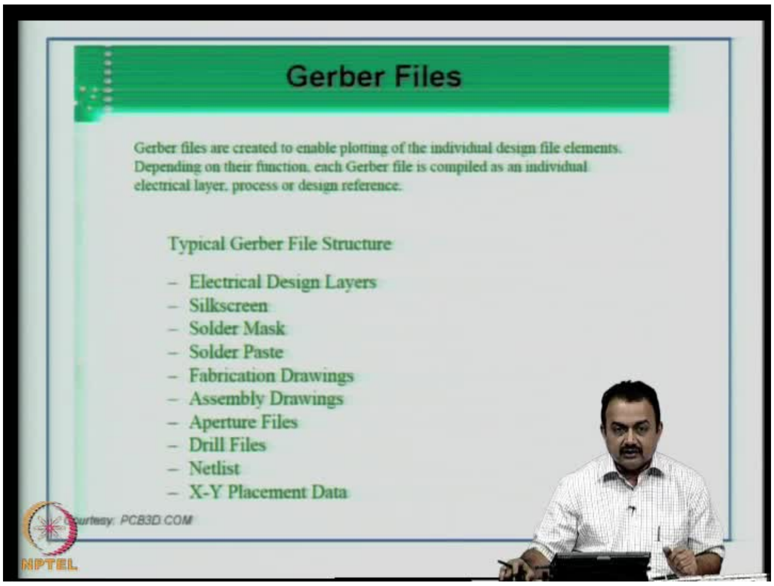
Design for testability: I have talked about this earlier, design for test involves placement of test points into the PCB. So you have to design what types of test points you should have: one is an in circuit testing and other is the flying probe test. If it is going to be a equipment based, then your placement has to be accordingly well suited for accessibility to your flying probe tester. DFT analysis is executed and test points are audited for compliance and testability.

(Refer Slide Time: 12:04)



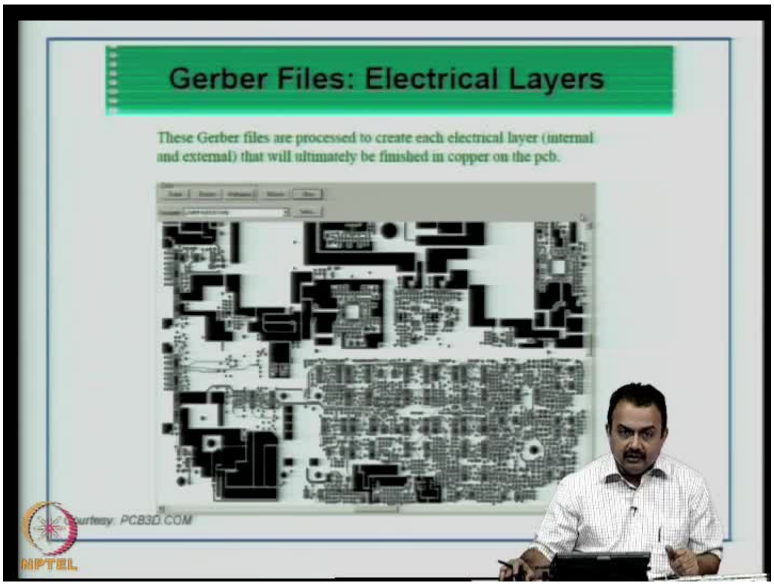
Verification of the electrical circuit is an important requirement. The PCB is complete at the CAD stage and must be verified for the design rules, whether they have complaints or not. Verification includes clearance check in terms of let us say: via to via clearance, via to pad clearance, pad to pad, pad to track, track to track, track to edge clearances and so on. Net and Copper plane connectivity duplicate nets layer to layer connectivity and if it is some multilayer board, design rule violations that you have to periodically check based on a design rules, design for manufacturability and test points.

(Refer Slide Time: 12:49)

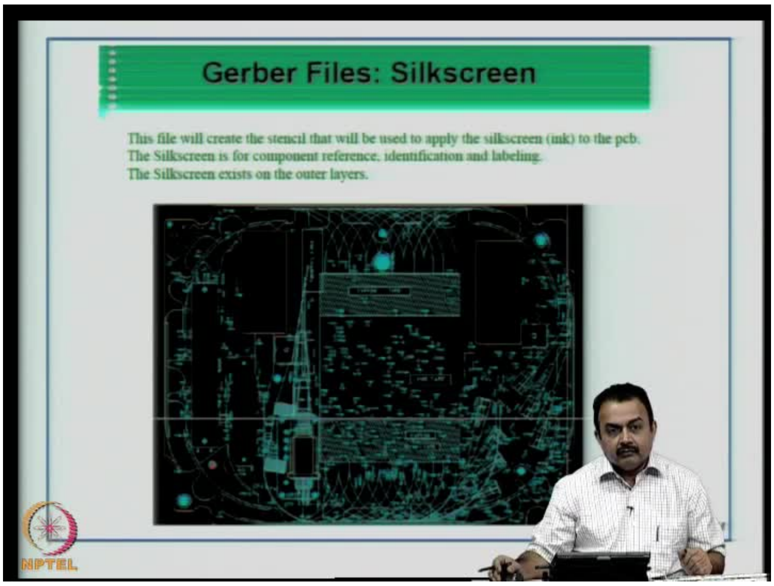


Then you create the gerber files. Gerber files are created to enabled plotting of the individual design file elements that you have completed and depending on their function, each gerber file is compiled as an individual electrical layer, process or design reference. So, what of the areas in which you can create a gerber file? As we have seen earlier, you can create electrical design layers: it can be multiple layers of layer 1, 2, 3 and 4 for a four layer multilayer board, silkscreen legend areas, legend layer, solder mask layer, it could be top solder mask, bottom solder mask, solder paste that is required for a surface amount of assembly technology, fabrication drawings documentation, assembly drawing documentation, aperture files or you can call it as decodes that is similar to your gerber file that is used for looking at the pad openings, pad sizes for your electrical layers. You can also create drill files that need to go for manufacturing, netlist and X-Y placement data for assembling.

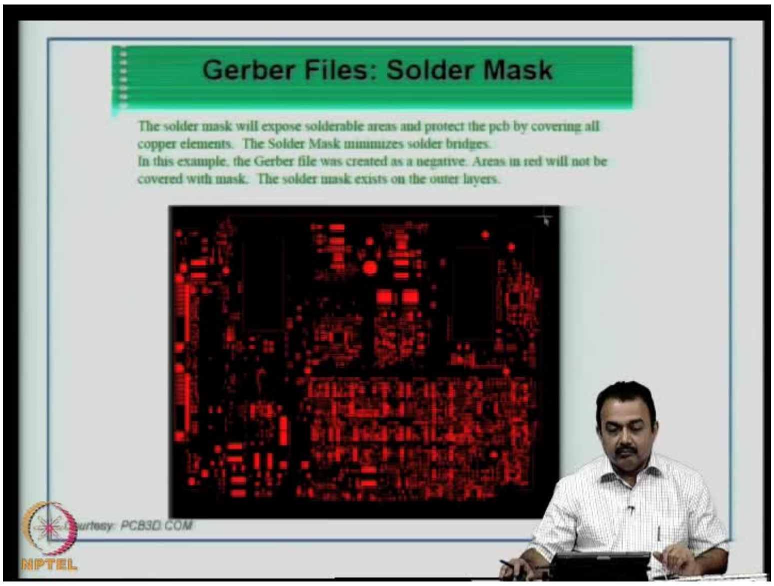
(Refer Slide Time: 14:18)



