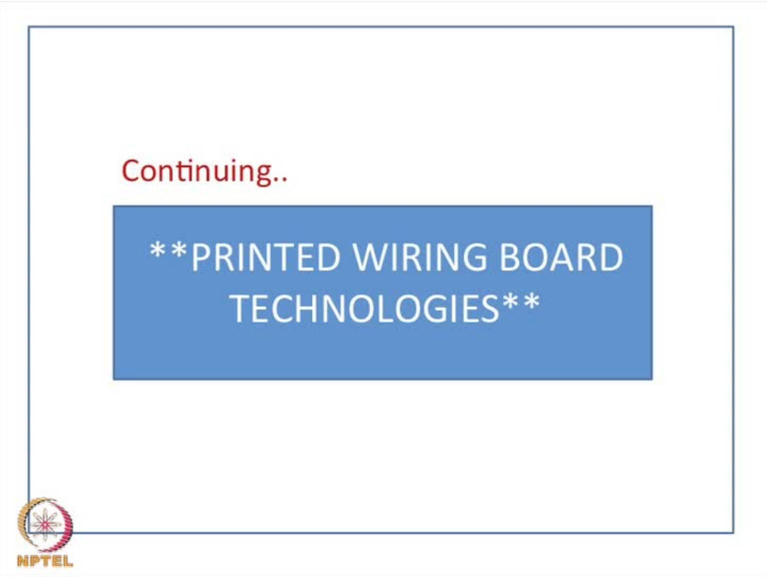
An Introduction to Electronics Systems Packaging

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

Module No. # 06
Lecture No. # 31
Conventional Vs HDI Technologies
Flexible Circuits
Tutorial Session

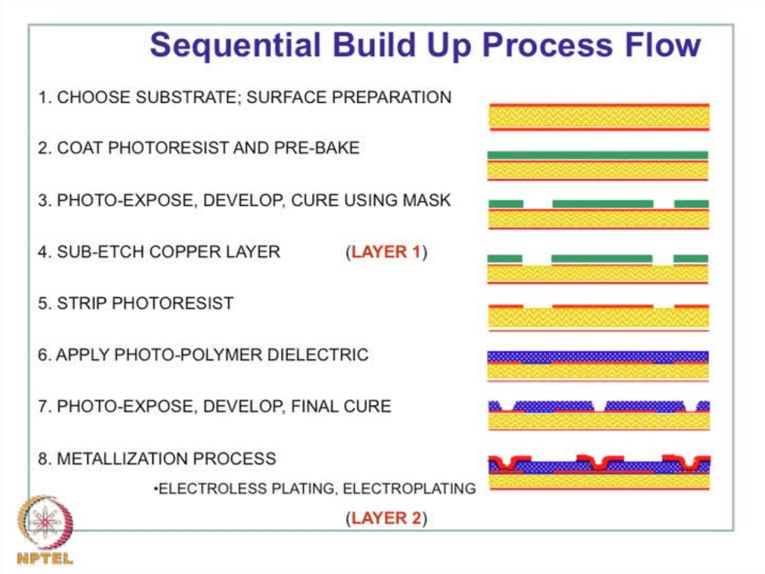
We will continue with the chapter and the module on Printed Wiring Board Technologies.

(Refer Slide Time: 00:15)



Remember, we are looking at all the basic technologies of system level Printed Wiring Board manufacture and the advanced technologies like the High Density Interconnect technologies.

(Refer Slide Time: 00:33)



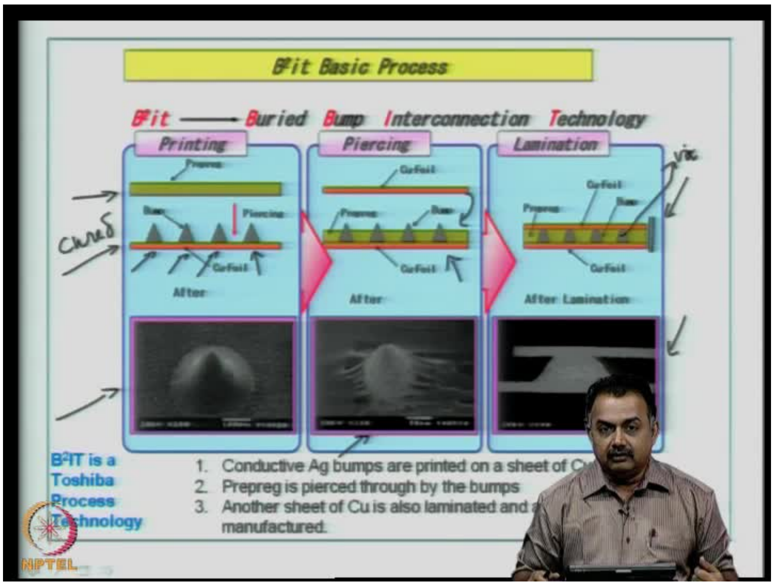
In the earlier classes, we have seen essentially, what an HDI means and what are the roots to manufacture an HDI board.

