An Introduction to Electronics Systems Packaging

Prof. G. V. Mahesh

Department of Electronic Systems Engineering

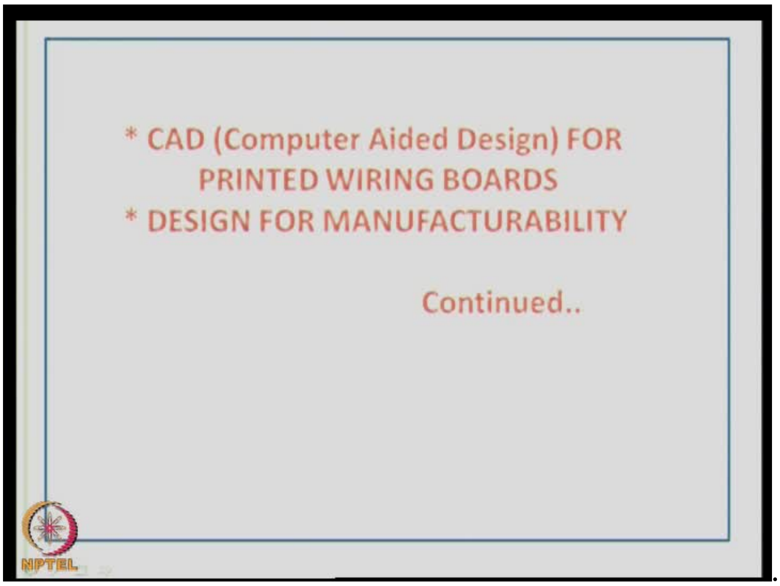
Indian Institute of Science, Bangalore

Module No. # 05.

Lecture No. # 20

Components of a CAD package and its highlights

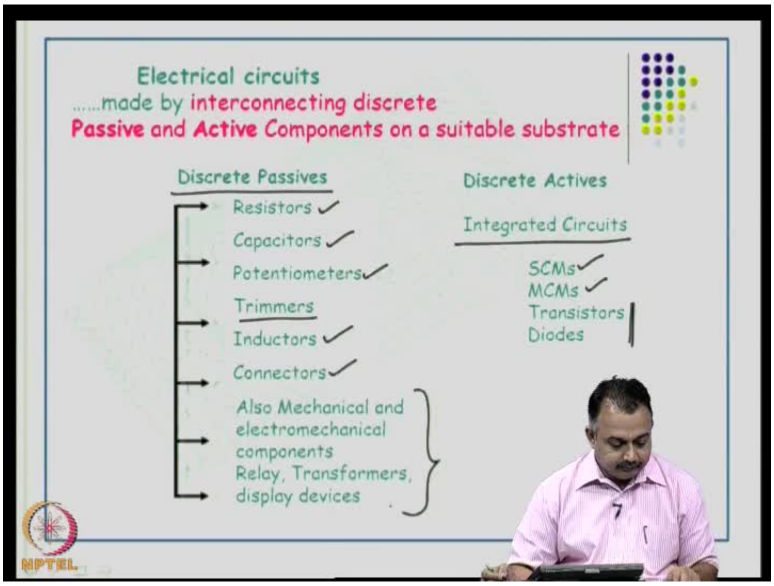
(Refer Slide Time: 00:15)



Let us today continue this chapter on Computer Aided Design for Printed Wiring Boards. Alongside, we are also looking at the subtopic called design for manufacturability and I would also probably introduce few aspects on design for reliability at the end of the particular chapter.

In the earlier class, we had introduced the basics of CAD, the need for CAD, and CAD for Printed Wiring Boards; essentially means that you are taking a major step forward in designing system level Printed Wiring Boards, in defining system level package substrates, because as you know the Printed Wiring boards - as you call it - act as system level substrates, much more functionalities are built into the organic or the ceramic substrate as the case maybe depending on the application. Therefore, you need not look at Printed Wiring Boards as just a simple means to house your components and simple interconnections between components. It is much more than that, because, today CAD totally defines the size of the product. The size of the Printed Wiring Board in some sense, as you saw in the last class, defines the size of the product, the density of the product, CAD also plays some major role in package electrical design and package thermal design. Once you do the basic interconnections of components, you are going to do a lot of simulation, both electrical and thermal, to make the product more reliable. That is why I said towards end of the chapter, we will also discuss some key points on design for reliability

(Refer Slide Time: 02:34)

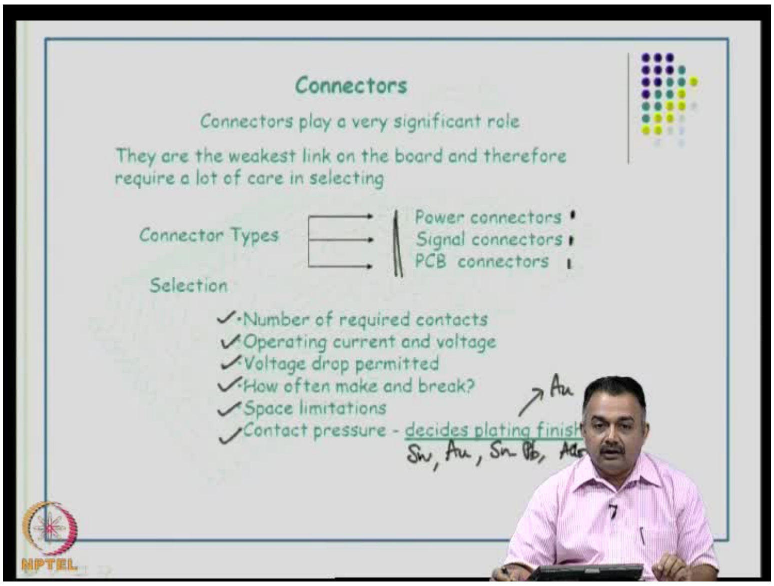


If we can proceed continuing from the last class, we defined in the last class three major points has to be the selection of the components and the need for CAD. Today, if you look at electrical circuits, they are made by interconnecting discrete passive and active components on a suitable substrate. In the last class, I mentioned that it is a designer’s choice. It is a designer’s prerogative to select a suitable substrate for a particular defined application.

Once the substrate is selected, you are basically going to add or mount various active devices and passive devices and then build a circuit along that. Now, the density of this particular activity can be low, medium or very high; that depends on the number of interconnections, the number of components. That is why we use the term component density and board density. What are the basic active devices that we can think about? Basically, they are the intergraded circuits, individual ICs, microprocessors microcontrollers, various transistors, and so on. You can also have on the board mounted single chip modules like your BGA is single chip module, your CSP is single chip module and PGA like your Intel processors or the AMD processors are single chip modules which will be mounted on the Printed Wiring Board.