(Refer Slide Time: 14:38)



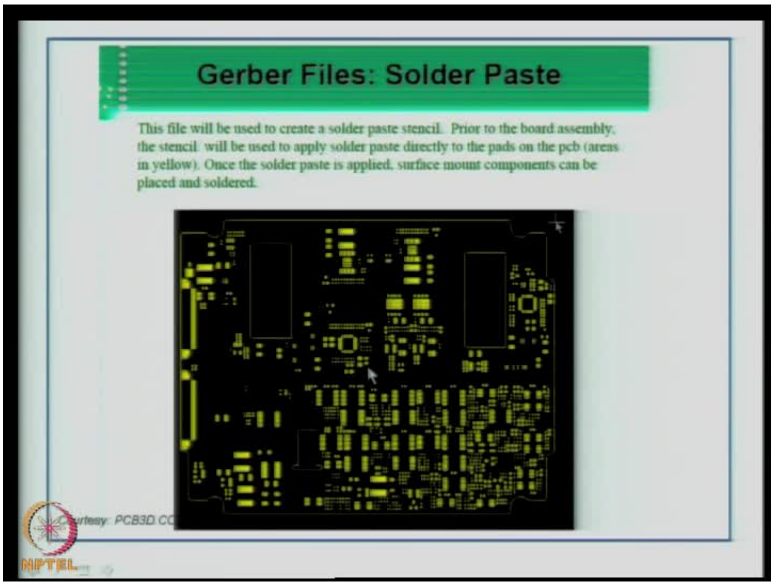
(Refer Slide Time: 15:22)



Gerber files are processed to create each electrical layer internal and external both, that will ultimately be finished in Copper on the PCB. So, this is an example you see of a plotted gerber file for a particular electrical layer. You can create multiple electrical layers. Now this is an example of a visual capture on a screen of a silkscreen legend gerber file. This file will create the stencil that it will be used to apply silkscreen printing, that is an ink that is used to the PCB. So, this is basically text information that will tell you where the component is located or need to be placed, identification of a component and proper labeling of the PCB. So this will screen legend text that exists on the outer layers. You do not have to do it for the inner layers because there is no component in the layers. You can create a solder mask gerber file, which will exposed solderable areas only and protect other area of PCB by using an epoxy ink. We have seen this in one of the earlier classes. So for that you have to create a mask and this is again done by printing process or other advance methods like dry film methodology or curtain coating methodology is also available.

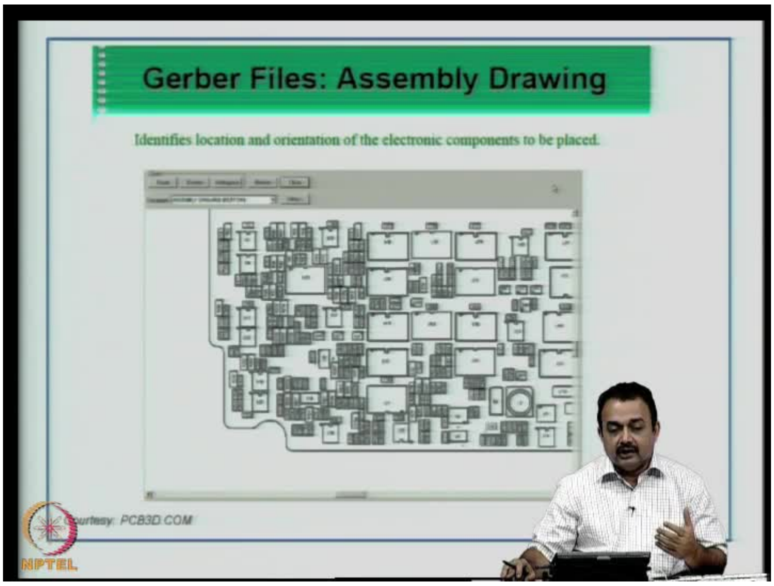
The solder mask utility minimizes solder bridging between components because it protects the entire PCB, except the pad areas. So in this example, the gerber file is created as a negative. Areas in red that you see here, will not be covered with the mask and the other areas will be covered with the solder mask. The solder mask exists on the outer layers only so needless to say, solder mask will be applied only on the top and bottom layers of a Printed Wiring Board and not on the inner layers.

(Refer Slide Time: 16:31)



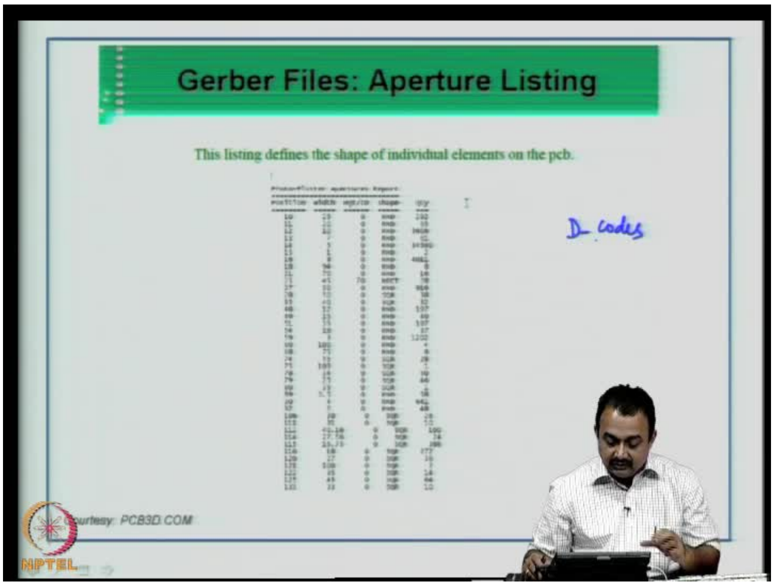
This is the file that is use to create solder paste mask. Gerber file for solder paste printing is a methodology that will be used for mounting surface mount components after the board is complete. So instead of this is one of the methods for assembling surface mount components. So we will get to know about this when we talk about surface mount technology, but it will be better to know that as a designer you can also create this solder paste gerber file. So you can see the openings here, that is the surface pad areas of all the components on top of which your solder paste will be printed or it can also be syringe dispensed in very minute volumes and on top of which the surface mount components will be placed and then it will be sent for reflow soldering and that is how the attachment takes place, of the surface mount devices on to the surface of this board.