(Refer Slide Time: 00:46)

Process	Company	Material	Lines/ Spaces	Via/ Land	Via Process Diameter
DV-Multi	NEC	Epoxy Film	80-50/ 80-50 µm	100/250 µm	Photo
IBSS	Ibiden	Epoxy Film	75-50/ 75-50 µm	150-100/ 250-150 µm	Photo
ALIVH	Matsushita	Aramid Epoxy	60/90	150/300 µm	Laser
PERL (Plasma Etched Rdistribution Layers)	Hewlett-Packard HADCO Worldwiser	Epoxy film PI/ Aramid	75/50 µm and 75/50 µm	125-90/250-165 µm>	Plasma Plasma/Laser
Build-up Substrate	Fujitsu	Epoxy	40/40 µm	90/140 µm	Photo
VB-2	Victor	Epoxy	10-95/100-75 µm	200-100/µm	Photo
B²IT	Toshiba	BT Laminate	90/90 µm	200/300 µm	Paste/Bump
Multi-Layer Build-Up	Shinko	Multiple	40/40 µm	50/110 µm	Laser/Photo
SLC (Surface Laminar Circuit)	IBM Yasu	Epoxy Liquid	75/50 µm anD	125-90/250-165 µm	Photo via
Hitavia	Hitachi		100/100	200/500	
Viathin	Sheldahl	PI	50/37.5 µm	60-25/140-75 µm & 85/50/200-165 µm	Laser
ViaPly	CTS	PI/Aramid	75/75 µm	125/125 µm	Photo
TLPS	Ormet	PI	50/50 µm	25/200 µm	Photo
DYCOstrate	Dyconex	PI	100/125 µm	75/300 µm	Plasma

We have also seen some of the process steps to fabricate a High Density Interconnect system level Printed Wiring Board and we have seen what a microvia is. So, we will continue with those aspects and look at some of the advanced features.

(Refer Slide Time: 01:03)



Today, we are going to talk about a High Density Interconnect process that is called B squared I T process; otherwise, known as Buried Bump Interconnection Technology; B squared I T that is the acronym used; this technology is from Toshiba. This is a trade mark of Toshiba Company. After we describe the process, you can see that this is a method to sequentially build up the circuit. Now, all the advanced technologies that we are describing here in the HDI process is one way of bringing some kind of a novelty into creating a microvia. This microvia structure is what enables a High Density Interconnect structure because you are able to reduce the feature sizes of the conductor lines, increase the number of layers to an optimum of let us say – 6 to 8 layers, and then provide a lot of fanout area for packages like a BGA and CSP to be mounted on these kind of system level structures so that you get a high performance. Essentially, you will see that these technologies are ideally used for handheld products.

Let us briefly describe the B squared I T process – Buried Bump Interconnection Technology. The first step on the left side, you see that there is a picture of the silver bump (Refer Slide Time: 02:51). Silver is a conductive material probably mixed in an epoxy media. So, silver paste can be used with suitable thixotropic conditions.

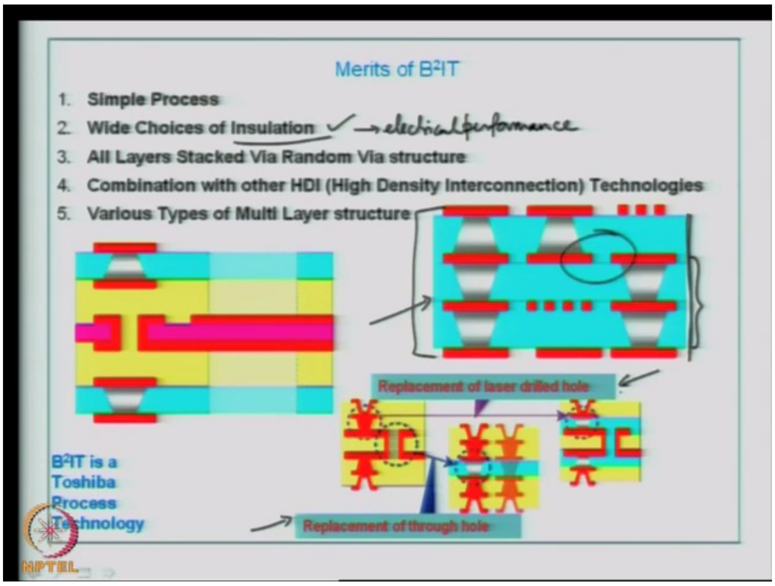
If you look at this illustration here, a copper foil is taken; thin copper foil is taken. Now, using stencil printing and based on the microvia design that you have generated for that particular layer, a series of bumps have been created as you can see here (Refer Slide