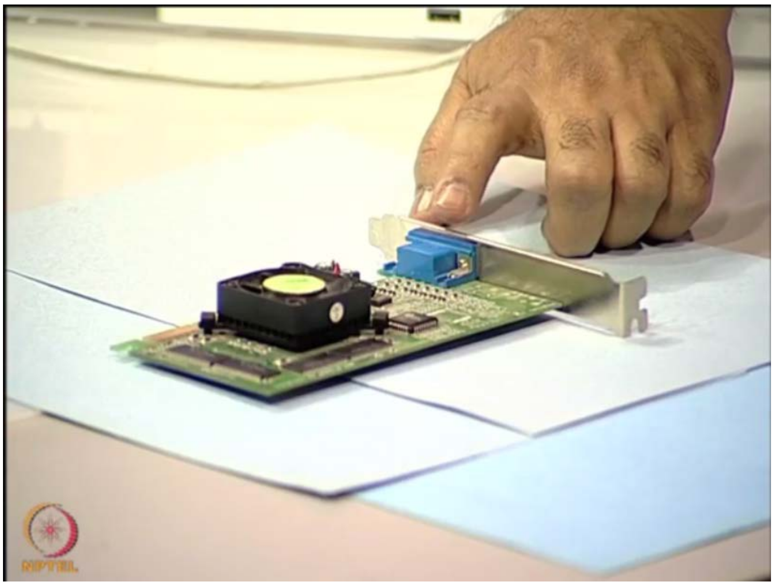
You can also have multichip modules connected, various transistors, diodes, etcetera. These are the basic active devices that could be used. In the case of discrete passives, you will be using resistors, various forms, various types, and various values, capacitors of various values and sizes and different types. Because you can have resistors, capacitors in through hole format or in surface mount device technology format then variable resistors, potentiometers, trimmers, inductors, inductor coils, connectors. There are various types of connector available in the market today and the connectors are very essential if you are integrating two or more boards in a particular large system.

(Refer Slide Time: 06:23)



Also, you will be using various mechanical, the electro mechanical components like relay, transformers, display devices, liquid crystal display, other forms of display, and so on. So, how are you going to tackle the entire scenario given here? If we have a high density board, obviously, you will have different types of these components that need to be intergraded, interconnected on a single substrate platform? How well you can utilize your CAD system to its complete advantage or complete utility value and then make a very good design out of it and whose output can easily get integrated with your electrical simulation or thermal simulation? Now, these connectors very often, we do not give due important to the connectors. We normally give important to the active devices. But connectors for one Printed Circuit Board to another Printed Circuit Board play a very significant role.

(Refer Slide Time: 06:50)



If we can look on this board, you can see, there is a connector here, there is the connector and there are different types of connectors. This is also known as connector. This is a H connector and you can also have different types of other connectors. Now, these will get integrated with other boards. For example, this could be a daughter card that can go to motherboard. Therefore, there should be a reliable connection between two Printed Circuit Boards. So, connectors play very significant role. Probably, they are the weakest link on the board and therefore it requires a lot of care in selecting what type of connectors you require for the particular application.

The connector types are power connectors, signal connectors, and simple PCB to PCB connectors. If you look at the power connector, because of power handling requirements, a large power, we may have to choose very carefully the connector type. The size maybe different. Signal connectors, on the other hand, can be very small size because of the current carrying capacity and the power handling big different from the power connectors. PCB connectors essentially could require to be used for connecting 2 modules on a server. You may have 16 different boards connected to a major platform, a major board and individually you can pull out these Printed Circuit Boards and attach to the motherboard by means of simple connectors.

But these connectors have to be reliable. Therefore, the selection is based on the number of required contacts, number of pins that is required for this connector contacts, then what is the operating voltage and current. So, accordingly as I said earlier in the classification of connectors, you will have to choose based on the operating current and voltage requirement. Because the type of finish that is provided on the connectors will be different for each of these classified connectors, what is the voltage drop permitted, does the material that is used to this connector allow that, how often you are going make and break the connection using these connectors. In some cases, you will be pulling out the PCBs on to the mother board back and forth large number of times.

Therefore, these small pins have to be very sturdy, the plating on these connectors have to be essentially gold finishes because of the contact pressure that is used therefore, you look for the contact resistance that is required and therefore, you will choose connectors based on the plating finish. Typically, you can have tin plating finish. You can have gold plating finish. Earlier, we were also using tin lead plating finishes and other thing like alloy 42 and so on. Obviously, there will be a wide choice. We have to choose the connector based on the space limitations, the number of times you are going to make and break a circuit using these connectors.

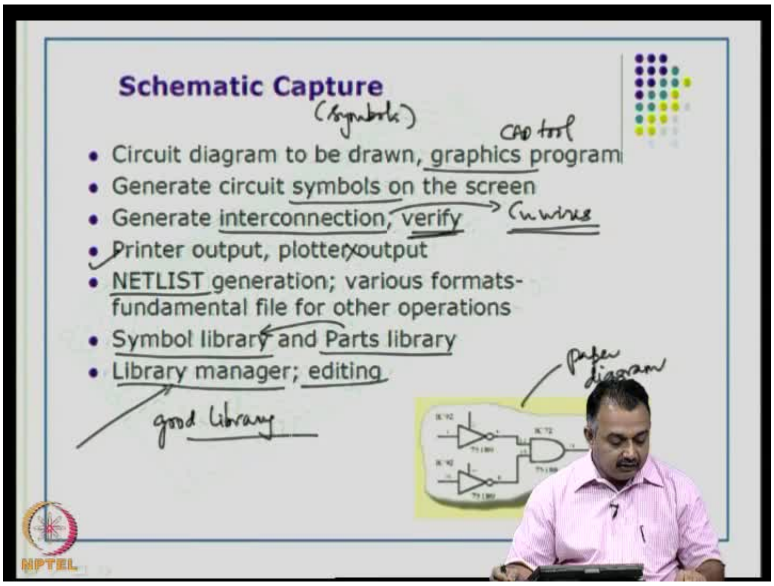
(Refer Slide Time: 10:33)

Now, the CAD emphasis, the approach to CAD for a designer is that the effects of a CAD layout on the final product quality and the cost of the manufacturing has to be definitely considered. Otherwise, utilizing the entire power of CAD tool is lost. So, you have to really dig into the CAD software tool and look at how well you can utilize to get a final product, that is of high quality and at same time you have done entire process sequence using CAD, where it is manufactureable first of all and it lowers the cost of manufacturing.

A designer should have a general understanding of the end users assembling capabilities. We talk about manufacturing first. Then we also talk about assembly. So, a designer should be able to quickly recognize the loop holes in manufacturing, the prerequisites for a particular application in manufacturing, the key issues in assembly, and what failures can happen if you do not take care of each of these two very important aspects. Overall view of the manufacturer's capabilities should be understood; because, today, the board fabrication is very complex process. We have various technologies available today in PCB manufacturing. Therefore, there are many companies which give various technologies; various chemistries that have been researched for a long time and that have been bought with a large capital investment. Therefore, if the manufacturer wants to deliver high quality Printed Circuit Board, he also needs to interact with the designer and provide these advanced facilities to the designer so that the entire combination of providing a high density, high reliability board can be fruitful.

