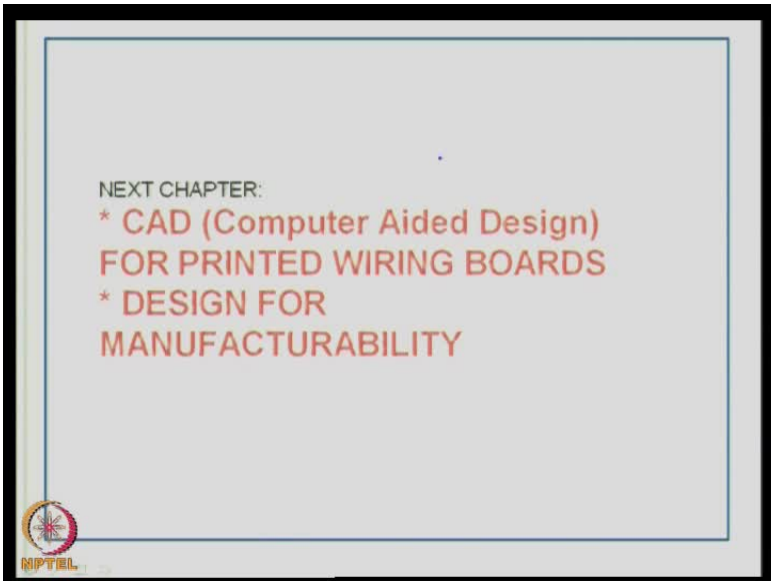
An Introduction to Electronics Systems Packaging
Prof. G. V. Mahesh
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 05
Lecture No. # 19
Quick Tutorial on packages
Benefits from CAD
Introduction to DFM, DFR and DFT

(Refer Slide Time: 00:15)

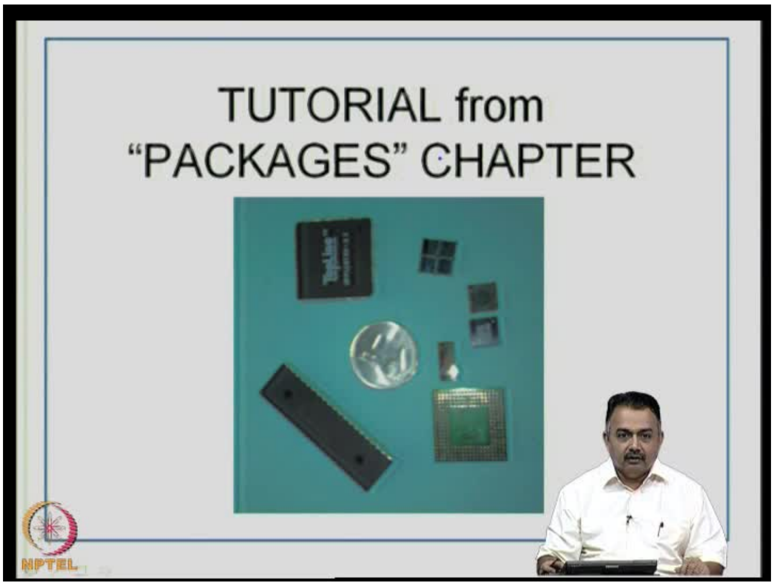


We will move with the next chapter in this course on electronic systems packaging.

The next chapter is based on Computer Aided Design that is CAD, for Printed Wiring Boards. We will focus CAD systems for Printed Wiring Boards. And we will also talk about the concept of design for manufacturability. Many of you may not be aware of this term, but today this has become an essence and a requirement in electronic systems packaging, whether it is at the chip level or at the board level or at the system level.

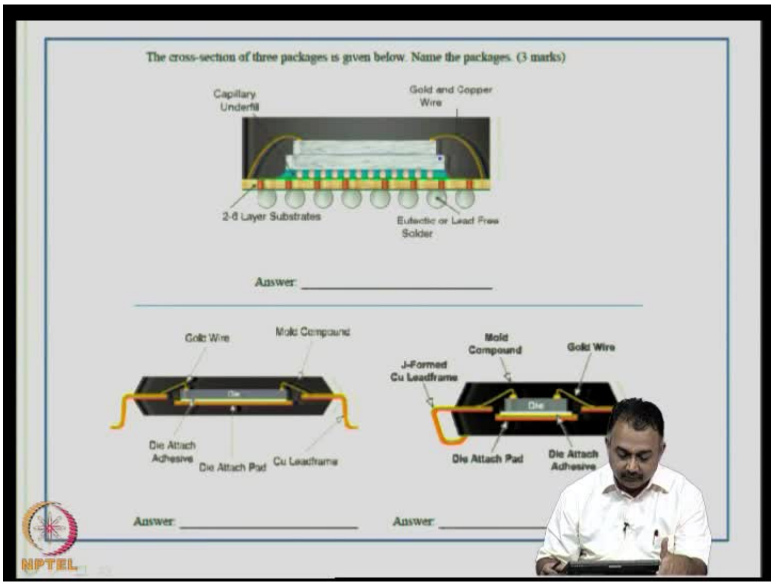
So design for manufacturability is more than a buzz word; many often it is considered a term, a fancy term but then really not applied. So we will see some aspects of design for manufacturability and how it affects the entire process of reliability, testing and so on and how a designer needs to know this term of DFM as well as a manufacturer who needs to know about electronics design.

(Refer Slide Time: 01:45)



But before we go into this chapter, I would like you to just record the previous chapter on packages, where we have spent lot of time on discussing various packages and formats. We will briefly spend some tutorial, some kind of a questionnaire, before we actually go into the CAD chapter. This will be some kind of a refresher, questions and let us see how we go along.

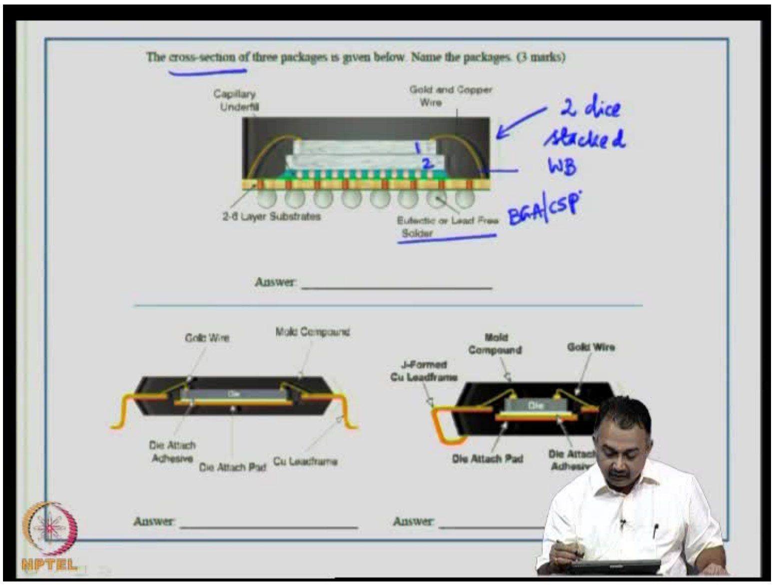
(Refer Slide Time: 02:12)



This is basically to make you more comfortable, more familiar with the cross sections of packages which I have emphasized in the previous chapter.

Now if you look at the first question, what we have here is basically a cross section of a package. Now in the first package, you can see here that there are in this particular figure, there are two dice and this is 1 and this is 2. They are stacked basically. So you must know in this particular configuration the dice are stacked. Then you have wire bond. This is a wire bonded configuration. Then if you look at the final package to PCB interconnection, this is basically solder balls and you can call it as BGA or a CSP kind of interconnection.

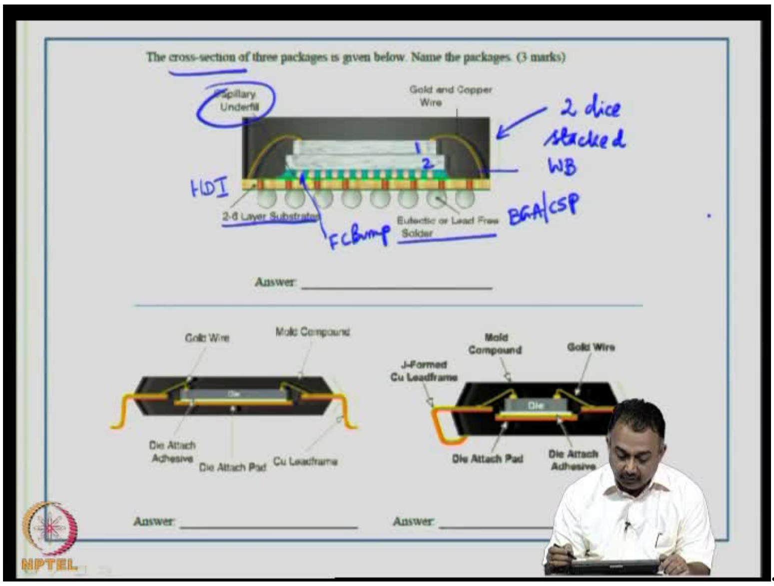
(Refer Slide Time: 03:28)



So, from the figure, it is really very difficult to find out whether it is for a BGA or a CSP but nevertheless the pitches for this kind of configuration is now well within our understanding. And now you can see here in between that, this is mentioned here as a 2 to 6 layer organic substrate or it could be an inorganic substrate too. So, what it means is, there is a high density interconnect that houses this package.