(Refer Slide Time: 17:42)



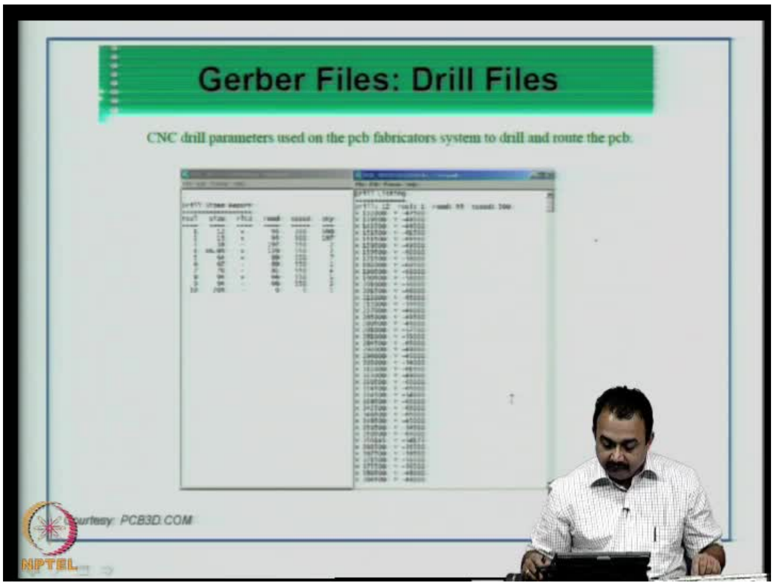
What you see here in this figure is an assembling drawing, which is an important document that is required for any process of fabricating a PCB which requires future reference; it basically gives you the location of the components and importantly it will give the orientation of the electronic components that is used. It will also give an idea that if you do too much of experimentation with the orientation, you will end of with the poor design, a poor manufacturing yield.

(Refer Slide Time: 18:20)

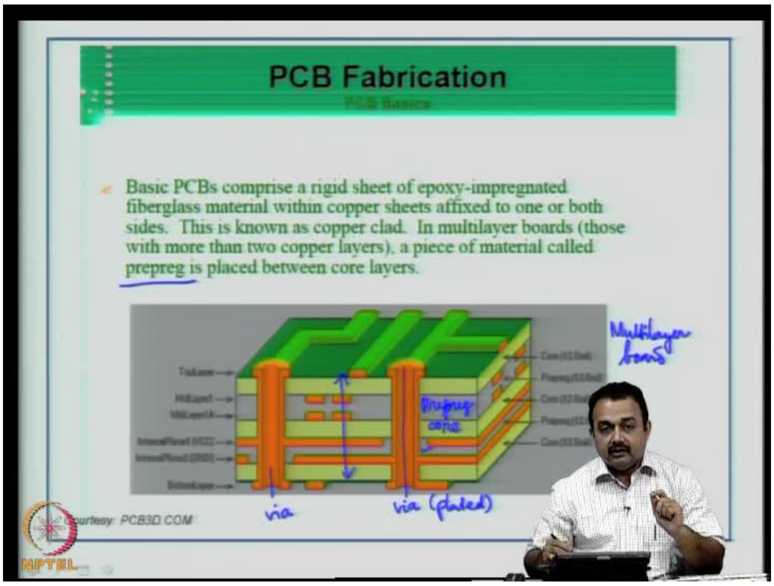


What you see here now is an aperture listing called the D-codes. So the D-codes are basically a listing of the various pads that are used in the particular layer and so the photo plotter that is used for this mask preparation will reach this aperture listing and the aperture in the light source of the photo plotter will open the aperture according to the dimensions mentioned in this list. So this is the utility of this aperture listing.

(Refer Slide Time: 19:05)



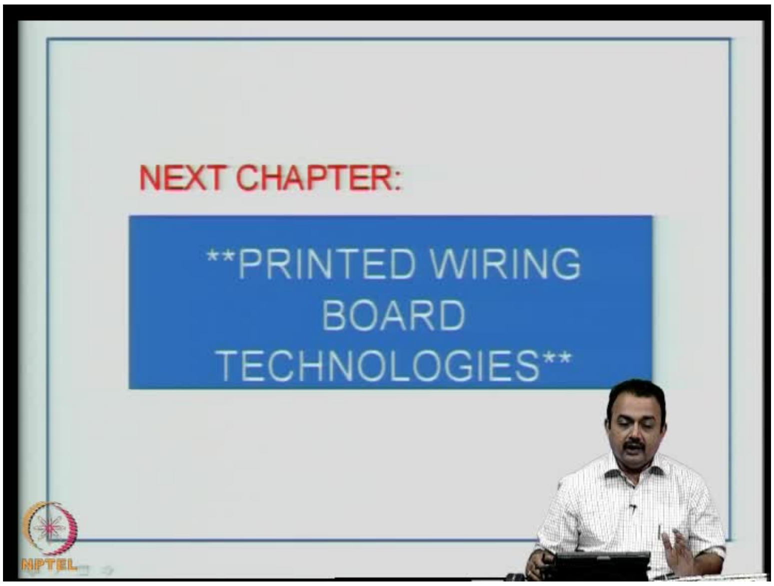
(Refer Slide Time: 19:33)



Then you can create drill files as I said, basically it contains x-y coordinates. It will also list the number of drill bits used in this particular design for drilling and for example, if you have 0.5 mm drill bit then what is a number of holes that need to be used with this particular 0.5 mm drill bit and so on. So this is the manufacturing requirement and this data is fed to the CNC drilling machine. So, that completes the gerber file requirements that you will be able to create, once you finish your CAD work and now the board goes for PCB fabrication. So as we seen earlier PCB are Printed Circuit Boards or Printed Wiring Boards, comprise a rigid sheet of epoxy impregnated fiber glass material within Copper sheets a fixed to one or both sides. So we can have a single sided Copper and double sided Copper and therefore, it is known as a Copper clad laminate. In multilayer boards those with more than two Copper layers, a piece of material called prepreg is placed between the core layers. So if you can see in this particular figure this is an example of a multilayer board.

What you have seen is a cross section; there is a basic core that is used and this is the core and then we have the dielectric prepreg material, this is the prepreg. Then you can have a another prepreg here and then the Copper is built around this and that is how you create Copper layers in the inner layer of multilayer structure and this is the via, that is used to interconnect the Copper layers; this is also a via and this is plated therefore, it is known as a plated through hole structure and you can have a ground and V_{cc} layers well designated in this multilayer and these can be connected to the top or the bottom electrical layer through the vias.

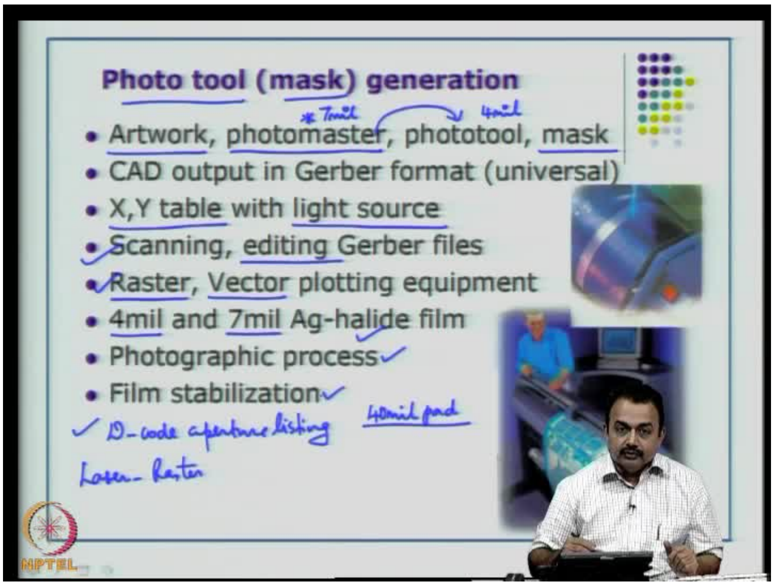
(Refer Slide Time: 21:27)



So now with this background on mind, we will enter the chapter on Printed Wiring Board technologies.

So what we intent to cover in this is, all the basic fundamentals of Printed Wiring Board technology, the fabrication, the materials, the simple processes that make a combination of an extended process called plated through hole process and those that require high density interconnects. Therefore, we are going to start from a simple Printed Wiring Board manufacture for a single sided board to the current technologies that exists for high density interconnects substrates.