Reasonable understanding of processes like high density interconnect is important. Because today as I had been mentioning when we talked about packages, high density interconnects are the order of the day. Even if you want to make a BGA substrate, you have to go in for a high density interconnect multilayer; then obviously, a designer should be aware of the design rules and manufacturing processes that are closely related.

(Refer Slide Time: 13:21)



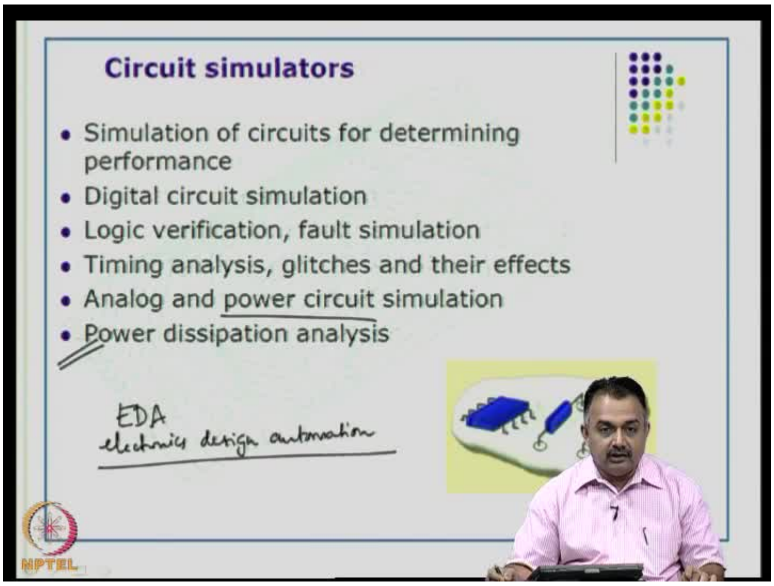
The first thing in a CAD process is that there will be a paper diagram. For example, assume this is the paper diagram that you have of a schematic circuit of an electronic circuit. You have to now do schematic capture using a CAD tool. What you do for that? You have to create a circuit diagram using a basic graphics program that would be available in the CAD tool.

Now, you generate circuit symbols on the screen like you have a symbol for a IC, NAND Gate, AND Gate, OR gate and so on. Accordingly, you go to the library of the CAD program, generate the circuit symbols or pick the circuit symbols that are the available in the in-built library of the CAD software. If it is not available, then you have to generate by yourself based on the data sheets that are available. You generate the circuit symbols on the screen. So, schematic capture is all about symbols. Very important! Now, you generate interconnections. Once you have inputted all the components that are required for that particular design circuit, you generate interconnections and verify. These interconnections are basically copper wires that you are going to realize to connect component to component. We will detail all of these aspects shortly. After this stage, once your schematic capture is done, you can take a printout. You can take printer output you can take a plotter output. Obviously, today, nobody uses a pen plotter but you can take a printer output for the study. The next step will be from this schematic capture to generate what is known as a NETLIST. NETLIST generation is a universal task, whatever be the CAD software that we are going to use. You have to generate NETLIST based on your schematic capture and this NETLIST will be the first source input for all other modules in any CAD ware. It could be in various formats. It is easily editable using your WordPad or Notepad or any other word processing tool. You can read the NETLIST and understand how a NETLIST is generated as the name indicates it is a collection of various Nets or interconnects that you have generated.

To do a schematic capture, the first requirement is make sure you have a very good symbol library and afterwards, you have a parts library that you need to annotate or relate to the symbols that you have picked for that particular design. Basically, if you are in an organization or if you are in a institution, where lots of design are being done, you have to spend lot on looking at a CAD ware that provides very good library manager and which can be editable, that can be utilized for your future different component symbols that may not be available online ok.

Make sure that when you buy a software you have a very good library. This is a must, because you are paying large amount of money for this particular source of information where you can take the symbols and start working with your design.

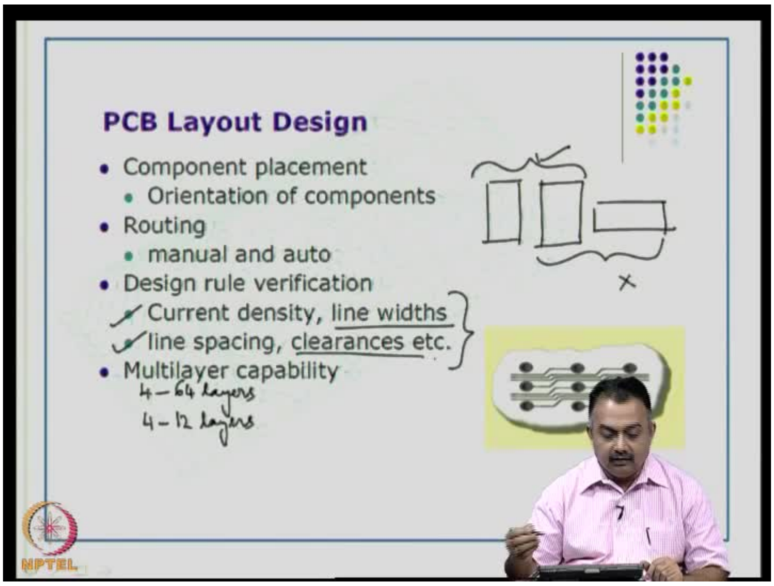
(Refer Slide Time: 17:28)



Some CAD software will have circuit simulators. Low-end software will not have simulators. But you can work with low-end software and utilize those schematic and NETLIST information to input to another high-end software which will have a simulator, in case your CAD program or your EDA tool, Electronic Design Automation tool. Electronics Design Automation; that is the term used for CAD programs which contain all the aspects of a Printed Circuit Board design. A circuit simulator will have simulation for circuits for determining the electrical performance.

You can do a digital circuit simulation, logic verification, fault simulation, you can do timing analysis of a particular Net, glitches in the Net, and their effects, you can minimize these parasitics, analog and power circuit simulation. Because of power circuits, you may have different requirements that you need to meet. Therefore, carefully you have to simulate. Then, power dissipation analysis again, as you have seen from the definition of packaging, power dissipation and power delivery are important terms that need to be looked into by a packaging engineer.