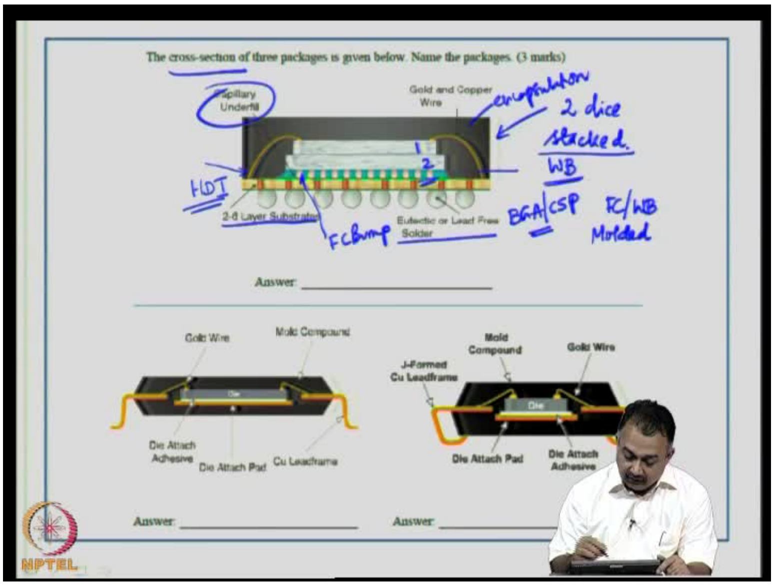
The other information that you have from this figure is that there are bumps, as you can see here, which basically is flip chip bump. And then under fill is used in this particular package that means we can confirm that this is a flip chip. That is chip number 2 is flip chip and that is not same with the die number 1.

(Refer Slide Time: 04:41)



The die number 1 is connected by means of wire bond to the substrate. So, this particular substrate takes up the interconnections of the both die 1 and die 2. Die 1 in the form of a wire bond connected to the common substrate and die 2 in the form of a flip chip array and that is also connected to this high density interconnect layer. Then on top of it, you see there is an encapsulation. This is the encapsulation. These are the components of this particular package and therefore, you should be able to write the name of the package.

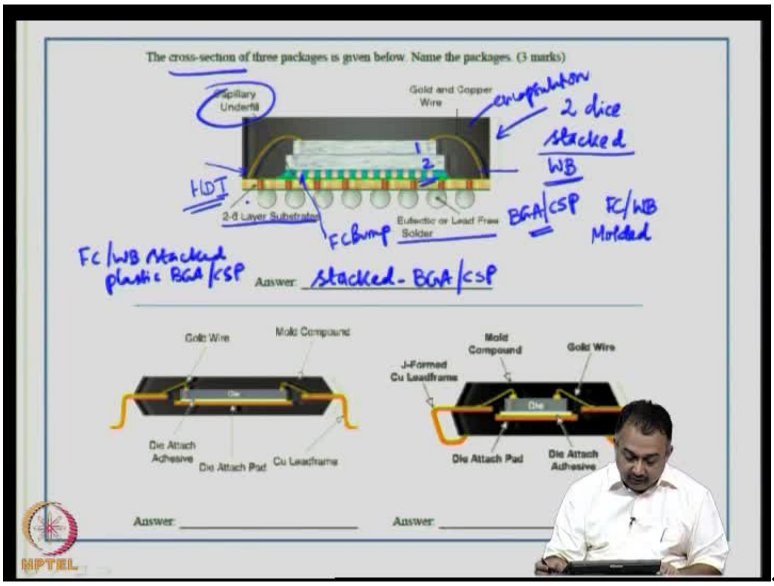
(Refer Slide Time: 06:00).



So, what name can you give? Basically this is a flip chip, a wire bond interconnect and then this is encapsulated, so you can call it as a molded package. Then there is a BGA or a CSP interconnection but the most important thing is you should call it as a stacked package. So this could be a stacked BGA or a CSP. So, the word stacked coming for this particular package becomes very important.

So you can expand the name in a different way. You can say flip chip wire bonded stacked BGA slash CSP and you can also say plastic BGA because there is an encapsulation and if you assume this is a organic HDI substrate, then it becomes a plastic BGA. These are the kind of nomenclature that you can give.

(Refer Slide Time: 06:56)



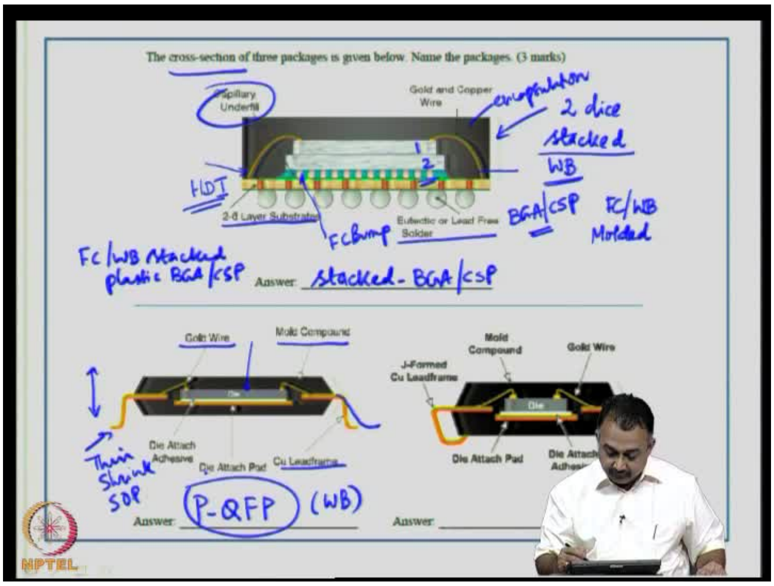
Now, coming to the next figure that we have here in this slide at the bottom left, you see there is a cross section of a device or a package that contains this kind of copper lead frames and you can see there is a die at the center. Then there is a molding compound which protects the die. There is a gold wire bond, which takes up the interconnection, that is the first level interconnect from the die to the lead frame, this is the lead frame.

And there is a die attach adhesive that makes up the platform for the die to sit on without any vibration shock or without any air gap. Then we have a die attach pad that is a platform, so these are the components that you can see in this cross section of this particular package.

So you can see that this lead is not galving. Galving will have a different type of a shape. This is a normal quad flat pack kind of a configuration. This will be basically a QFP. We can say plastic quad flat pack and in the nomenclature, you cannot really explain about the wire bond. So P-QFPs are well known to have wire bonds. If you look at the cross section, this is a plastic quad flat pack.

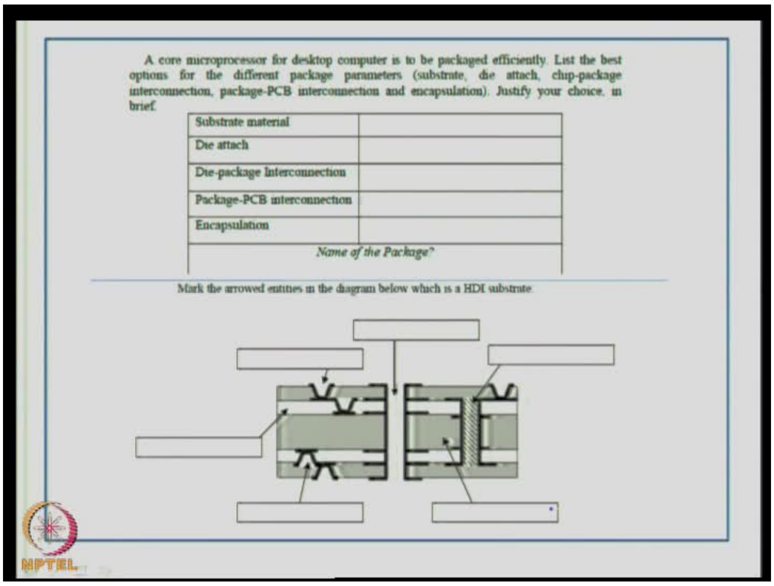
And if you have information about the height, then you can really talk about whether it is a thin package or a shrink package and so on or even a small outline package. If you are able to identify this as a QFP, then it is fine.