Time: 03:29). These represents some kind of conical structures. The material is so formulated that after stencil printing and based on the height criteria that you give between the stencil, the stroke pressure, and the base; that is, the copper foil, you are able to produce some kind of a structure that is very sharp. The reason for having this sharp top edged structure is that the next step what you are going to do is introduce the prepreg material. All of you know what a prepreg material is; it is an insulator material that is used as a sandwich between two conductor layers. Here, you are going to introduce the prepreg and you are going to pierce through the structures that have been printed. Remember, these structures will be cured so that they are held in place. The prepreg is now pierced through the bumps.

The next picture you will see here (Refer Slide Time: 04:48) – this is a scanning electron micrograph that you see here. Again, what you see here (Refer Slide Time: 04:54) is a picture after the oven prepreg material has been pierced through the bump. One of the requirements after the stencil printing process is over is that, you require some kind of a bump structure that can enable insertion of these prepregs up to the base of the copper foil. Now, this picture you see here (Refer Slide Time: 05:19) is representing the completion of the piercing of the prepreg. Now, we introduce the copper foil; that is, the next electrical layer is now introduced to the same location. These are again mechanically forced to provide contact with the prepreg material by piercing the copper foil again. So, two process steps have taken place: one is piercing the prepreg material; there should be no air gap of course, during lamination, or all air gaps are removed during the curing process because the epoxy flows. The copper foil is also introduced.

Here, you can see this particular micrograph (Refer Slide Time: 06:11) as well as in this cross section that a laminated structure has been built. Once the copper foil is introduced, you have a sandwich structure and you also have the via structures, but the advantage of this technology as you can see here is – there is no plating required; vias are formed in situ by printing process and this is a filled via. So, these via structures are very rigid; they do not cave in during the curing process or during the subsequent build up processes. So, what Toshiba has introduced in the form of B squared I T process is that you are able to prepare these kind of microvia structures using printing and simple copper foil prepreg methodology just as you would have done in conventional multilayer boards. So, a double sided B squared I T board is ready. Now, you can prepare another set of B squared I T process and like that build the structures and then interconnect these structures as your design permits.

(Refer Slide Time: 07:31)

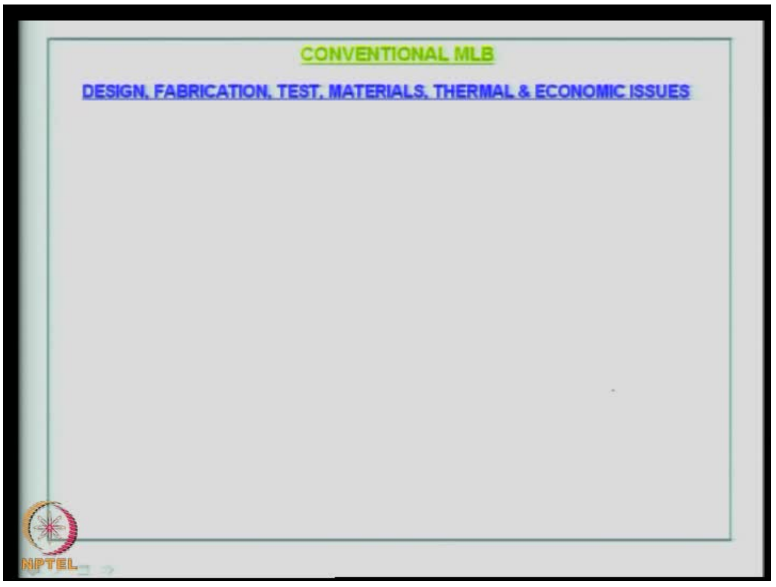


The merits of B squared I T is that it is a simple process; it has got wide choices of insulation because instead of the prepreg material for example, if it is an epoxy, you can use an epoxy, or if you get other materials to choose from, it can be a polyimide, it can be a Teflon sheet, and so on. So, based on the electrical performance requirement, you can choose your insulator material. All layers are stacked via the random via structure. Basically, you will see if you have a 6 layer or an 8 layer B squared I T board. The basic interconnection is by filled microvia structures. You can even stack them as you can see here (Refer Slide Time: 08:24) – it can be staggered like this, interconnected in the inner layers, and it can be a small stack like this or it can be a complete stack starting from the top layer to the bottom layer. So, you have a lot of flexibility in your microvia design or your interconnect design.