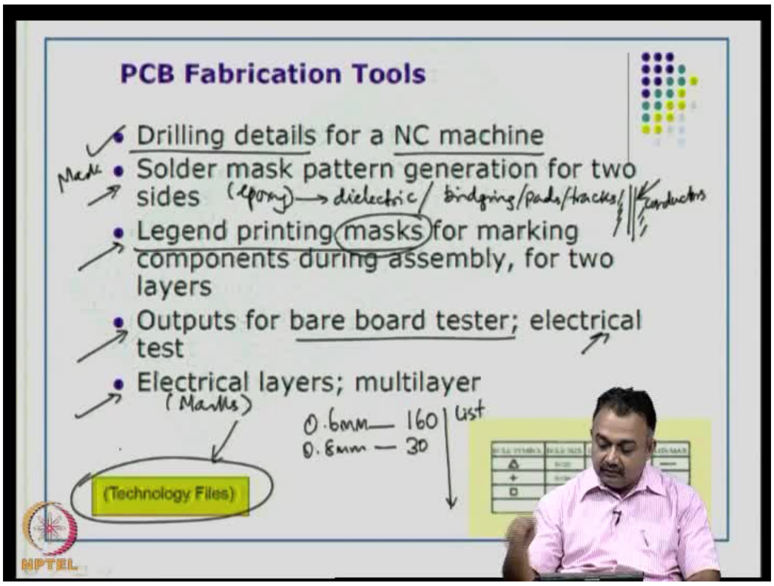
(Refer Slide Time: 19:03)



Then, the next module in any CAD ware will be PCB layout design, which means here you are working with component placement. In a given area which you determine as the PCB size, you have to place the entire set of components both the actives, the passives, the electromechanical components and other parts in to that given space. So the real-estate here is expensive. You have to make sure you utilize the CAD program to make a very good layout. The orientation of the components also is the key factor in deciding or achieving high density. For example, if you have to integrated circuits; (Refer Slide Time: 20:07) you need to align them in the same orientation inside of having one like this and one in this direction, so a combination like this is good whereas a combination like this is not accepted.

There are various reasons for this; not only just from the aesthetics point of view, it occupies more area, it provides difficulty in routing your traces, it provides difficulty during assembly of these components. So, the layout module will also contain various aspects in placement. Then comes routing, there is manual and auto-routing of the interconnects that you have specified in the schematic capture. Then you can do a design role check or design role verification basically to look at current density, the line widths that you will require for manufacturing, line spacing that is specified by the manufacturer, clearances and so on. Your layout should also be able to generate multilayer boards right from 4 layers to even 64 layer boards have been successfully fabricated. But, obviously in a commercial board or in a handheld device or a communication product or desktop motherboard you would not require these many layers. Typically, the handling capacity will be around 4 to 12 layers. Higher layer count is usually for more strategic applications. Multilayer capability should be possible using your layout. We will also talk about various other parameters that I mentioned here like time, the clearances that are required at a later point of time.

(Refer Slide Time: 22:03)



Now, the output of this activity that we have seen so far; the activity that I have listed so far, right from schematic to simulation, to the layout design tool will be to create outputs for drilling. Because, as you know, if you look at this board here there are a number of holes which will interconnect components. Even in the case of this particular board here, there are various through hole components, on the back side also you see assembly being done. Therefore, if it is a 2 layer board and if you have to interconnect these two layers, we have to create a mechanical hole and this mechanical hole has to be made conductive. So drilling becomes a very important step, a very essential activity for multilayer fabrication of your designs. The input from your CAD based on the layout that you have done will be utilized for drilling using a numerically controlled drilling machine. Then another output that can be generated from your CAD program is to generate a solder mask pattern for both sides. I will explain briefly what is the solder mask? We talk much about solder mask later. Visually as you can see this picture, the green areas are the solder mask material.

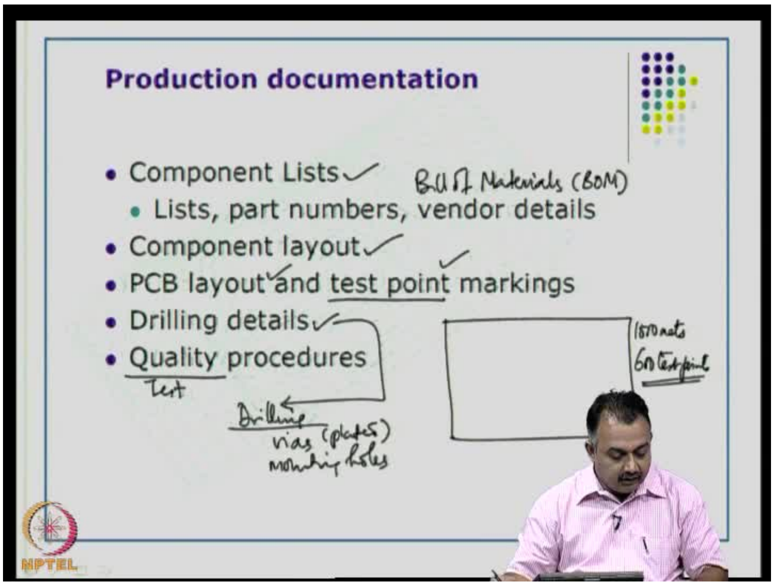
Solder mask is nothing but an epoxy based that is being used here. What is the function of this solder mask? If you look at this particular board, all the pad areas are open. You can see the gold color areas. These are the pad areas which are open on both sides and other areas, which do not have the component pad mounting are covered with green based material called the solder mask. Now, it is not easy to apply solder mask in the non-circuit areas without utilizing the layout assembly details. That is why; we generate the solder mask pattern for both the sides of a double sided board. You will use photo imaging techniques to make sure that your pads are open, the holes are open and all non-circuit areas are closed. What does it do to the solder mask? Here, it is an epoxy and therefore, it is a dielectric material.

What is the function of the dielectric material? It protects all your conductor areas. It takes care of bridging of any pad or let us say, your tracks and takes care of providing better electrical performance like elimination of crosstalk noise and so on between two closed conductors. So, if you imagine two conductors here; then imagine there is solder mask in-between these conductors and the outside area obviously provides a better electrical performance. Imagine in a high density circuit, these lines will be very close and also the pads will be very close to each other. A pad will be very close to a via structure and so on. The other thing that you will have to generate or if it is possible to generate using a CAD is a legend printing masks. If you look at the normal finished PCB like this, close to the assembly the component mark, you will see white numberings being done that denotes the name of the component their reference designation of the component and so on. This is the final finished that is done on top of the solder mask. This is done in white ink. This is done by screen printing process. Therefore, for this also you have to generate a mask. You have to generate a mask for the solder mask and for CAD; you can directly send the file to the manufacturer to drill the number of holes that is required. Drilling a file will essentially contain information like number of holes to be drilled in a particular size of a PCB, the different dimensions of the drill bits that is needed to be used for that particular design and for each of those sizes. For example, if you take 0.6 mm drill there may be 160 holes then if you take 0.8 mm, there maybe 30 holes; so this kind of a list is generated by your CNC drill output and it also gives the x y coordinates of the places where the drilling has to take place including the mounting holes. What is a mounting hole? You see in this particular PCB again, at the edges, you will see holes that are used for mounting large holes, even in the center of the PCB in some cases. These are the mounting holes that are used. It is not providing any electrical connection. It is a mounting hole that it is used for using the PCB to be fixed on to a chassis of a product.