(Refer Slide Time: 09:06).



If you look at the second figure here at the bottom, very similar configuration, there is a wire bond, there is a molding compound, there is a die attach pad, attach adhesive that attaches the die to the lead frame and then there is a difference here- that the lead frame is a J formed copper lead frame. So this could be a quad flat J bended lead. This is how you try to find out what kind of a package it is, if you are given the cross section of a package.

(Refer Slide Time: 09:52).



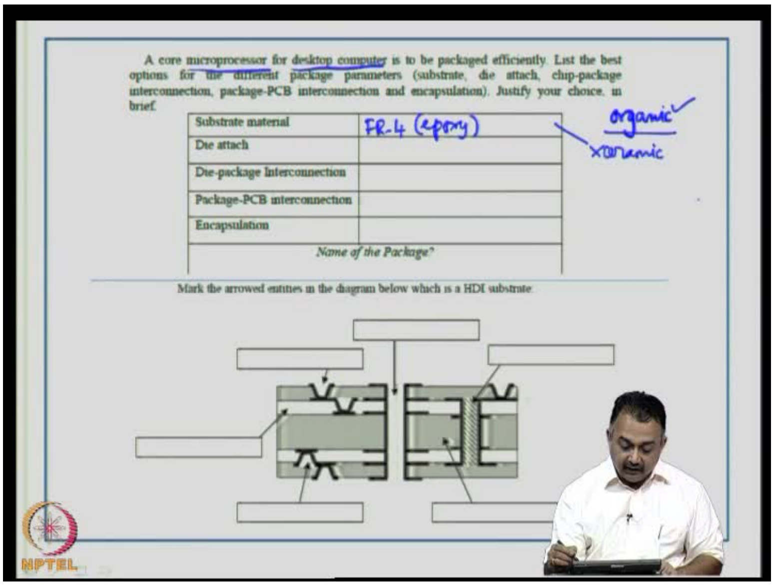
We look at the next problem, it states that there is a core microprocessor for a desktop computer and that has to be packaged efficiently; efficiently in terms of thermal and electrical issues. List the best options for the different package parameters in terms of substrate, die attach type, chipped package interconnection, packaged PCB interconnection and the type of encapsulation that you would like to have. Justify your choice in brief. If you are given such a situation to name or design the package materials for a core microprocessor, then what will you choose what is your choice going to be?

The first thing is that it is a core processor. So it is a very active device, core device for the desktop computer, which means that there will be a mother board, which is fairly large in size and as you know today, the mother boards are mostly organic in type because of cost issues.

The next thing is that if it is an organic substrate, accordingly, you have to choose the package materials. What is a substrate material you will choose for an organic package?

You can go in for FR-4 which is one of the very common organic materials. It is also known as epoxy based material. Because of the cost factor, this particular desktop computer is going to be manufactured in millions, so you have to look at cost. So obviously here you will not go in for ceramic substrate you will go in for an organic substrate.

(Refer Slide Time: 11:57).



Now for the die attach type. If you go in for a cavity-up or a cavity-down approach, you have to look at cost issues. Cavity-up is much less expensive and easy to work with; because you can place it on the surface of the substrate whereas, cavity-down approach will require modifications in the substrate. So you can go in for cavity-up approach.

Die to package interconnection. There are different options for this. The first thing that you will think that it will be cheap is a wire bond. The other one could be a flip chip. You do not want to use tab, because tab will not give you the number of I/Os that you require. So the important thing you have to take care here is the number of I/Os in this particular microprocessor is going to be large. So you have to choose a die to package interconnection where you will get the number I/Os to be large. Wire bond has got limitations therefore, the obvious choice will be using a flip chip because it offers area array type of interconnect.

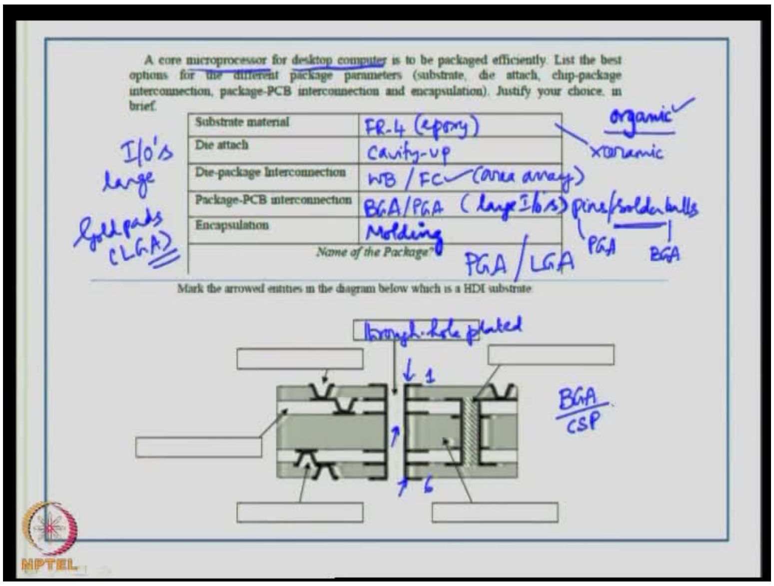
Then you will go in for the package to PCB Interconnection. The choice could be from the flip chip you are going to fan out. You can either use a BGA or a PGA. You can use BGA or PGA, because both of them will give you large number of I/Os. The encapsulation could be by means of molding. You do not want to do chip on board kind of glop top. It will be a regular molding process that you will be doing. Finally, what type of package will you have? This will basically be a pin grid array. There is one more choice that you can opt for looking at current day technologies. It can be a land grid array. So land grid array can also be an area array package but if you look at package to PCB interconnection, it can be either pins or solder balls.

So, depending upon the cost factor you can either go in for pins or solder balls. If you go in for pins, it will be a PGA package. If you go for solder balls, it will be BGA package and if you go in for gold pads then it will be a LGA package. So these are the options that you have. But if you are asked to specifically name one, you can easily go in for a ball grid array type of package because today working with BGA has become well defined.

Now the next question is mark the arrowed entities in the diagram below which is a high density interconnect substrate. I hope you are familiar with this cross section. Let us begin with identifying the sections in this particular figure.

Now this is a multilayer substrate. You can see there is a interconnect through hole happening between the top layer and the bottom layer. This is a through hole and this is plated. So this is a plated through hole which connects by means of plated copper here and here and through the drilled hole. Connectivity is established between the top and bottom layers and these are specifically used in the high density substrate for a BGA or a CSP.

(Refer Slide Time: 16:11).

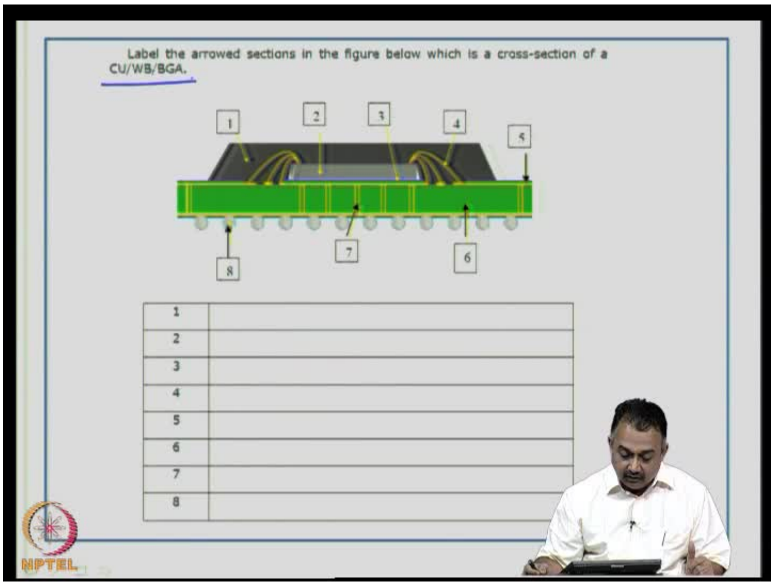


You can see a similar configuration here for this particular hole. But you can see that it is filled with the conductive material therefore, you can call it as a filled through hole that has been generated for interconnecting layers 2 to 5 of the high density interconnect substrate. The 1, 2, 3, 4, 5 and 6 nomenclature are for the copper. Now, the next arrowed section is for this. This is basically a core substrate. The core substrate can be a plastic substrate like an epoxy material and there are variants of plastic organic substrates.

This one is a via structure and here you can see this particular section; this is again a dielectric material that separates two copper layers. This is also a via structure. Now the difference between this via and this via is that this is a blind via because the via is opening from the top and ends at the second layer. Whereas, here, this is a buried via, because after generation of the via it is embedded inside the substrate. Therefore, it is a buried via. These buried and blind vias are very difficult to repair and rework once they are assembled together during fabrication.