It can combine well with other HDI processes. Basically, it can merge well with your PCB processes because if you want to plate at the top and the bottom, add more copper and do the imaging processes, or if you have space available in the structure, you can even interconnect through a final through hole if you desire. If you are not very sure about the reliability of stacked microvias, you can introduce technologies that we use in conventional multilayer boards by plating and drilling, but the advantages of using B squared I T will be lost if you are again introducing a mechanically drilled structure.

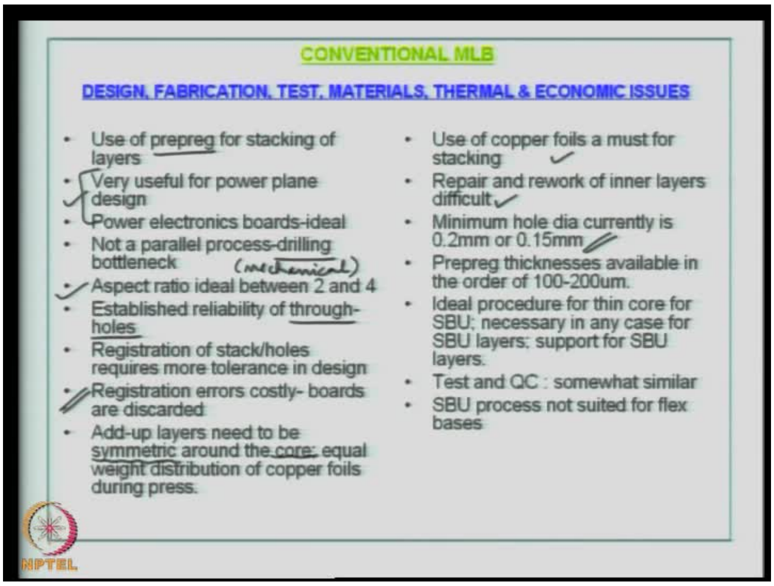
The focus or the merit of B squared I T is that, it presents a problem or it presents a solution to a problem of laser drilling and mechanical drilling. So, you can eliminate laser drilling and you can even eliminate mechanical drilling. So, B squared I T is a (Refer Slide Time: 09:53) replacement for a through hole and it is a replacement for a laser drilled hole. So, we are able to design high performance high end multilayer structures. This is a system level structure that we are building that can accommodate flip chip BGA, CSP, QFN packages, and so on. So, B squared I T is very ideal for handheld products where there is not too much of a power dissipation that you can expect and the sizes are typically fairly small enough so that the construction is of high yield.

(Refer Slide Time: 10:41)



That completes three or four techniques that I have mentioned in the High Density Interconnect category. There are various other techniques available in the market today – all are variance by changing the material or changing the curing process or adding a new step in the conventional multilayer build up. So, one needs to really look at what technology one can adapt for a high throughput in manufacturing, but the base point is that all of these processes – the SBU process will use a microvia either laser drilled or photovia technology. At the same time, it should be sequentially built up for High Density Interconnect performances.

(Refer Slide Time: 11:44)



Now, we will look at a summary of the conventional multilayer boards that we have discussed over the few hours. We will look at the salient points in terms of design, fabrication, electrical test, materials, thermal, and economic issues. What I am now presenting is a complete summary for you to look at. This is like an executive summary that you can look at; look at the basic advantages and the disadvantages of conventional multilayer boards. We will come to the SBU technology once we discuss the conventional boards.

Let us look briefly at some of the points. In the conventional multilayer board technology, you have to use a prepreg material for stacking the number of layers that you require in the design. This is very useful for power plane design. It would be sort of unwise if you look at SBU technology completely for a board that is power design based and that is emanating a lot of heat, and where you have to tackle heat issues to a very large extent. So, conventional multilayer boards will still be the order of the day if you are looking at high percentage power plane design. Ideal for power electronics boards. So, these two (Refer Slide Time: 13:09) would go in one category.