The next one is you can also generate outputs for bare board testing. Obviously when a board is finished, you have to test the board for shorts and opens. The short is a electrical short and open is an electrical open, no connection. So, during the process, there could be reliability issues. At the end of the process sequence, your CAD NETLIST can be used for automated test require input to do a bare board testing. Basically this is an electrical test which is very essential which is compulsory to be done by any manufacturer today.

You can also generate masks for all the electrical layers for a multilayer board. If you have a 32 layer to be done then you have to generate 32 masks; if you want to build a 2 layer board, you build 2 masks. Now, it is time for me to show you what a mask is? If you look at this capture here, what I am showing is a plastic sheet. This is a poly olefin layer and you can see here plots of the pads of the components and the traces. This is the mask for 1 single electrical layer. If you have 8 layer board, you will have 8 such electrical layer masks in addition to this solder mask the legend print for the top and bottom sides. This is the very precision process and this is a simple black and white contrast but, the black and white are of high contrast that provides better imaging once the process start.

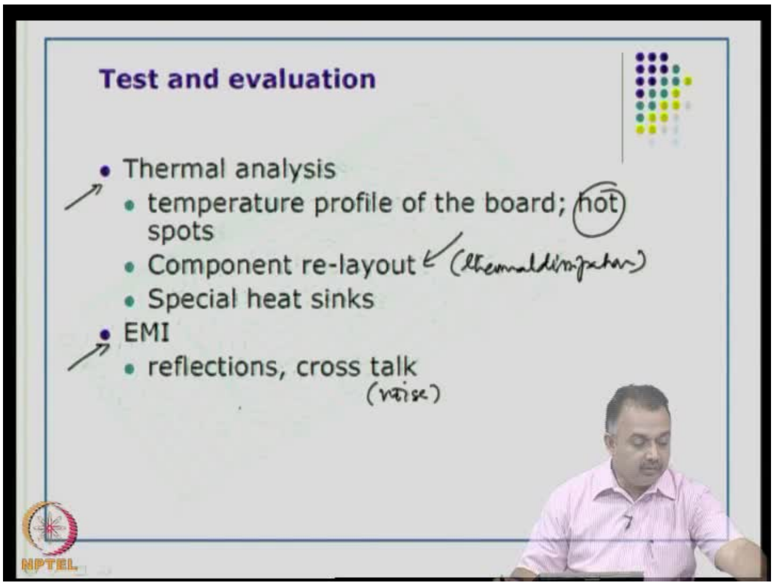
(Refer Slide Time: 30:29)



All of these together is an activity called creating technology files. At the end of the routing process, you will create a set of technology files that would be sent to the manufacturer for doing all of these PCB fabrication processes. I hope this is well understood. In any project, whether it is a student project or a industry project or whether it is a research project you have to do a lot of documentation.

You can generate the following documents from your CAD software tool. You can create component list, you can create a list of all the components that are used; it is known of bill of materials, BOM which indicates the component name, number of components used, cost of each component, total cost of components in the entire design including vendor details and so on. In industry, B O M, bill of material is very essential to determine cost of manufacturing. Component layout printouts can be taken. PCB layout printout documents can be generated, test points, so I was talking about bare board testing. In a particular board, there maybe let us say 1000 Nets; but there will be about 600 test points, so these x y coordinates for a test point maybe required for qualifying the board and this data is meant to be flying probe tester your automated equipment that can quickly give the short and open details and as I said earlier, documentation for the drilling. When you are doing drilling process, there maybe various vias that your are going to generate, some are mounting holes as I said, not necessary to be plated but these are plated to interconnect different layers. In some cases there will be special cutouts that maybe required for physically mounting your board on to the chassis of the product. All of these are done in the drilling process and those details need to be documented

(Refer Slide Time: 33:19)



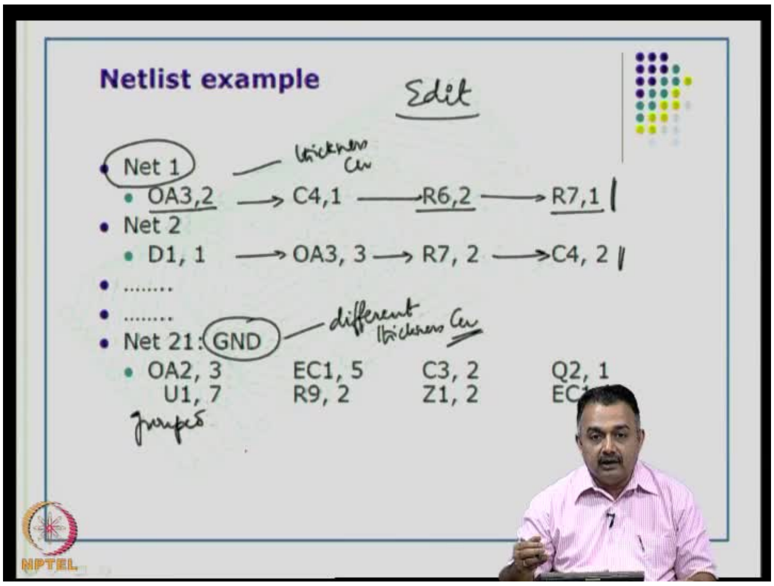
The quality test procedures have to be followed for any good industry that qualifies itself for high quality PCB fabrication. At every stage, you have quality check right after drilling, right after imaging, and so on. We will see about that when we come to the fabrication process. When you come to test and evaluation, apart from the electrical test, another important test the people are doing to the today is the EMI qualification, Electro Magnetic Interference. Here you take care about reflections, cross talk, noise reduction, and so on.

Then the other important thing is thermal analysis. Thermal analysis can be separately done using the layout details of your CAD program. Here, you can check the temperature profile of the board and you can find out hot spots on your board. What you mean by hot spots? These spots are danger areas because of the heat dissipation problems, cost by poor layout of your board. If you grouping a 3 ICs together and all of them are going to dissipate, let us say 25 watts and that needs to be dissipated quickly but, you have not provided enough mechanism, then you have to make sure that you separate these devices far away and also provide additional cooling mechanism. After your layout and placement, you will have to do a thermal analysis. Then if it is not ok, you have to do a component re-layout till such time that your thermal analysis is ok. Thermal dissipation issues are very important.

How do you take care of that? You can add a heat sink. What is a heat sink? I am going to show you a simple device, which is a heat sink that can be mounted on a top of a device. For example, here I have a Printed Circuit Board and there is a active device here. Suppose this is dissipating lot of heat, then I can mount it on top of the heat sink and try to remove the heat as quickly as possible before damaging the tracks and other structures. This is a heat sink material.