(Refer Slide Time: 22:17)



Now, we talked about gerber files- the gerber files is actually sent to a place if you have in your own institution a photo plotter, it can be fed to the photo plotter and you can create the mask. Otherwise, you have to look for a vender who will create this photo tool or a mask and then give it to you or sometimes the manufacturer has this photo plotter which will be in a clean room area because you are going to handle a very sensitive materials like the photo films and then this mask, whether it is a single layer or a multilayered electrical layers and then as I said, all the various other utilities like a solder mask layer, legend layer and so on can be generated with the photo plotter. So we will begin with understanding what is the photo tool. So I will use the term mask or a photo tool interchangeably, both mean the same and different people use different notations. Artwork is basically used when manual taping was done as a means to generate the electrical layers and those methods are obsolete. So you can call photomaster, that is a phototool can be a photomaster if you want to preserve it for a long time and if you can use some thick poly olefin layers to generate your mask and from the photomaster actually you can create a phototool. The phototool or the mask is basically used for actual handling and fabricating the PCB's which is used in the workshop floor of a Printed Wiring Board industry. So it is very convenient to use the term mask or a phototool. Now this phototool is a result of a CAD output in a gerber format; it is a universal format. So basically the photo plotter will have a light source and it will have a X,Y table. Now on this X,Y table you are going to place your silver halide film and then your light source is going to interact with the silver halide material based on, as I said the

D-codes listing we talked about the d code aperture listing.

So this will be used by the gerber equipment, to resolve where the pads are going to be placed, what is the size of the pad and it will also draw the lines that connect between pads and vias and so on. So basically it is an activity, where we will see an X-Y movement on a table with the light source hitting the actual intensity that is required to react with the thin layer of the silver halide that is present on a plastic film.

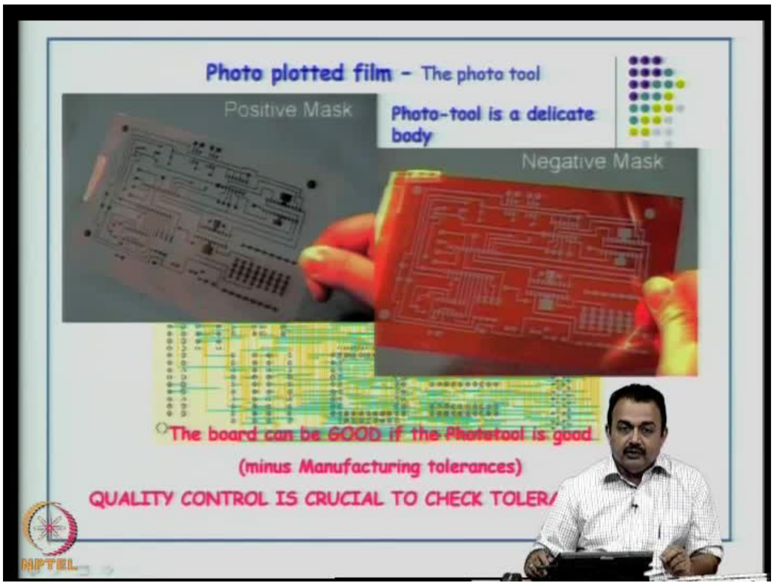
Now the manufacturer can scan and he can edit your gerber files if required, but mostly they scan and look for an error, look for compatibility in manufacturing and look for DFM rules that cannot be met and they will interact with you if they want to change some of the parameters or features. For example, it could be some net which is very small enough, which they can probably not manufacture and therefore they will ask your permission to modify that. So you have to be careful with the editing of a gerber files.

Now photo plotting equipment comes in, either raster format or a vector format. But today most photo plotting equipments are raster and laser based. The initial photo plotting equipment that came into the market was mostly vector photo plotters. Vector photo plotters basically are very slow and what it basically does is, it opens the D-code of a particular value; let us say it is 40 mil pad, that is listed in your D-code. It will finish all the 40 mil openings in your entire design before closing that aperture and moving to the next pad size. So it is basically time consuming, tedious and typically for a 100 by 160 mm photo plot file it would take almost 2 hours, but today the same size you can do it in less than 15 minutes because of the raster technology, that you normally see in your laser printer. So the printing is basically from top to bottom, left to right and so on. So it is basically grabbing the information and doing a quick plotting using raster technology.

Now the silver halide film is the important component, that is used to provide or produce your masks and they come into different sizes 4 mil and 7 mil; 4 mil is basically 100 microns and 7 mil is 175 microns. So typically, for a photomaster you will use a 7 mil film and preserve it and from the 7 mil you can create a 4 mil photo tool or a mask that can be used in the manufacture. Once you feel that the 4 mil film has been damaged or it has lost its dimensional typeness you call it, then you can make another photo tool from the photomaster; that is the advantage of using the thick and the thin silver halide films. Then once the exposure is done to the silver halide film you will have to do a photographic process, which is a regular photographic process. But, today you see we are all using digital cameras and in the earlier times we were using photographic film inside a camera. You expose it and then you develop it using a photographic process and it is the three step photographic process.

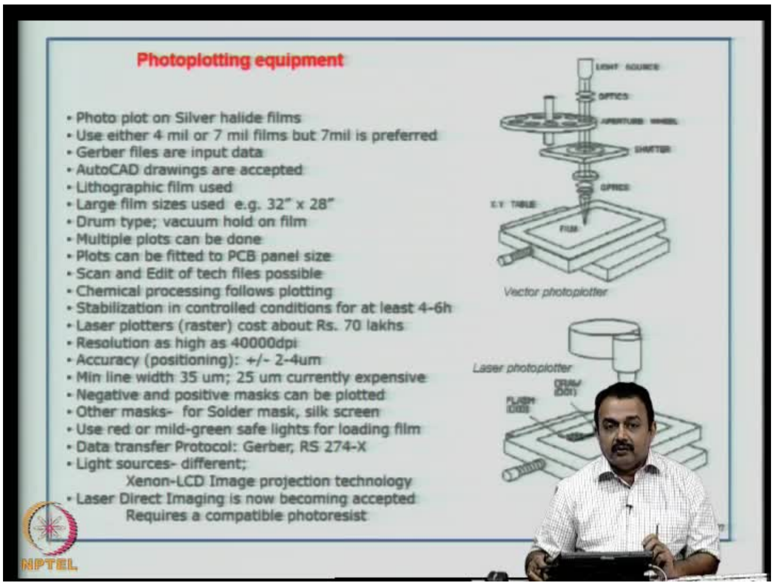
Now that is totally stopped in the commercial aspect and you see most of us use digital cameras and directly we take the positive prints. So earlier we use to go through the process of creating a negative and from the negative we take a positive print, but this mask generation is a direct printing; whether we want a positive print or a negative print you can do using a photo plotter. But the photographic process has to be undergone because you are going to create a very stable silver halide photographic film and then once the chemical process is over, you do stabilization of the film before it can be used in the workshop floor.