It is not a parallel process. Every panel has to be processed individually and quality checked. In this, the drilling – the mechanical drilling is a major bottle neck. So, what we have discussed so far is to see how we can replace mechanical drilling with the options that we have seen.

Now, in a conventional multilayer board, you have to look at aspect ratio ideally between 2 and 4. So, if you are a designer and if you are going to fabricate a board, or design a board that is to be fabricated using conventional multilayer boards, then look at aspect ratio and keep it between 2 and 4.

So, we have discussed enough about aspect ratio. Established reliability of through holes. We have seen how a through hole is manufactured, plated and quality checked. The key to your conventional board is the reliability of the through hole because sometimes if the aspect ratio is high, you cannot really guarantee the reliability of a through hole. Through hole means you are mounting certain through-hole components apart from the surface mount devices; sometimes you will have a hybrid board. Therefore, through holes apart from basically interconnecting and not providing component mounting, will also have some cases of where the through hole is required for component mounting. So, reliability there is also to be well defined.

The registration of stacked layers / the holes requires more tolerance in design because as you know, there are guidances based on IPC military etcetera in manufacturing as well as in materials, but you have to have a good understanding about the tolerance levels dimensions and accordingly you have to incorporate it in your design.

Registration errors will be more costly. In some cases, boards are discarded. So, make sure as a designer, you understand in a multilayer board design, what kind of tolerances you have given especially if you are drilling for example, layer 1 to 8 and if you finally, see one of the inner layers is not connected at all. So, this to me (Refer Slide Time: 15:47) is very important both from the design point of view and manufacturing point of view.

Add-up layers need to be symmetric around the core, which is what we have emphasized in conventional multilayer boards. If you take a 4 layer board, it means there is a 2 layer core and 2 layers are added – one at the top and one at the bottom, which need not be the case in a SBU/HDI process, but in a conventional board, it has to be symmetric (Refer Slide Time: 16:19) around the core. The core is usually thick to maintain the rigidity. The symmetric requirement of adding one layer at the top and one layer at the bottom in your design is basically because, you want to provide equal wait distribution of copper foils during the multilayer press and also, it enables maintaining the rigidity of the board, maintaining the planarity; otherwise, you will end up with the warped board. So, warpage is a major issue after fabrication.

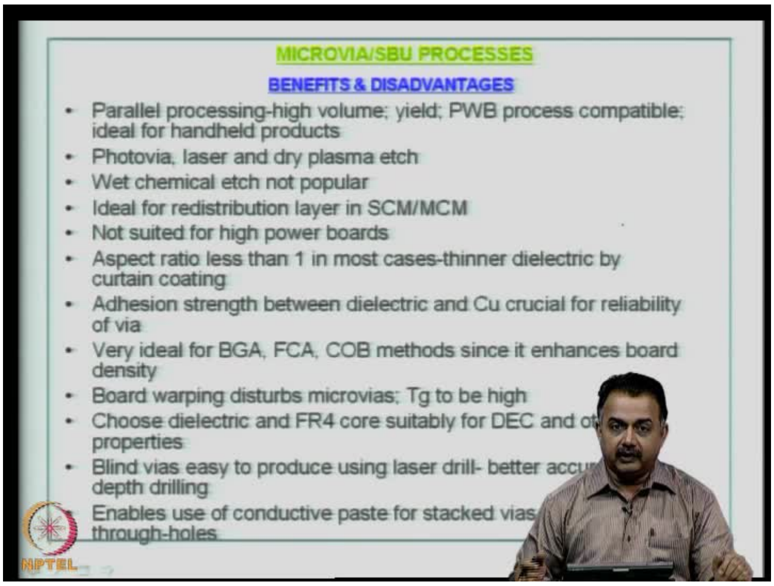
Use of copper foils is a must for stacking (Refer Slide Time: 17:00). So, you will use a prepreg as well as copper foils. Repair and rework of inner layers is difficult. So, whenever we have seen at least a couple of methods to fabricate the conventional multilayer boards, the inner layers have to be immediately quality checked for electrical shorts and opens as well as any process defects on the surface. Isolation of tracks is very important and you have to make sure that these are maintained once the number of layers are stacked in a multilayer press. So, we want to maintain the originality of the inner layers in terms of electrical performance as well as dimensional stability.