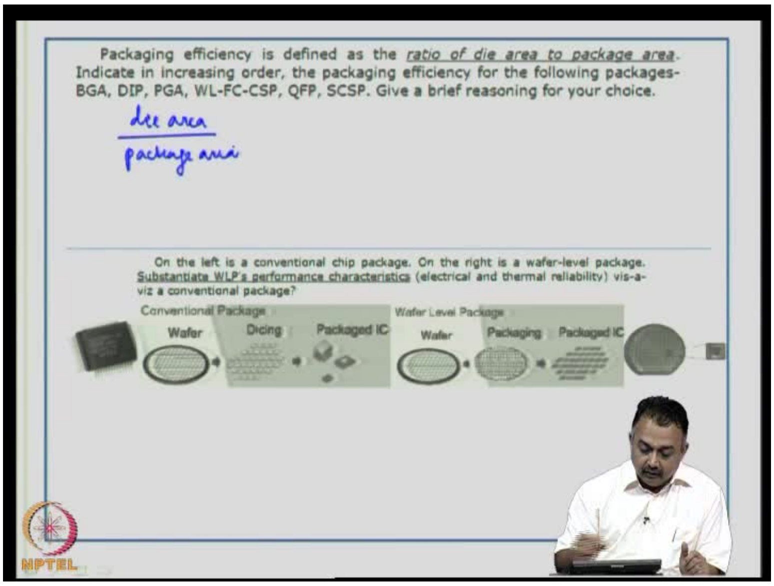
So, this is for a HDI substrate. Now, we look at another cross section of cavity-up wire bonded ball grid array. The package name is given. It is a cavity-up wire bonded ball grid array. All you have to do is name these components in this package.

(Refer Slide Time: 18:30).



So the first one will be item 1 will be encapsulant and the encapsulant is usually made out of epoxy; so you can also call it as a mold. Then next one will be basically, what you are seeing is a Silicon Die. The third one that you are seeing in this figure is the Die attach. It can be a glue or adhesive as you can call it but the general name given is a die attach material. And the fourth one you see is a wire bond. The wire bond can be gold or aluminum as the case may be depending on the situation. Now the fifth one is basically what indicates here in this figure is the interconnect copper on a HDI substrate. The sixth item in this particular figure is the core of a HDI organic substrate. The seventh item here you see is basically the via interconnects or via connect structures that is generated on the HDI substrate it can be a through hole. Then, the eighth one finally is the ball grid array or the solder ball interconnections. This is how you identify the package components. Now, finally, there is a statement here which says packaging efficiency is defined as the ratio of die area to package area.

(Refer Slide Time: 20:55).



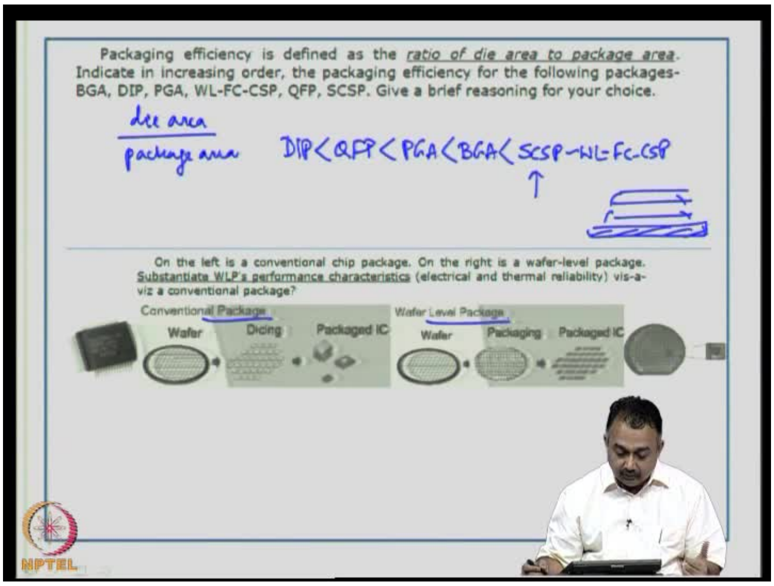
Now can you indicate in increasing order the packaging efficiency for the following packages namely BGA, DIP, PGA wafer level flip chip, CSP, QFP and stacked chip size package?

If you look at packaging efficiency in this particular list, DIP will have the lowest packaging efficiency as you know. Then, comes the QFP packages, which have better efficiency, but definitely less than a PGA and definitely less than a BGA.

And then we talk about stacked chip size package and in some sense almost equal to of wafer level flip chip CSP. Stacked chip size package will probably have the highest packaging efficiency because we are packaging them and interconnecting them on to a common substrate.

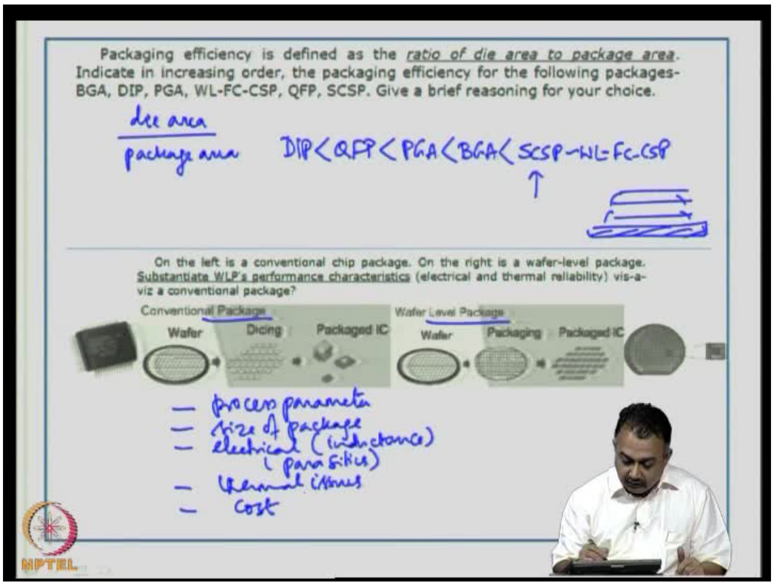
So, if you look at the package name you must be able to identify with the package efficiency. The other question that is very important is you have a conventional chip package and you also have a wafer level package. I have already described the difference between these two.

(Refer Slide Time: 22:23).



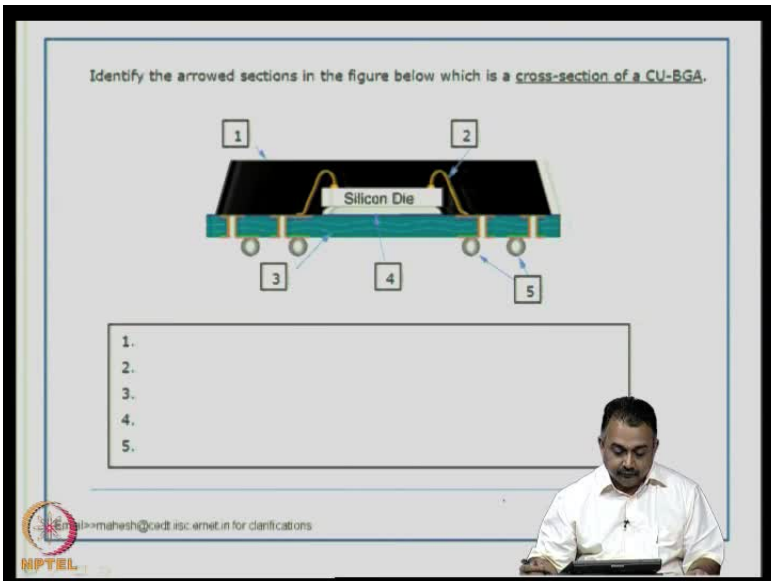
Basically you must be able to give some few important advantages of a wafer level package over a conventional package in terms of manufacturing, in terms of electrical, in terms of thermal issues, and in terms of material issues. So, you can try to answer this question at your own leisurely but keep in mind the issues like process parameter, the size of package, which will be better, which will be lower in size, electrical issues. Talk about inductances and other parasitics; then you talk about thermal issues and so on.

(Refer Slide Time: 23:27).



Finally, cost. So try to answer this question in terms of these points and you will able to substantiate why wafer level packaging is being considered today compared to a conventional package.

(Refer Slide Time: 23:42).