(Refer Slide Time: 30:24)



Now what can you see here is a photo plotted film and this is called the photo tool. It is a very delicate body; on the left side you see in the particular slide you are seeing a positive mask and the right side you see a negative mask. What is the difference? In the positive mask, the circuit areas are black and in the negative mask, the circuit areas are open. So basically, the requirement of using a positive or a negative depends on the PCB process sequence that you have designated for a particular board. We will exactly see when to use a positive mask and when to use a negative a mask. But the left one you see here is, basically black areas are circuit areas and the transparent areas are non circuit areas and here you can see on the right side is the transparent areas are circuit areas and the amber color area, which actually is not the silver halide film here, is the non circuit area. So this has got the lot to do with the process steps. Now if the phototool is good, then your board is good and if your phototool is bad in terms of very poor contrast or broken lines or not well defined images, then your PCB is going to be the same because it is going to be a replica that you are going to transfer from the mask to the Copper surface. So make sure that you do enough quality control when you are creating masks. If you can look at this sample here what are basically shown here is on this side you see, this is the positive photo tool and here you see, this is the negative photo tool. The color difference here is basically, this is a di-azo film where-as this is a silver halide film. So I am now going to explain to you what is the difference between a di-azo film and the silver halide film and when to use each of this.

(Refer Slide Time: 33:02)



Now a typical Photo plotting equipment if you buy or if you look at the specifications, most of what I have listed here will be there. If you look at the figure that I have used here, basically there will be an X-Y table what we have seen here; a film will be placed here on the X-Y table and the equipment will have all the optics because there is a light source and the optics has to collimate this light with a particular intensity and wave length on to the film, that is very important. The distance between the X-Y table and the light source or the lens is very important. Now there will be an aperture wheel which will open the apertures according to the D-codes that is found in your aperture listing from the CAD data. Now typically in a vector plotter, one of the apertures will be open and all the X-Y coordinates where this particular aperture is used, the exposure will be completed where as in a laser vector photoplotter as you can see here, here also there is a light source. Now basically because of the raster format you will see irrespective of the D-codes, they move very fast and the apertures are open very fast enough and basically you are not looking at sequential operation you are looking at the complete film and the laser photo plotter and the laser source is so sharp that you can do this entire operation much faster than the vector of course, the light source used here earlier was different compare to the light source that we are using a now a days. Now basically there will be a photo plot on silver halide films you can use 4 mil or 7 mil films, but 7 mil is preferred because the dimensional stability of 175 micron film is preferred because you might use this in the workshop floor and it will be exposed to various factors like humidity, dust, temperature variations in the exposure equipment and so on. Gerber files are the input data and AutoCAD drawings can be accepted if you want to use some mechanical drawings and then to make a mask, lithographic film is used so that is why it is called lithfilm. Large film sizes are used in the equipment like 32 inches by 28 inches. So you can have multiple plots. Multiple plots can be generated on the film and each film can accommodate various designs, so you do not have to waste the photographic film.

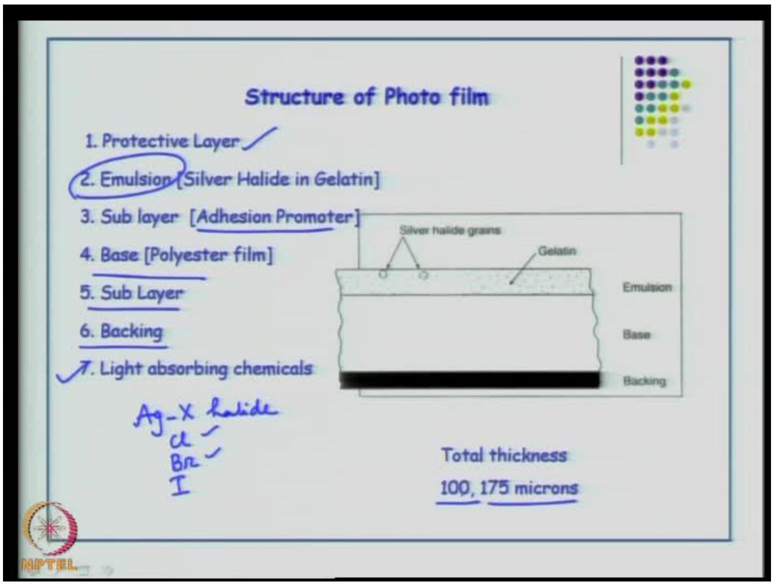
Now there will be usually vacuum drums that will hold this films, so there is no air gap between the film and the X-Y table and also if there is an air gap you can expect variation in contrast of the image. So the vacuum hold is very important for these films. Multiple plots can be done and plots can be fitted to PCB panel size. That means the entire plotting that you do here can replicate the PCB. So this has a lot to do with understanding the manufacturing capability in a particular company. So plating capability for a large size and then all the other sub processes that we are going to see are whether it can be done in the large area or it is an issue.

Now scanning and edit of tech files is possible, chemical processing follows plotting process, stabilization is to be done after the plotting is over, after the chemical process is over and the stabilization time is usually 4 to 6 hour in controlled conditions. In the lab typically where temperatures can be around 21 degree centigrade 55 percentage related humidity, so that the film can stabilize and only after stabilization, you give it to the manufacturing. Typical raster photo plotters can cost in were from 50 to 1crore, that depends on the range and the capability and the resolution and the accuracies and the smaller line which that will be used.

Resolution is as high as 40,000 dpi and accuracy or the positioning part of it is plus or minus 2 to 4 microns as today equipments are available. Minimum line width, typically people require 1 mil that is 25 micron but, very expensive and most machines offer 50 micron minimum line width. You can directly create negative and positive masks, other masks such as solder mask and silk screen can be generated. You have to use red or mild green safe lights for loading the photographic film because otherwise if you have white light, the material will be a expose to white light and then the material will be degraded. So you have to use the recommended red safe light in the lab, where you are using the photo plotting equipment and this continuous for loading the film and until the photographic chemical processes is complete. The data transfer protocol is typically gerber RS274-X. Now what about the light sources, there are different light sources that are being used: earlier people where using Xenon-LCD Image projection technology is also be used, Laser Direct Imaging is used today which is highly accepted. Initially it was not cost effective, but today with more volumes being used it is becoming affordable.

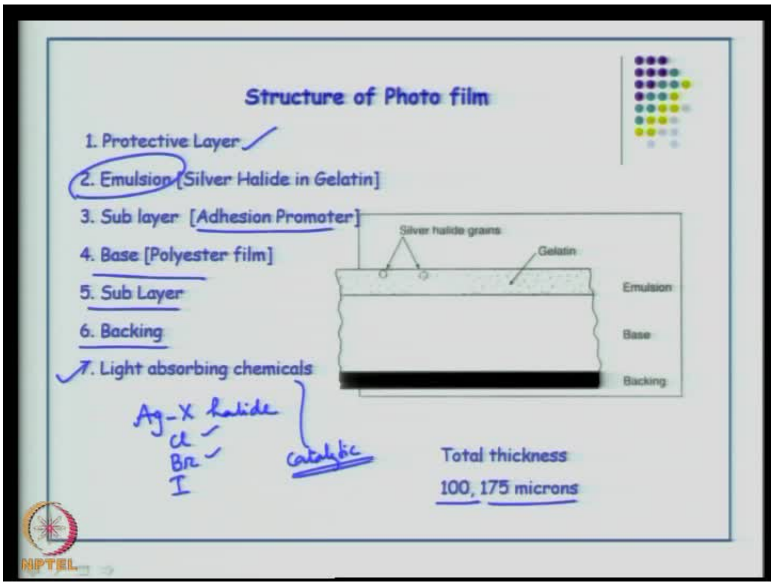
Now if you want to use Laser Direct Imaging, we have to use the compatible photo resist material. So this is in short, what all you can accept from a photo plotting equipment.