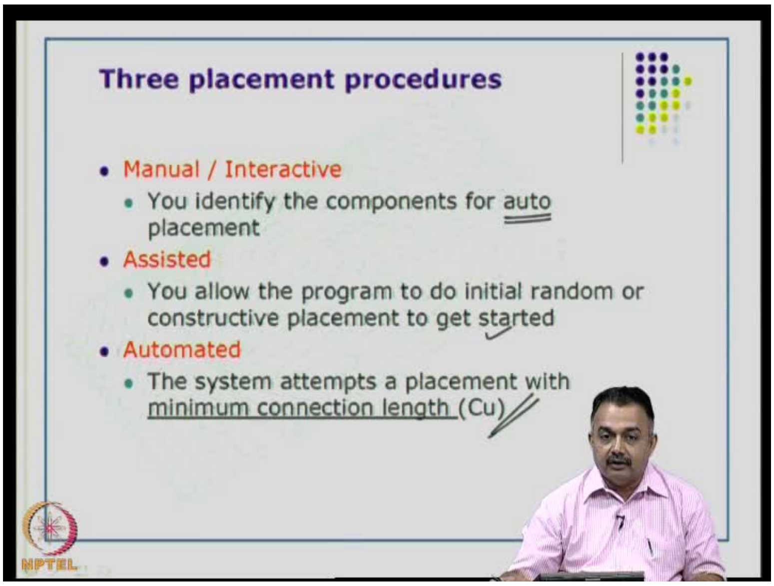
In other cases, you also use a fan you can see in this case, a fan is used on top of the integrated circuit. Obviously this material is light-weight but it is positioned in such a way that it is mounted on top of the active device to remove heat. So, in your thermal analysis you will have to carefully look at whether you require heat sink or a forced cooling method like a fan or you do not require these at all if it is not a major problem. So, this material of heat sink is aluminum material and you can see this is a light-weight and this is coated black. You can see the special corrugated structures here and these structures are provided basically to provide more surface area and more surface area means more heat can be removed from the active device. For an effective cooling, this is coated black anodized aluminum. These are entire set of technology that is available today which a designer should be aware of.

(Refer Slide Time: 36:59)



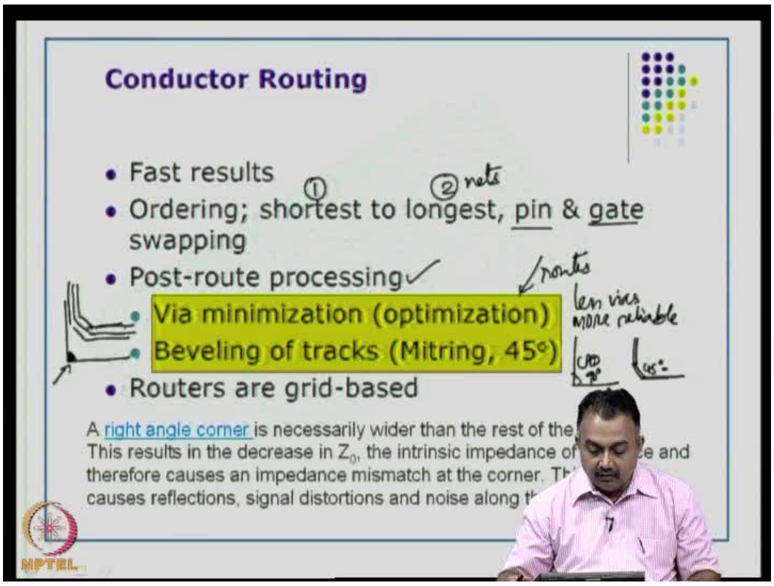
If you look at the slide, what is the NETLIST? You must know how a NETLIST is generated from your schematic capture. For example, you take a Net 1, if you look at any edit program tool like WordPad, load your NETLIST you will see these kind of listings. What does Net 1 say, it connects OP-AMP 3 pin number 2 to capacitor 4 pin number 1 and from there the Net goes to resistor 6 R 6 pin 2, then it goes to R 7 1 and that complete the Net. So, this is Net 1.Net 2 again D1 1 to OP-AMP 3R 7 C 4 like that, you can have hundreds of Nets. You see that typically in a NETLIST, the ground points is all grouped together.

(Refer Slide Time: 38:52)



The ground connections are grouped together and the reason is you can assign different track width. So, you can have a different thickness of copper for the ground plane to remove heat. Thickness of copper can be different. For these, the thickness of copper can be much less because it is basically signal lines. In some cases, you can use these as a heat sink copper and therefore, you can have ground in different layers in a multilayer board, so you can interconnect with vias and push all this ground Net in to the particular layer. This is the advantage of looking at a NETLIST and you can also edit them. In the layout tool, as I mentioned earlier, there is the placement tool. You can do manual placement. You can do interactive placement or you can do assisted placement that is partially used utilizing the expertise of the CAD algorithms that are there in the tool. So, you identify the components for auto placement but the rest of it you can do manually. In the assisted part, you do some random or constructive placement to get started and the rest of it can be done by automated placement.

(Refer Slide Time: 40:26)



Now you can do finally, if you are not interested in manual or assisted, then you can do automated. But experience always says that you have to better learn to do manual placement, because you know the circuit better, you have done the paper work and you know which component needs to go where and which component needs to be closed to which device. Now, the reason for that is you have to make sure that you have minimum copper length for a particular Net. You do not want too much copper added onto your board. Therefore, a good placement means a good attempt to place your components with minimum connection length. That should be the main goal, because your router also will look for minimum interconnection length.

There comes the next part of the tool called conductor routing. Fast results using CAD and the ordering of routing when you start the routing program, it will first finish the shortest length and then go to the longest Nets. You can also do pin & gate swapping. If you give this command or if you ok this particular process, that means if you have different ICs or in a particular IC, if we have different pins with similar functionalities, I/Os, during routing, if one of the pins is too far away then you can allow it swap to the next pin. So, instead of pin number 1 it can use pin number 8 to achieve a complete routing. Also the gates can be swapped. Once the routing is done, it is going to take a long time.