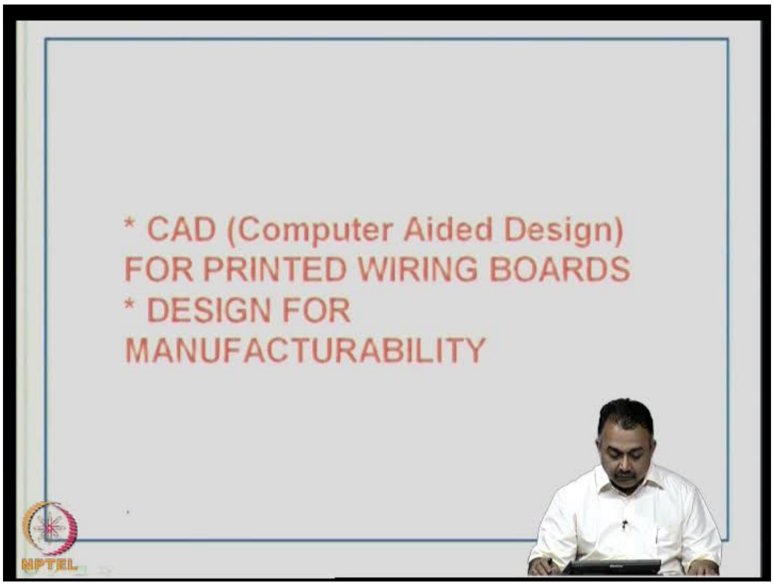


Finally, we have another cross section of a package and we will try to identify the arrowed sections in the figure. This is a cavity-up ball grid array. By now, I think you are familiar with the various part of a component. Finally, we will try to write down once again item number 1 here is a mold plastic epoxy encapsulant.

Then 2 is a wire bond either aluminum or gold. Item number 3 is basically the HDI substrate. Item number 4 is the Die attach and 5 are the BGAs.

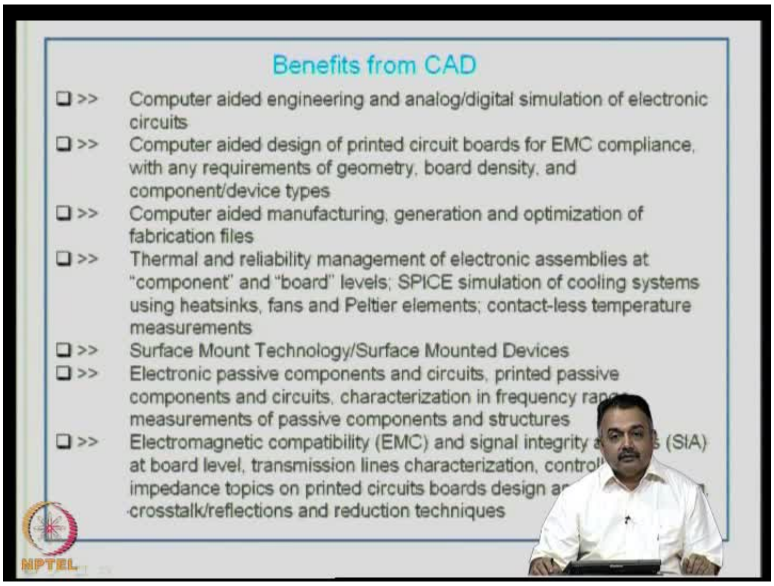
So you can look at the text books that have been referred to and look at the web also for cross section and try to identify the components for various packages we can also email for further clarifications on this packages chapter.

(Refer Slide Time: 24:53).



Now, we will move to this chapter on Computer Aided Design for Printed Wiring Boards and design for manufacturability. Why should we do CAD?

(Refer Slide Time: 25:04).



CAD is very essential to realize your electrical circuits. So, we call it as computer aided engineering and analog and digital simulation of electrical, electronic circuits. Once you have a paper circuit in mind, you have to translate it into CAD because the benefits from CAD are varying very large and more advantageous than just paper based schematic.

The Computer Aided Design of Printed Circuit Boards are especially done for electrical simulation like EMC electromagnetic compatibility compliance with any kind of geometry, board density and any type of component or devices that you may want to choose.

So, a mixed board that is for example, a analog digital board can have varying requirements in terms of analog and digital. In terms of EMC, that is electromagnetic compatibility or electromagnetic interferences. Because the influence of one component over the other and the interconnects that you are using to interconnect these varying components along with the passive devices that you are using can create a mix of problems in terms of electrical and thermal. So your CAD in some sense can really take care of all of these issues if it is analyzed in a systematic way. The benefits from CAD include computer aided manufacturing, generation and optimization of fabrication files because today all manufacturers are well versed with computer aided manufacturing.

Even when they start manufacturing a Printed Wiring Board, the product planning that is the entire planning of the printed circuit board fabrication is atomized. The input data comes from the CAD in terms of the individual PCB size, the total number of PCBs in one panel. Then they are looking at issues like minimum wastage of raw material, optimal usage of chemicals and technologies and minimizing cost.

So computer aided manufacturing can go a long way in interacting with the manufacturers or involving the designers with the manufacturers. When you look at fabrication and if you are going to manufacture thousands of PCBs, cost becomes an issue. Therefore, you will have to optimize your design to get very high yield because today manufacturing multilayer boards and high density interconnect structures are expensive.

You can also use CAD to a good advantage in terms of thermal and reliability management of electronic assemblies at the component and the board level.

You can do SPICE simulation of cooling systems using heat sinks fans and Peltier elements; contact less temperature measurements and so on. Basically what the statement says here is that you can do thermal analysis of your board in addition to the electrical simulation that we were just talking about. There are CAD systems that can get highly integrated with your circuit and look at reliability issues for your board from the thermo mechanical stand point.

You can work with surface mount technology and surface mount devices because the assembly of surface mount devices is largely wholly dependent on the CAD data. This is not true with plated through hole components. This is very much true with surface mount devices because the automatic placement of surface mount devices is dependent on the CAD data generated.

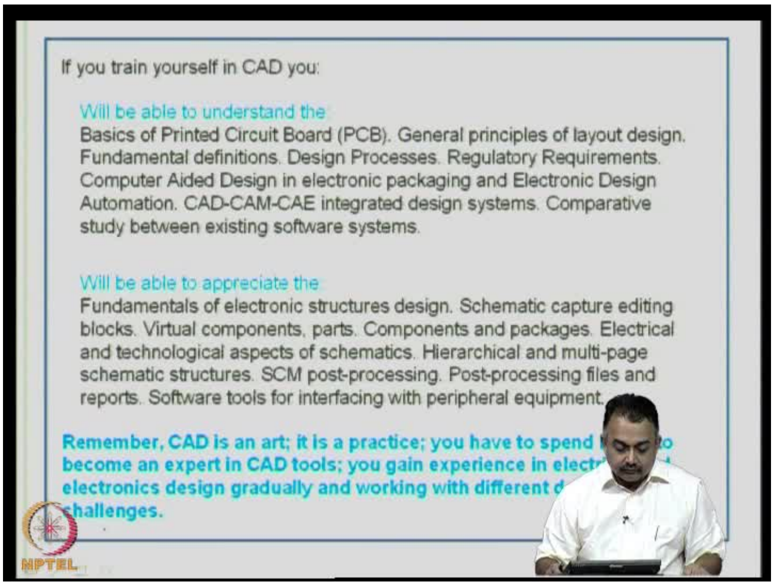
Electronic passive components and circuits, printed passive components and circuits, characterization in frequency, range measurements of passive components and structures can be done with efficient CAD analysis.

Then as I said before EMC, signal integrity is now key word in electronic circuits. Most people do signal integrity to minimize the parasitic effects of a circuit when it is powered.

Then topics like transmission line characterization, controlled impedance related to design PCB design and manufacturing, crosstalk and reflections, and reduction techniques. For example, when you draw two lines on a Printed Circuit Board and let us say this is separated by 4 mils that is 100 microns. Now, you want to realize and understand, what the crosstalk effect is that is generated from this 4 mil line and 4 mill separations. Does it really affect your overall circuit? This kind of analysis can be done.

Now, without doing this analysis, if you send it to manufacturing and if the board is not to work; it is going to be a huge loss and lot of time spent will be useless. Therefore, that is why we say design for manufacturability is very important.

(Refer Slide Time: 30:57).



In another sense for example, if you have designed 4 mil line in your printed circuit board design, but your local manufacturer is not able to manufacture a 4 mil line that is 100 micron. Then, obviously you have to spend a lot of time in redoing your circuit to upgrade the 4 mil line to a 6 mil line or a 8 mil line.

Now, you have to analyze whether for your circuit, is 8 mil line not going to work; then are you going to work with 4 mil line; but then you have to send it to a manufacturer abroad or somewhere else so that you can get the board done but it is going to be expensive.

So, the realization of the actual need for your Printed Circuit Board and the realization of who is going to actually manufacture the board without defects and without change in the final format of your design becomes an essential ingredient in the thought process during your CAD and manufacturing.

If you train yourself in CAD Printed Wiring Board CAD, you will be able to understand the basics of Printed Circuit Board.