(Refer Slide Time: 39:45)



Now we talked about the photo film that is used in the photo plotter. What is the structure of photo film? So basically it contains active silver halide, this is the most important thing that you will have to look for. There will be an emulsion which contains the silver halide in a gelatin format. Now you have to protect it till it is being used, so there will be a protective layer and then there are some sub layers where you are using some adhesion because your base polyester film needs to aider to your impulsion layer; even the protective layer needs to be carefully aider to the impulsion layer till all the handling of the photo film is complete during its process.

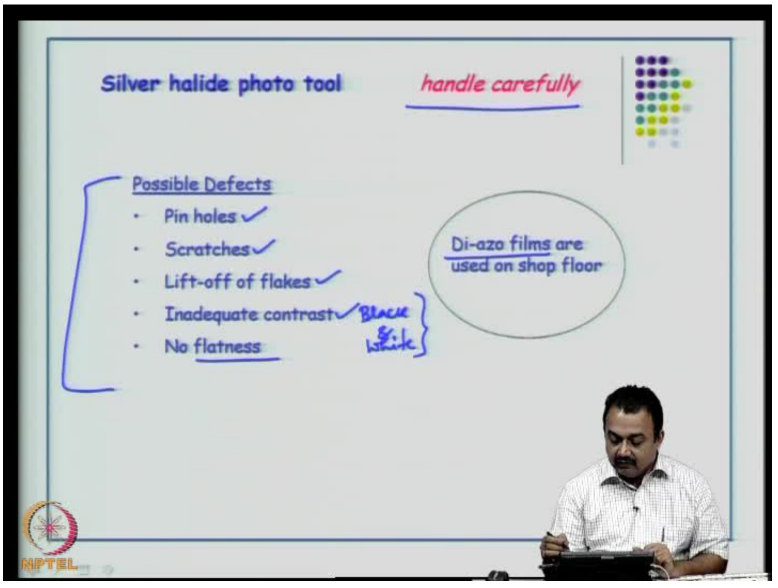
(Refer Slide Time: 41:52)



So there will be a polyester base and there will be sub layers, there will be other backing layers and finally, there will be some embedded light absorbing material because as you know this is a catalytic reacts, light source falls on the sensitive areas of your photo film which is basically the silver halide material, the grains of silver halide. Now what is the silver halide? It can be a Chloride, Bromide or Iodide. But typically, Iodides are not used Chlorides and Bromides are used in the photographic industry. The thickness as I mentioned earlier can be 100 micron that is 4 mils or 170 microns thicknesses that is a standard.

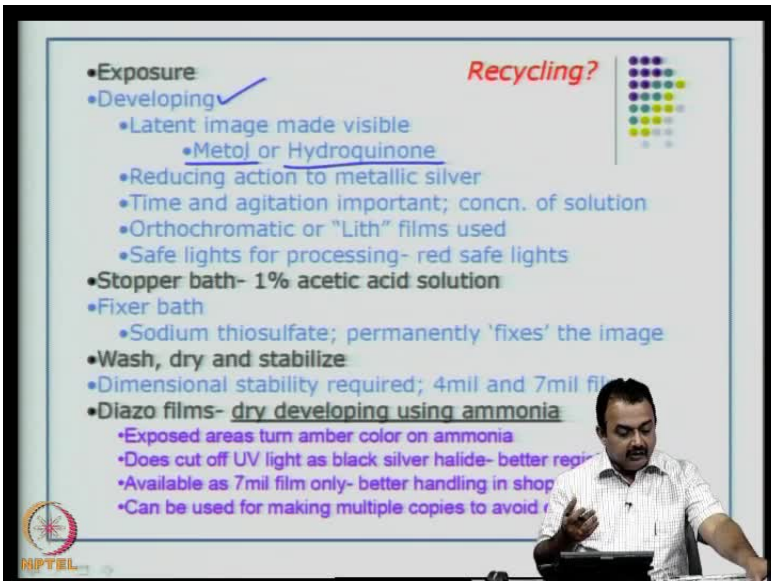
Now I was talking about the light source falling on this film. In the actual process of a photographic process or the photo plotting what basically is required is, there should be some light absorbing material which will start a catalytic reaction. So this process has to propagate through the entire material: silver halide material that is embedded in gelatin and that is very uniformly spread in your photographic film. So this will cause changes chemically and physically and also you will see changes in the color so that is the basic chemical process that you are seeing when you have the light exposed in the photo plotter to the silver halide film.

(Refer Slide Time: 42:34)



Now silver halide photo tool like the mask that you are seen here, you have to handle very carefully because you cannot afford to have defects. You must have a defect free photo tool and what are the defects you can see. You can see pin holes, scratches, lift-off of-flakes and inadequate contrast between the black and the white areas. So if you want a good resolution when you do a photo imaging on to PCB, the black and white areas in your mask should have very good contrast that depends on the photographic chemical process that you are going to do and flatness, you should have a very good flat surface. So make sure that there are no dense and so one of the defects can be sometimes you can see there is no flatness, there is war page or there is some kind of a bend or a sharp corner due to bending and so on. So identify if any of these defects are there and if any defects are there, then they cannot be used for the manufacturing because these defects will get transferred as it is in terms of Copper on to the PCB.

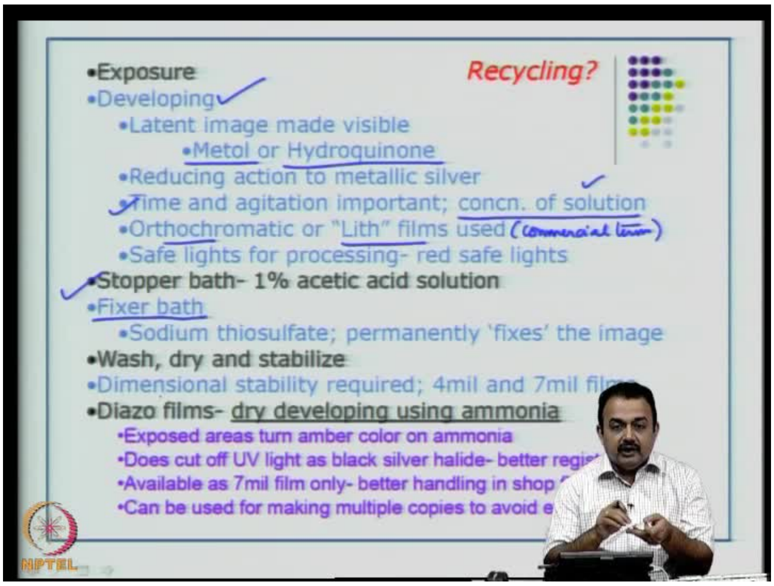
(Refer Slide Time: 44:22)



So the success of handling a photo tool is very important because it directly depends on the success story of your PCB manufacturing. Now the silver halide film can be transferred into a di-azo film that is the trend today. People are not using silver halide film directly and people are converting it in to di-azo films which I just showed you in the sample that is used in the shop floor. So what is the photographic chemical process that one will actually have to do once the exposure in the photo plotting is done? Basically the photo plotter does only light source exposure and after that is complete then you can say that the photo plotting process is complete. Then the film is unloaded from the photo plotter and then it goes for a developing process. Now in the developing process, which is a chemical process you have the developer solution. What is the developer solution?