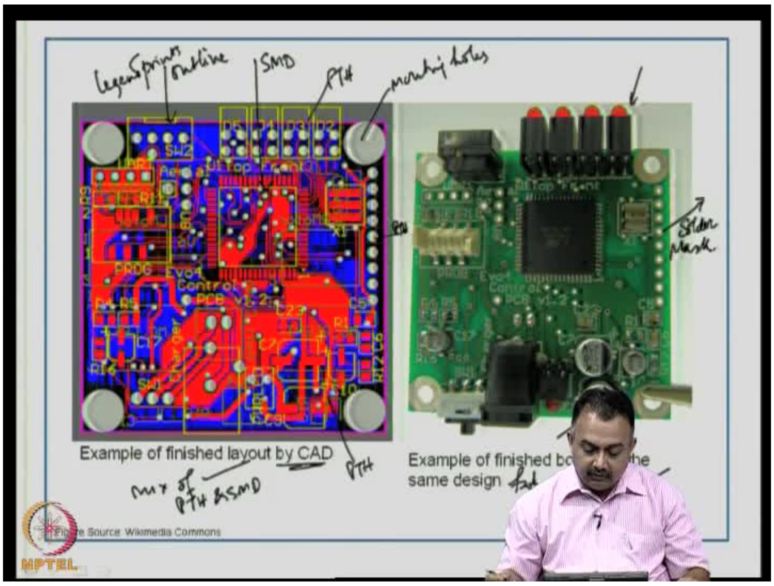
If it is a very high dense board and your CAD program is not going to do 100 percent routing, like you have done some manual placement you also have to learn to do manual routing process for your interconnects. You should not depend entirely on the power of the tool to complete 100 percent routing. It is very difficult to get 100 percent auto routing. At least, 10 percent will be incomplete and you have to do manually. But in some cases, because of restrictions and because of electrical requirements, it is better that you do manual routing first and leave the rest of the simple traces to auto routing algorithm program.

After the routing is done, you do what is known as post route processing. What is that? When you talk about a double layer board to connect layer 1 to layer 2, it will put a via and interconnect from top to bottom but in this process it will add lot of vias during the first level pass. You have to minimize the number of vias. Less vias more reliable it is, because if you look at the manufacturing point of view, any via added, it can be a failure. Until and unless the process is foolproof, try to minimize the number of vias. That is why, we say optimize your routes once the routing is complete. Then you bevel the tracks, that means you have a 90 degree track created by the CAD program but then 90 degree is unaccepted format for both electrical performance as well us for manufacturing purposes. Typically you would like to create a 45 degree, take a 45 degree turn and provide these traces.

The reason for moving from 90 to 45 is that a right angle corner is necessarily wider than the rest of the trace. This results in the decrease in Z_0, that is the intrinsic impedance of the trace and therefore, it causes an impedance mismatch at the corner.

When the signals are passing through, it causes reflection and creates electrical parasitic problems. There will be distortion in the signal and noise generated along the trace. From the manufacturing point of view, if you look at a right angle like this, in processes like imaging and etching and so on which you going to see, this corner will be a major problem. This can chip off, if the processes like etching are not well maintained and you also gain more area when you convert from 90 to 45. Because, if you want to run bus conductors like these, for digital signals, you will get more area, so convert your traces from 90 to 45. The routers are grid based. Before starting a routing process, set your grid distances.

(Refer Slide Time: 45:12)

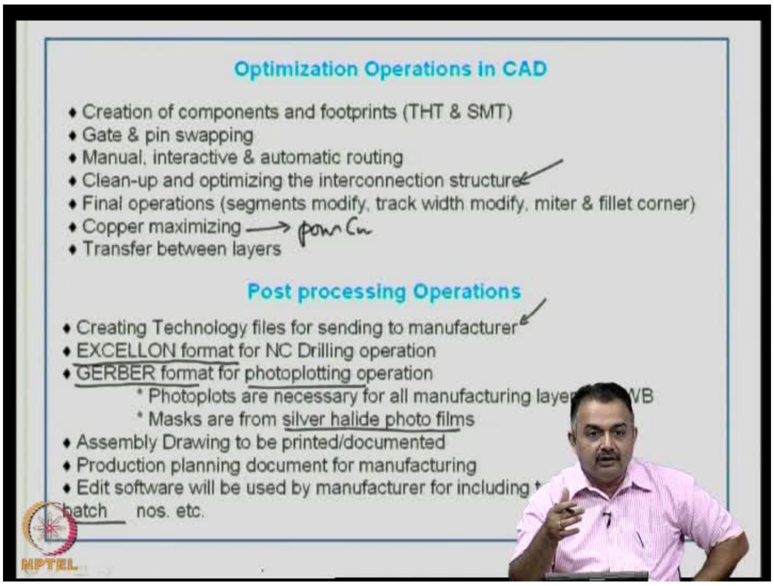


That can help in getting reliable completion. This is a picture that I am giving you as example of a finished layout by CAD whatever things that you can see. On the right, you see the same design has been fabricated and assembled. So, fab plus assembly done here. Here it is a mask. Let us say a CAD view of a layout. You can see these are mounting holes and you can see lot of through hole components.

For example, these are all through hole components. I write it as PTH plated through hole. This is again a P T H. You can also see surface mount. This is the surface mount device. These are all through hole components, so it is a mixture of through hole and surface mount device layout that you see here in this picture. The red and blue here indicates the traces on two different sizes which means for this process you will have to generate two masks; one will be as per the blue lines the other will be as per the red conductor traces that you are seeing here. The red is much thicker, because basically those are the ground planes that are generated and yellow traces that you see here is basically indicating the package outline and these are legend prints which I talked about.

You can also see component name written here, which is legend print. Apart from the pad area and the through hole openings, all other areas will be covered with the solder mask you can see in this picture, the green color, this is the solder mask. In this right picture, on the right, you see the components have been assembled and completed. It is a mix of through hole and a surface mount device. A good optimization has been done to utilize this small area for highbred assembly of through and surface mount devices. Some of the components are very tall, you can see like this, a display device on the edge. Some kind of connectors of the edges, tall components here on the edge so repair and rework is easy. So this is the kind of output that you get.

(Refer Slide Time: 48:00)



Once you finish your CAD work, post processing, we have to optimize. As I mentioned before, what are the optimization operations that you can you in CAD. Creation of components and footprints through hole technology and surface mount technology. If you are working with a particular device, which is not available in the tools library and which you going to use it very often, you create a library of your own which can be utilized and which subscribes to the current existing standards, acceptable standards. Gate & pin swapping can be done.

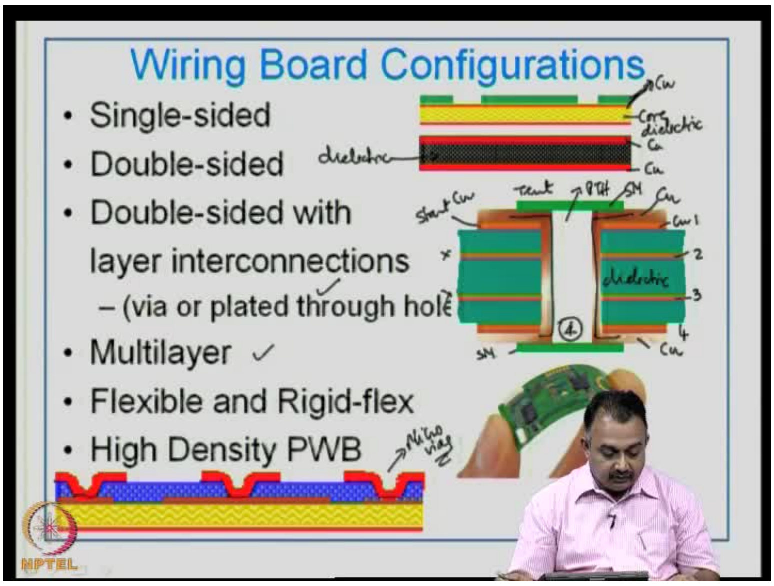
Manual, interactive and automatic routing can be done, clean-up and optimizing the interconnection structure is very must, once you finish your first level pass of routing. In some cases, you may have to do at least 5 to 6 times, the routing process by varying the layout by changing a few components achieving better interconnection, shorter lengths and in terms of thermal also and converting 90 to 45 corners and so on. So, the final operation will be you can modify the segments manually. You can modify or change the track later without going back to the schematic so mitering and filleting the corner can be done. Copper maximizing, in some cases, you want poor copper in the inner layers. We call it as poor copper because you are adding more copper to fill a certain area that can act has heat sink ground plane. Transfer between layers, you can move track from one layer to the other post processing.