As a designer, you must know the process of PCB, you must know what is known as a layout design, layout of Printed Circuit Board design, basic fundamental definitions, design processes that are generally adopted worldwide.

Regulatory requirements that can qualify your board and make it as a leading edge and make it for example, if you want to export your design after manufacturing your design in-house, you have to understand the regulatory requirements in terms of material issues, thickness issues, line width feature sizes, electrical requirements, thermal and so on. You will be able to understand the CAD in electronics packaging as a whole and the EDA Electronic Design Automation tool set.

And there will be obviously when you work with CAD, you will be able to appreciate the difference between various software systems. Compare to study can easy come because if you are going to work in CAD continuously you may have the opportunity to work with 3 to 5 different software at different stages.

I would like to mention here that CAD software, you can migrate from one CAD software to other very easily because the basic principles are the same. The tools and the menu and the commands may be different for different software; but the approach of designing a particular Printed Wiring board design is almost the same. Only thing is some may be low end, some may be high end software and cost obviously is an issue.

If you train yourself in CAD, you will be able to appreciate the fundamentals of electronic structures design, schematic capture editing blocks in a software package, virtual components parts, components and packages, electrical and technological aspects of schematics. The important of schematics right until the end of the completion of the design will be well understood. Because schematic is the starting point but at the same time you can back-annotate to schematic at any point of time during the entire process of your working with the design.

Hierarchical buildup of your various circuit pages into a single project or a schematic can be well understood. Schematic post processing, what you do after the schematic is completed, post processing files and reports, how do you generate them and software tools for interfacing with peripheral equipment.

An example of the peripheral equipment that is used in a Printed Circuit Board fabrication is a photo plotter or your drilling machine or your assembly equipment.

Now I want you to read this statement very carefully, remember CAD is an art. It is the practice that you have to spend a lot of time to become an expert in CAD tools.

As I told you, there are different CAD tools available in market. We are not typically talking about a particular package being advantageous over the other. Each has its own advantages and disadvantages.

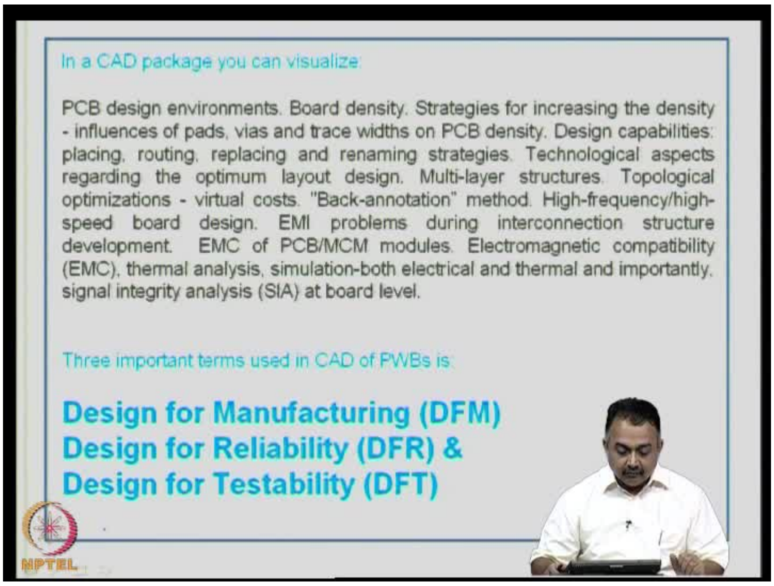
It becomes the prerogative of the company to choose particular software based on some very key issues that the software can do but generally most software can do the same job.

Remember your CAD work will be different from my CAD work. If you and I are going to do the same electronic design conversion into a schematic and then into a layout and then into a finished PCB design, your work and my work will be totally different; because the concept, the understanding and the implementation of each of us will be different and both can be accepted, both are correct because the interpretation can be different.

The basic thing that you have to look at in the final assay is how well the design is going to perform in a particular environment. So, you gain experience in electrical and electronics design gradually. It takes lot of months of hard work and working with different designs and challenges to really say that you become an expert in CAD.

CAD is not a tool that can be taught to you every day. If you choose a software, you have currently online help but some expertise or some advice and guidance can be given initially but it requires yourself to spend lot of time in looking at the advantages of each software to the special features that a CAD can have to capitalize and utilize the software to the maximum.

(Refer Slide Time: 37:39).



Generally, in a CAD package you can visualize the following things PCB design environments can vary. So, you will be able to understand that you will know the terms like board density. You will understand strategies for increasing the density of boards.

How does the density of board play an important role because today you are looking at the Printed Wiring Board as a system by itself and if you load packages like BGA and CSP or a flip chip, obviously your board density has to be high and the board material and the design have to play an important role, because you do not want to loose the electrical performance of the package when it is mounted on the board.

Therefore parameters like pad sizes, via sizes, and trace widths on the Printed Wiring Board will become or will have a very telling influence on the total board density.

Your CAD package will at any point of time give you the board density parameters. You can easily find out what is the pin density, what is the component density, what is the space utilization that you have done and so on. Design capabilities you can master. That is, placement, routing, replacing and renaming strategies of components, choosing the right foot prints of components for a particular schematic symbol and so on.

You will able to you will be able to understand the technological aspects regarding the optimum layout design, multilayer layer structures, topological optimizations, virtual cost, Back-annotation from a layout to a schematic because you went to make the schematic perfect.

High frequency and high speed board design today is very important. EMI problems during the interconnection structure development, EMC of PCB or multichip modules, EMC thermal analysis, electrical and thermal issues and importantly signal integrity at the board level can be well understood.

I have now listed here three different terms that are very much part of the PCB design culture today and most people need to be aware of it. The first thing is design for manufacturing which I briefly mentioned the second one is design for reliability and the third one is design for testability.

Now, if you look at the Printed Wiring Board that you have designed; it is going to be manufactured by somebody whom you really do not know and then it ends up as a very bad manufactured product.

Now, if you look and analyze the design, you have made some errors in the design and that has got reflected in the manufacturing. The manufacturer did not tell you why he is not able to do certain features because he has got limitations in the technologies that he has. Therefore, it is a DFM problem similar to the line width example that I have mentioned. There are various such examples that you can cite, where a designer, if he has not consulted with the manufacturer during the process of design; then your time is wasted, your money is wasted.

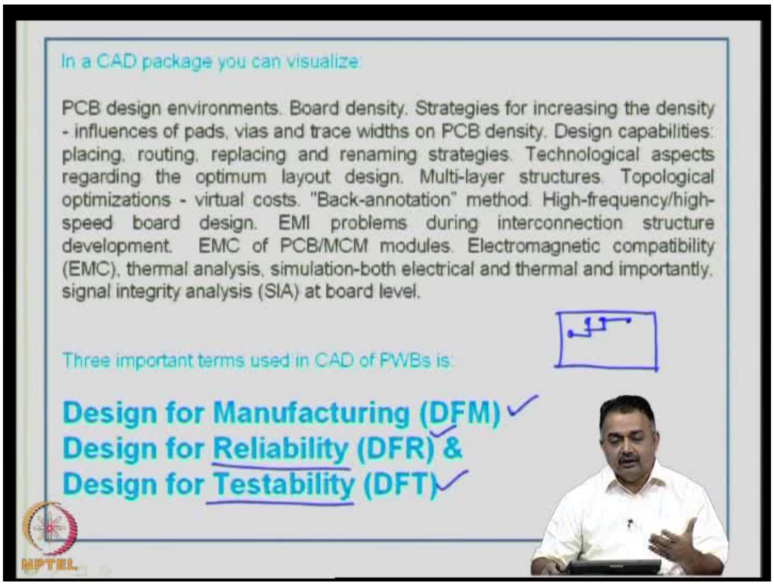
If you look at a particular project for a Printed Circuit Board or the end product like a hand held product, 70 percent of your time is spent on the design and 20 percent is spent on the manufacturing and just about 10 percent is spent on assembly and testing.

So you can look at large time frame or time spent on the design issues. If you can take more care to really analyze and make sure that your system or product is 100 percent efficient, then you have to look at various angles in your design including manufacturing feasibility.

Then reliability issues; when you talk about reliability, it is not just the working of the PCB, it is a entire system that you are worried about. In that case, you have to look at various materials that you have used. If you choose a particular plastic BGA, you have to make sure that the plastic BGA is really suited for that application. If it is for an automobile application, why cannot you choose a ceramic BGA instead of a plastic BGA? Will it enhance the life of the product? Questions like that can be asked and it is true for every component and every layer built in the Printed Wiring Board.