Typically it is called Hydroquinone or Metol commercial. These are commercial available chemical and this is the one that is recommended by the manufacturer of the silver halide films. In this particular process you are a making the latent image visible so if you take the exposed film and if you are going to inserted it into a photo developer, you will see that the process taking place. This conversion of the dark areas coming up on to the film can be visibly seen and that is how you have seen the latent image made visible, once the exposure is done there is an image which we cannot see. Now we are making it visible by doing this developing process. What it basically does is, reduces the silver halide to metallic silver. It is a reduction reaction and that is what is expected from the developer.

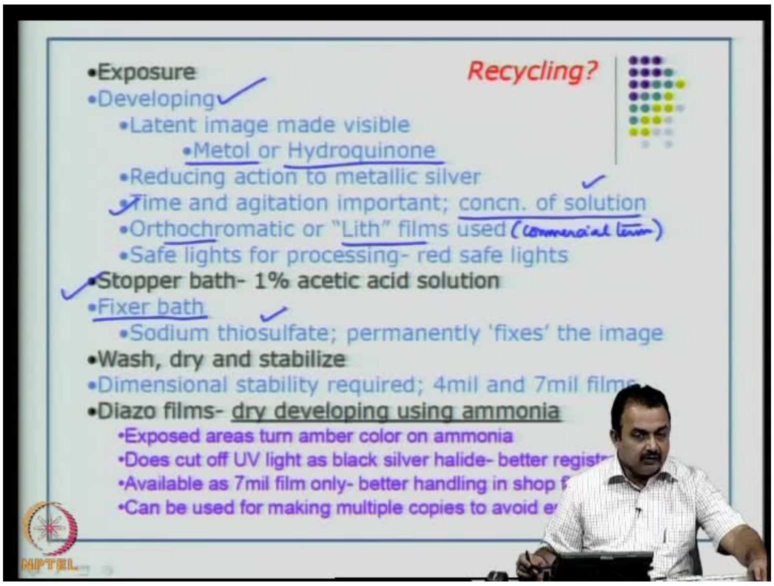
(Refer Slide Time: 46:12)



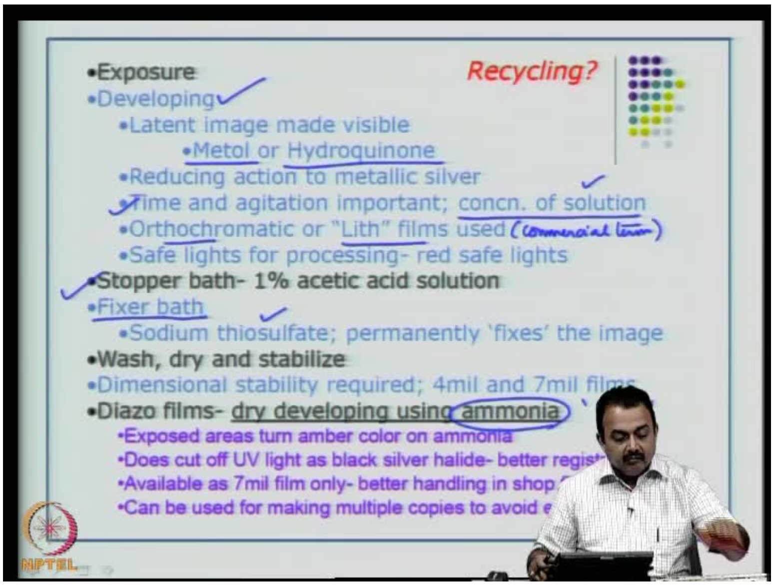
Time and agitation of the developer bath is very important; therefore you have to look at the concentration of the solution to achieve a very good developing process. Orthochromatic or “Lith” films are used where the commercial term is lith. Then you have to use safe lights, once the latent image is made visible you can stop the reaction at any point of time using a stopper bath. It is basically a 1 percent acetic acid solution. You take out the film if you can really read at that point of time that the developing process is complete, you can stop the reaction or if you want the developing process to continue more time then you can put it back into the developer solution. But, such allowances are not acceptable and these are timed process cycles. You have to be very careful with the timing process because these are very quick process; typically a developing process will be over within 90 seconds and again that depends on the stability of the developer bath. Time is very important, agitation is important and the concentration of the solution is also important.

Then once it is complete, you can put it into a fixer bath. The fixer bath- the capability of fixer bath is that it permanently fixes the image. So what you see here in this film, you cannot do any damage to the circuit areas here, unless you take a knife and scratch out the black areas; this is a very permanent image, very stable image and it has stabilized well. So the fixer bath will create this kind of a permanent image. After that is done before that the fixer bath chemically is typically a Sodium thiosulfate.

(Refer Slide Time: 48:09)

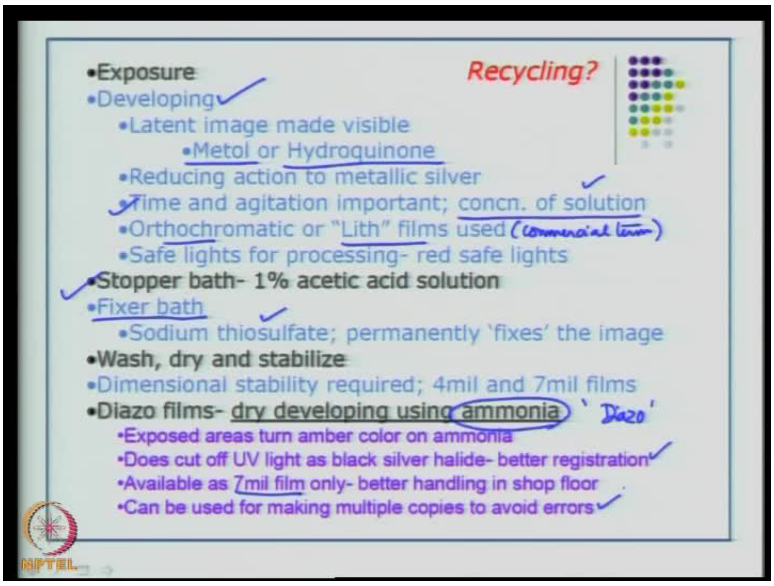


(Refer Slide Time: 49:17)