Then what are the post processing operations? You can create technology files as I said, first sending in to the manufacturer. For NC drilling machine, there is the universal formats called EXCELLON. All CAD tools will usually generate an EXCELLON format file that will be used for NC drilling operation by manufacturer. Then there is the universal format called GERBER for your photoplotting operation.

The mask that I showed you has pin generated by using a GERBER format. These masks are usually made from silver highlight photo films. If you focus once again on this mask, this is originally was silver highlight film, the black spots that you see here is basically metallic silver actually oxide; so the other area where silver highlight was present has been removed by a photo imaging process. Essentially the starting material for this is a silver highlight photo film that you normally be used once upon a time in photographic but today most of us use digital cameras.

Assembly drawing needs to be printed and documented. Production planning document for a manufacturing and edit software will be used by the manufacturer for including. This is important, because a manufacturer will look at a Gerber file, he will edit it and add tooling holes, he will add his logo, he will add batch numbers of manufacturing and so on. Whenever you look at a board at some corner, you will see all these details of manufacturing batch number, or the company logo, and so on for easy identification of your design work much later.

(Refer Slide Time: 52:00)

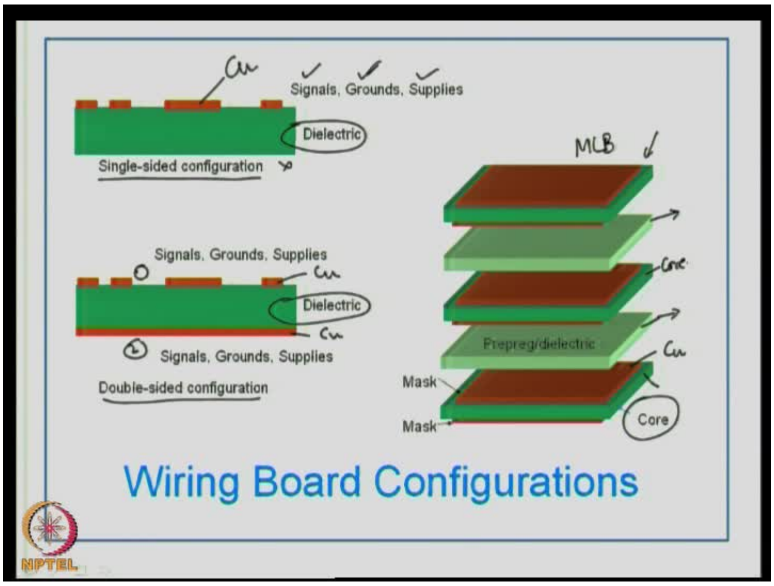


We will come to looking at the wiring board configurations, because if you want to discuss more about CAD, you must know what are the configurations that you have in the Printed Wiring Board? The first thing is a single-sided board. What does that mean? It has copper on one side this is the copper and this is the core dielectric and this is the copper here that you see and you can see that the other side, there is no copper. Typically in this case, this is a single-sided board. The assembly is done on one side, on the other side, there is no assembly done. So, this is a single-sided board. You can have a double-sided board which means you can have copper on both sides separated by a dialectic material.

In most cases, you can also do assembly on both sides, For example, here you can see assembly has been done on both sides. Then you can have double-sided board with layer interconnections using vias or plated through hole. If you look at this picture, this is copper and this is also the inner layer copper, so this is also copper, this is added copper, this is original copper, so you have layers like 1 2 3 and 4 layers of copper and this is the through hole, plated through hole. Where is the plating done? The plating is done here. This is the plated. This one is the starting copper.

This is the 4 layer board separated by dielectric. This is the dielectric material, the green one that you see here, is the dielectric material and this is the solder mask that is protecting your via. You call it as tent. This is the typical plated through hole double-sided or multilayer interconnections. If it is a 2 layer, then you will have these 2 layers will not be there and you will have a double-sided board. Typically, this is the 4 layer board; so multilayer boards. Then you can have flexible and Rigid-flex board as you can see in this picture. This entire board is flexible.

(Refer Slide Time: 56:31)



I would now like to show you a flexible board here. As you can see, this is a rigid board, so this is not flexible. Same circuit has been done on a flexible board. So, this is the flex board. This is a rigid-flex board because as you can see this is flexible in the body and the edges are rigid. This is the rigid-flex. Flexible board combinations are possible. Then, we have high density Printed Wiring Boards, where you can see microwaves are generated on the top of a structure like this and that can be reproduced 3 4 times on the top and the same thing can be reproduced at the bottom.

This particular board, which I am going to show you, is a LAN adaptor card, PCMCA standard card. This is a 6 layer board containing flip chip, surface mount devices, and this uses microwaves for connecting those build-up waves of the order of 90 microns or so. This is a high density board. These are the various configurations that you have for Printed Wiring Boards. The same thing is explained here in a different illustration to make you understand much better and more comfortable. Here you can see a green layer, dielectric and here this is the copper that has been generated. This is a single sided configuration. Your signal, ground and supply lines will be on the top layer only. There is nothing at the bottom. In a double sided configuration, what you see here, there is a copper here, there is also copper here, you can have signal, ground, and supplies at the top as well as at the bottom and this is your dielectric material that separates that two conductor layers.

The thickness of the dielectric is very important in determining the electrical performance. It can vary from 0.8 mm through 23.2 mm. We will see about those things when we come to fabrication. What you see is a multilayer board, you can see a bear basic course structure starting and you can see the mask here that would be used to realize the copper traces. Then you have the Prepreg, the dielectric material that is added later. Assuming this mask is not realized as copper layer, then you will have to build multi layers by adding a Prepreg, another set of copper layers, another set of dielectric, another course structure with two copper layers and so on. So, we will end up with 2 plus 2 plus 2 plus 6 layers of copper separated by 2 dielectric layers that have been added later and these are the core dielectric materials.

We will talk about these structures later. Think about the difficulties in creating the electrical layers for each of this. We will continue with more on CAD utilization for various wiring board configurations in next class.