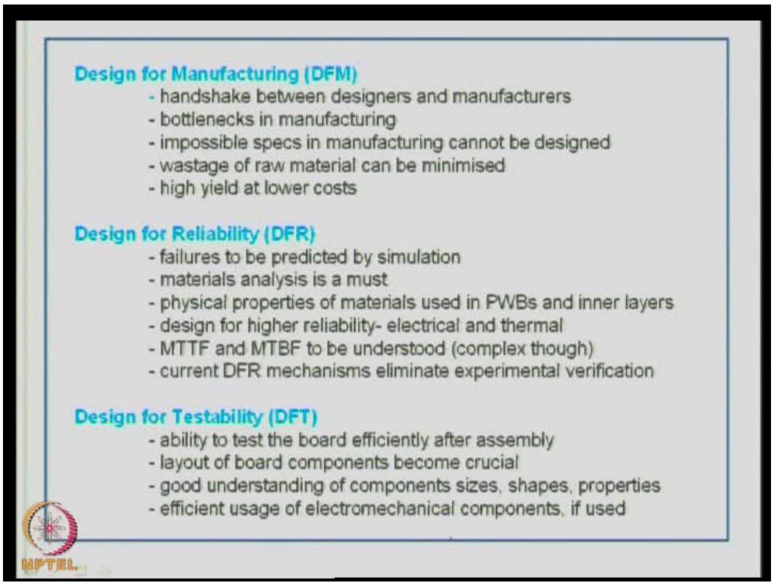
Then, the other issue is, you have used a set of components on the Printed Wiring Board; now can you test it? If there is going to be a requirement for testing your board regularly, do you have the components in the right place so that you can test the device, test the board, test the system regularly.

(Refer Slide Time: 43:10).



If you have a very bad placement, a very bad design done where you cannot test and offer correct advice to your customers. On testing then it is a very poor design. So what we say is at the design stage itself; look at manufacturing, look at reliability, look at testability, then your design automatically becomes a top level design that can compete with the global scenario.

(Refer Slide Time: 43:42)



Now some of the important issues that you have to think about in DFM will be, there should be a constant hand shake between the designer and the manufacturer.

Before you really finalize your project, bottlenecks are there in manufacturing. Therefore, you have to consider those when you design when you assign a particular pad. For example, if you want to create a 40 mil pad of a particular shape then make sure that your manufacturer is able to do that.

Now another example I can give is, you want to make a 0.4 mm PCB, a very thin PCB. There is an electrical requirement for that but make sure a manufacturer has the capability to do that otherwise he will end up with doing a 0.8 mm or a 0.6 mm thick PCB which might not work for your particular application.

Impossible specs in manufacturing cannot be designed or in other way unnecessary designs; what you call feature sizes that you have used in design cannot be manufactured in some cases. Therefore, there has to be a very good via media between the designer and the manufacturer.

And in some cases fancy designs which are really not required for a particular system can be avoided.

Now, a better design can help the manufacturer in terms of reducing wastage in the starting material like your PCB organic substrate or other materials.

So, especially copper wastage can be reduced and also the epoxy or the FR 4, very common substrate that is used today. You can minimize the wastage of these raw materials and in any business you want high yield at lower cost.

The other aspect design for reliability; some of the issues that you have to consider will be, failures have to be predicted by simulation. So if you have spent lot of time in your schematic, you can spend more time on the simulation part of it. The thermal as well as the electrical simulation will qualify your board so that the reliability becomes high.

Material analysis is a must. So, make sure you are choosing the right material. The physical properties of material used in Printed Wiring Boards and inner layers in a multilayer board or a sequential buildup technology board have to be very carefully considered including thicknesses and so on.

Design always for higher reliability. Mean time to failure and mean time between failures are very important reliability terms. You have to spend some time to understand what it means by mean time to failure and mean time between failures for a product. Your board is also a system product. It has to be understood. So, at least if you are chosen the right materials from the right process steps, these reliability terms can have some meaning. It can be understood.

Now, people are looking at today using design for reliability mechanisms to eliminate completely the experimental verification; because, when you do a PCB obviously you are going to do some kind of a thermal cycling.

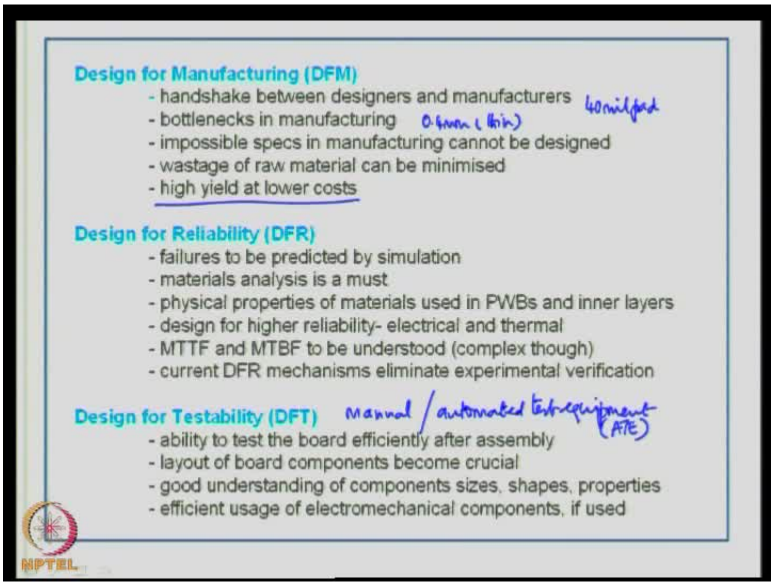
Let us say you are making a Printed Wiring Board and the board is subject to thermal analysis experimentally. That is you keep it in a an oven, you thermally cycle it, you raise the temperature from room temperature to 125 degree centigrade and the entire cycle goes from at 5 degree centigrade or 10 degree centigrade per minute rate of heating and then it goes for 100 cycles, 200 cycles and so on.

And you take out the board at regular intervals and look at the fault physically. You look at various defects. Now, what people are trying to do today is, at the design stage itself, if you have chosen the right material, you can expect failures, you can minimize failures and in some cases you do not even have to do experimental verification.

The third point is designed for testability DFT. In this, you have the ability to test the board efficiency after assembly especially after assembly. The layout of board components becomes crucial. Good understanding of components, sizes, shapes, and properties. Here, we are not talking only about manual testing; we are also talking about automated test equipment.

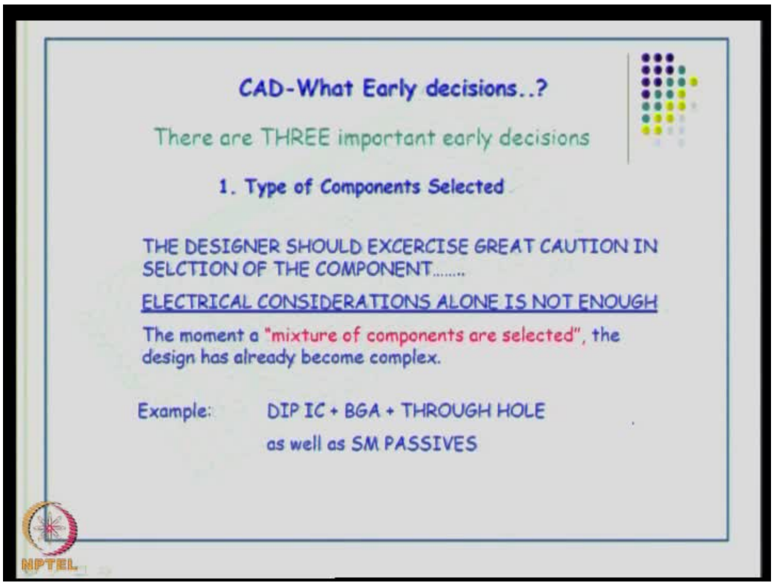
Today some people do manual testing, some people do automated test for assembled boards. In automated test, people use x-ray. In some cases people use acoustic microscopy; in some cases other methods to use or look at the various components if there is a failure or after some thermal cycling or after certain exposures of this Printed Circuit Boards.

(Refer Slide Time: 49:49)



So, if you want to use equipment then you have to do your layout of the critical devices very carefully; so that you can have access to these components from the equipments or manually. Then, efficient usage of electromechanical components if they are used, because you cannot clutter the board with large electromechanical components and if it becomes too difficult to test it; then you end up with lot of trouble in testing. So when you look at CAD as a designer, what are the early decisions that you have to take?