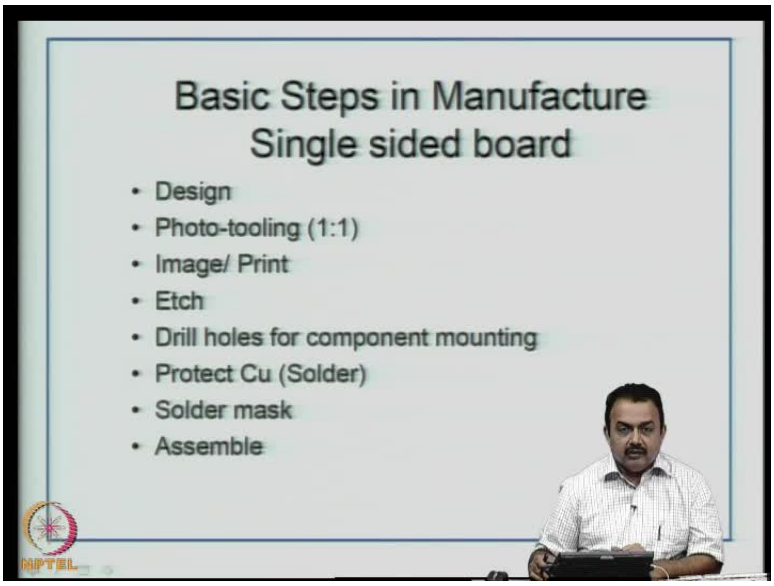
Now the concentration depends on the suggestions given by the manufacturer of course, if you have experience with this process, you can work with these concentrations and then you can wash, dry and stabilize the film; dimensional stability is required and now the di-azo film that you see here- this particular film what we see here comes from a typical, developer, fixer chemical process. But, where as a di-azo film that you see here is amber in color does not go through this process cycle. Now the former one is a wet process, where as this one is dry process. I will explain to you how a di-azo is obtained. Di-azo films are basically a film, a plastic material, poly olefin plastic material that is coated with the di-azo compound. It is a organic compound, it is a di-azo compound that does not require the chemical process that we have seen just now. It basically utilizes the dry developing process using ammonia. So once the exposure is done in the camera or in the photo plotter equipment or basically you can do a 1 to 1 contact of your positive or a negative image and you can do a simple light exposure using UV light to the di-azo film, it is basically a duplicating process; then you can take the film and dip it in dry ammonia for a couple of minutes and you can see the amber color coming up. So basically there is a different set of reaction that takes place between the di-azo compound and the ammonia. Now the advantage is that the amber color that you see in this film also cuts of UV light just as the black area in the normal silver halide film cuts of UV light. The amber color also cuts of UV lights with the advantage of: the amber color providing better visibility for us to do registration between this mask and your Printed Wiring Board. So that is the advantage of using di-azo film: it is thick, it is 175 microns thick and it can be handled easily and you can make as many duplicates as you want. Suppose if you are done with 2000 boards and if you feel that there is the dimensional problem with this film, then you can throw this and make another di-azo film very quickly.

(Refer Slide Time: 51:17)

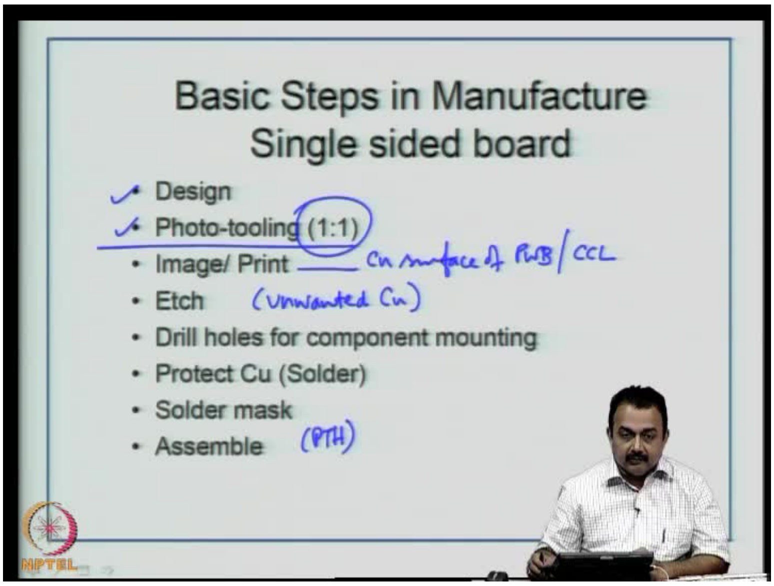


So exposed areas turned amber color on ammonia exposure and it does cut off UV light just has black silver halide does, therefore you can do better registration. Available as 7 mil film only, therefore better handling in shop floor and can be used making multiple copies to avoid errors. Errors that we have seen may be a pin hole, lot off flake or a dust sitting on the areas tracks which are very small and so on.

(Refer Slide Time: 51:46)

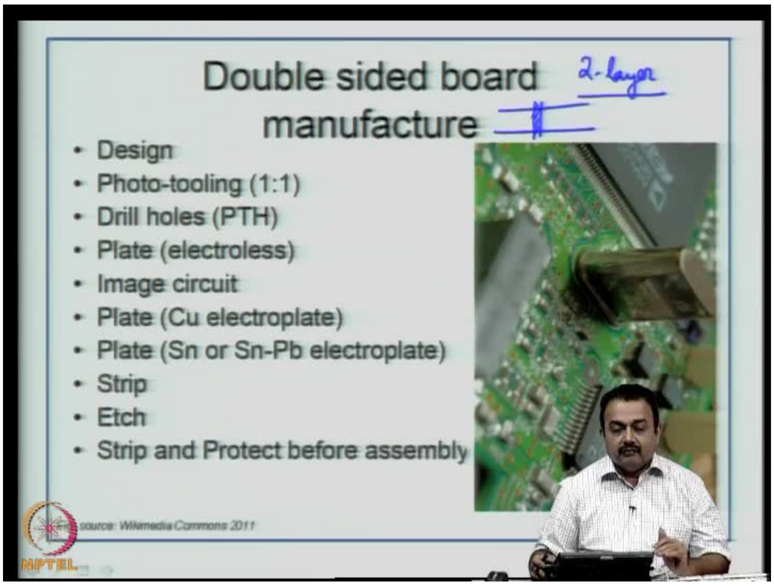


(Refer Slide Time: 52:05)



So this completes the manufacture of masks or the developing process, dry developing, wet developing process for creating photographic tools, photo tools or mask. Now we will proceed into basic steps in the manufacture of a single sided board. We start with the design, we are completed the design, we have seen what a photo tooling process is and so basically here you have to concentrate in creating a 1 is to 1 image that will go directly on to your Copper board PCB. The next step is, you image or print this image on to your Copper surface of your Printed Wiring Board or Copper clade laminate. Then once the imaging is done you etch out unwanted Copper. Etching is the process where you can remove any metal: you can etch a metal, you can etch a plastic, you can etch different metals with the different etchants or inversely a particular metal will have to be carefully removed by using a selective etchant. So in this case of Copper, you have to use selectively etchants that will remove Copper from the surface of the board. Drill holes for component mounting, for your through-hole components. Protect the Copper and there after the solder material. So basically you are first protecting the Copper with solder and then the entire board is protected with the solder mask and then finally, it goes for assembly. So it can be a typically in this particular process cycle, it is a PTH assemble because we are talking about drilling holes for component mounting. In the case of surface mount of technology you do not require drills for mounting your surface mount devices.

(Refer Slide Time: 54:01)



Then we go for a double side manufacturing process. Here the process steps are design, photo tooling that is you create a 1 is to 1 tool. Now the drilling of the holes becomes a prime importance because you have to connect the two sides of the Printed Wiring Board. So basically you talk about a 2 layer manufacture therefore, you will have a Copper layer on top and Copper layer on bottom and you have to interconnect them. Therefore, the drilling of the holes becomes very important and providing interconnection between layer one and layer two. Plate using electrolysis process first the holes and the hole wall, image the circuit using the mask that you have created, plate with electroplating Copper, plate Tin or Tin lead with electroplating process, remove the photoresist or the mask that you have used, then etch out unwanted Copper and finally, strip the photoresist or the mask again and then protect your board with solder mask and finish the entire process.

So this is the very simple listing of the single and double sided process. We are going to look into much detail of all the processes that make up the entire process sequence. This will be discussed in the next class.