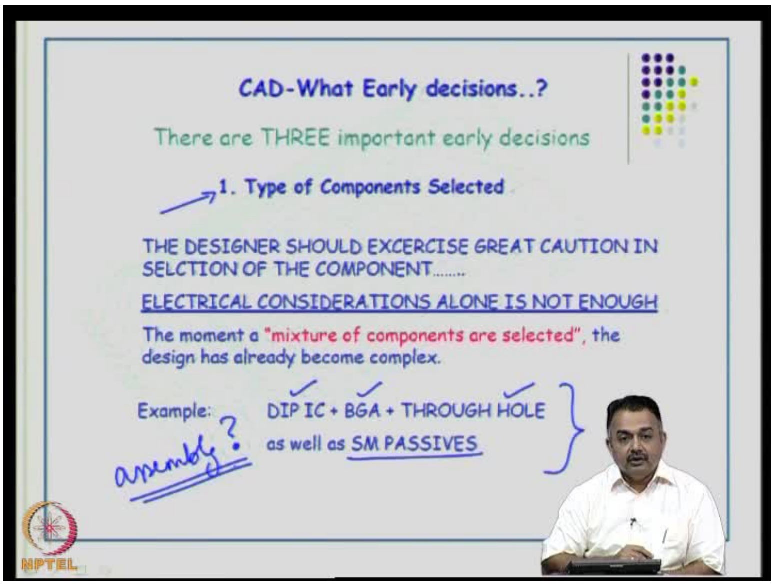
(Refer Slide Time: 50:31)



There are 3 important early decisions that you have to take. The first thing is the type of components that you have selected the designer, I consider you basically as a designer, you should exercise great caution in selecting the component.

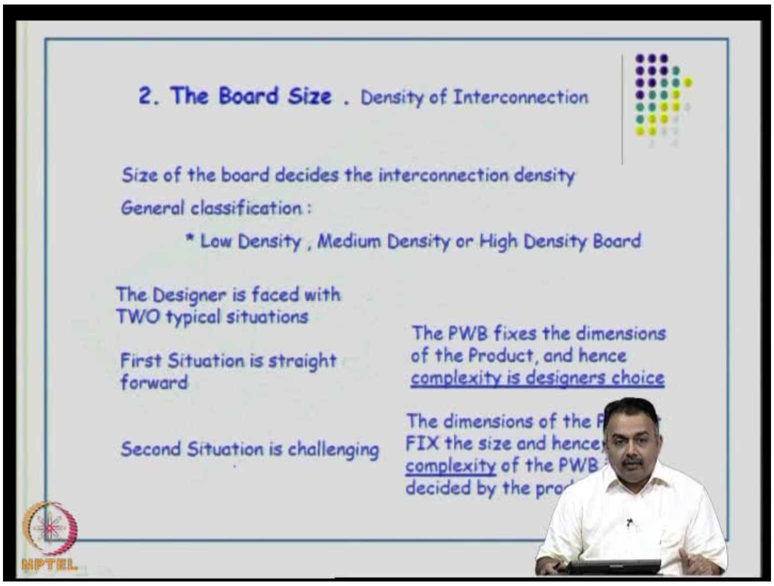
Electrical considerations alone are not enough. The moment a mixture of components are selected, the design has already become complex. For example, if you have a DIP IC that is dual inline package integrated circuit BGA; and through hole devices, other through hole devices, and then surface mount passives devices, so you have such a mixture of components then how is your assembly going to be done? This is a first question that you should ask when you are designing the board.

(Refer Slide Time: 51:27)



So when you start designing, you just not look at the PCB alone. After PCB is fabricated, you look at the board assembly. Only then your design is complete. So, this is a very typical situation. In some cases, you will have a complete through hole assembly; in some cases you will have a complete surface mount device assembly and in some cases you will have a mixture of components. Therefore, why assembly is a problem because, the board will be subjected to various assembly processes because of the different technologies involved and therefore, thermal shocks will be a major deterrent in the final assay of the product. So, during the completion of the entire process, your board can experience thermal shock and give failures; so you need to avoid that.

(Refer Slide Time: 52:28)

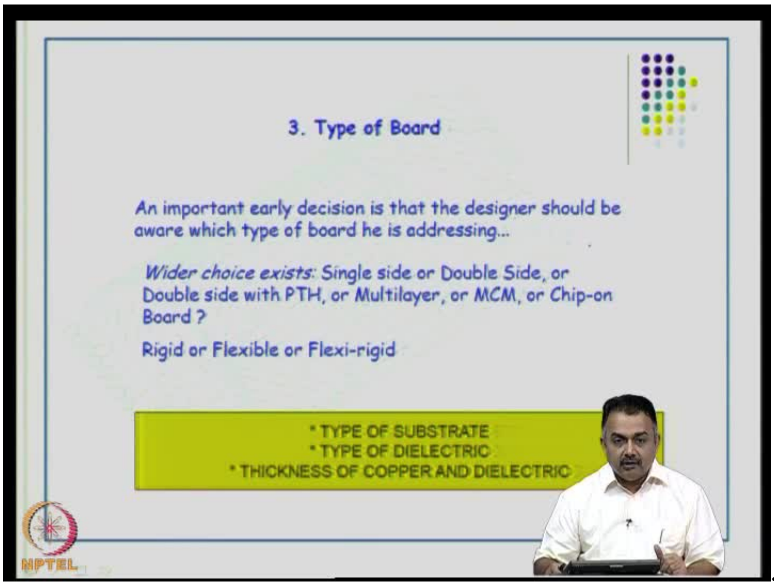


Then the second point is the board size or in some sense the density of interconnects. The size of the board will decide on the interconnection density and the number of components also will decide the interconnection density and the component density and therefore the difficultly in routing these interconnects. So, general classification is available known as low density boards, medium density boards, or high density boards. This is very general term.

Now, the designer is faced with two typical situations in designing a board on its way to a complete product, fabrication. The first situation is the Printed Wiring Board will fix the dimension of the product and hence the complexity is designer’s choice. So the product dimensions are not really understood yet. The PWB size will be a key factor in deciding the final product size.

The second point is the dimensions of the product is fixed and hence the complexity of the PWB is decided by the product itself and therefore, the designer will have much more job to do in fitting the entire circuitry on a given PCB size. Maximum PCB size is defined by the product itself. So, this becomes more complex. In the other case, you have more flexibility. So, these are two common situations and probably the only two situations that electrical designer will face when you are designing a Printed Wiring Board.

(Refer Slide Time: 54:20)



And the third point is the type of board that you are going to choose for this particular application. Important early decision is that the designer should be able to judge what type of board he is addressing for a particular product; whether it is a single sided board or a double sided board or a double sided with plated through hole interconnections or is it going to be a multilayer or a multichip module or is it going to be a chip on board, is it going to be a rigid board or flexible board or is it going to be a rigid flex combination?

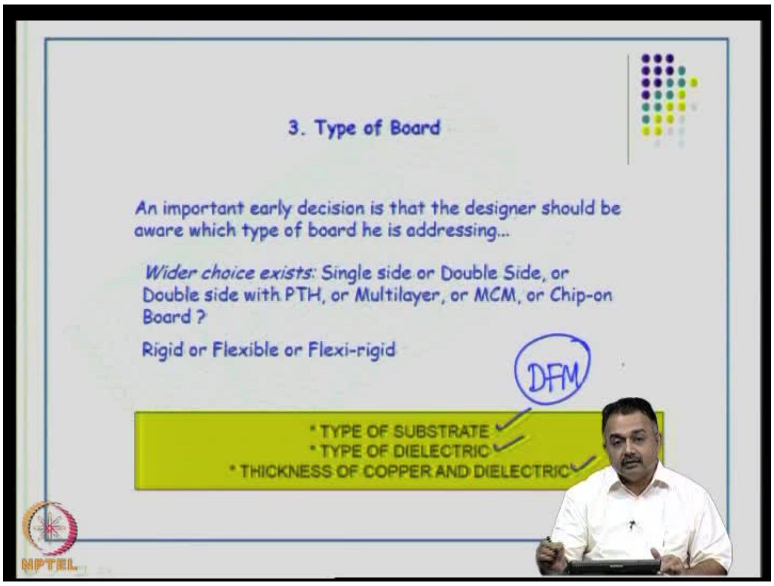
So, to decide on this, you have to look at the final product. You have to look at the functionality. The number of devices that are required, the number of interconnects that are required, the electrical issues that are required, whether you need to have a single or multilayer to separate the ground planes from the signal, how many ground planes you require and so on.

Any special issues like isolation pads, isolation areas, special requirement like large electromechanical components, RF, analog, digital, high bandwidth. So, you will have different requirements for a particular product. Accordingly, if you want to decide on the type of board, you have to choose the right type of substrate, the right type of dielectric material, and the thickness of copper in the dielectric that you are going to use is the designers' choice it is not the manufacturers' choice.

Many people think that the fabricator is choosing whatever material he has; which is not correct. The designer will have to specify what type of board you have to make including the thickness of the dielectric, the thickness of copper, the thickness of the total substrate, and the thickness of the plated conductor and so on.

So, a designer apart from understanding the basic core electrical issues will also need to know manufacturing issues. That is why I say, if you look at this point, DFM becomes very important.

(Refer Slide Time: 56:37)



So we will continue with this chapter on CAD and other issues in the next class.