The minimum hole dia that is used currently in conventional multilayer boards is typically 0.2 mm or 200 microns, but as I have briefed earlier, there are a few mechanical drilling machines available today with a few industries that can drill 0.15mm mechanically drilled holes. However, you can imagine drilling such hole and wetting the hole and plating the inside of the hole wall is definitely a challenging task as also a costly process. So, you have to pay a premium if you have a design, which contains mechanically drilled holes with these dimensions.

Prepreg thickness is available in the order of 100 to 200 microns. So, you can vary the thicknesses in the buildup according to your design. This is an ideal procedure for thin core for SBU. So, you can use a conventional board technique for preparing the first two layers of your SBU board. If you recollect, SBU circuits can be built up from a core structure, but the essential requirement is that you need to build a thin core using a conventional technique. Necessary in case for SBU layers and this is a support for SBU layers.

Test and quality control is somewhat similar between all the board processes that we have seen. SBU process is not suited for flex bases. So, when we come to SBU technique and a flex circuit technology, then you have different choices.

(Refer Slide Time: 19:36)



We will now look at the Microvia/SBU processes and look at the benefits and disadvantages. This is a parallel processing-high volume; yield is fairly high; the process is compatible with the PWB processes that we have seen so far; ideal for handheld products. Typically, one would not use a microvia structure or a board that has got very large footprint area and for example, to be used in a server board and so on. So, ideally used for handheld products, mobile applications, PDAs, iPads, and so on.

Photovia laser and dry plasma etch are some of the methods used to realize a microvia structure. Wet chemical etch is not very popular because if you look at the various drilling techniques for microvia, we are worried about the quality of the hole; we are worried about the undercut. So, please look at these issues when we think about microvia drilling; ideal for redistribution in a single chip module or a multi-chip module. As you know, if you want to create a BGA, which is a single chip module with a bare chip, you try to create a substrate, which is of 4 layer or a 6 layer basically to fanout the bumps from the bare die through the structure of the organic substrate, and then end up in a BGA. So, this is what we call as a redistribution layer in a single chip module.

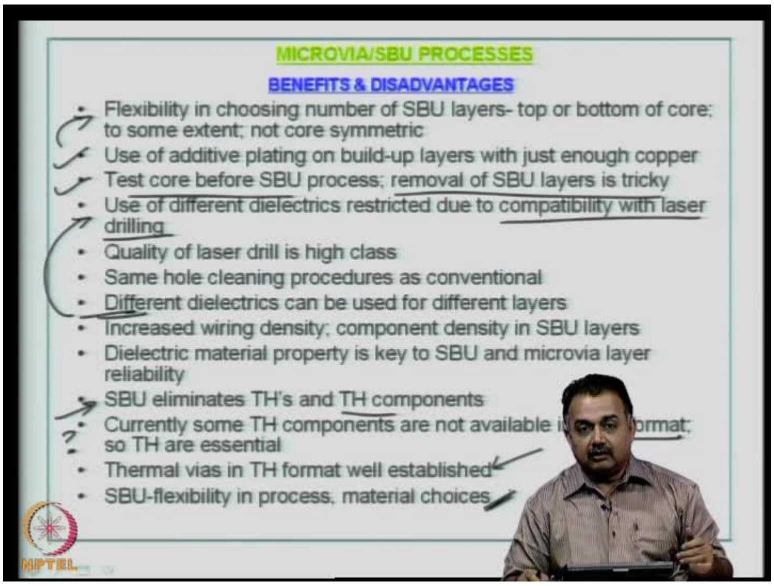
Not suited for high power boards. Aspect ratio; if you actually look at the thickness of the dielectric that we use in the SBU and the size of the microvia that we create in the thin dielectric, the aspect ratio is less than 1. If we use curtain coating, which can be used to reproduce very small thicknesses, our aspect ratios are fairly achievable. We can get very good yield using these processes.

Adhesion strength between dielectric and the copper is crucial for reliability of a microvia because whenever you create a microvia and then try to do an electroless plating to wet the microvia and establish connection between the top and the bottom layer, you are defining the reliability of the board as such. This is because, that is where the crucial functionality in terms of adhesion strength, the peel strength between the copper and the dielectric plays a major role in the stability of the board and also long shelf life of the board.

Microvia and SBU process is very ideal for BGA, flip chip attachment, chip on board methods since it enhances the board density. Board warping disturbs the microvias. Therefore, try to use dielectric materials and substrates where the Tg is high enough. So, if there is a warpage on microvia boards, this is going to disturb the microvia and disturb the adhesion between the dielectric and the copper. Choose dielectric and FR4 core suitably for dielectric constants and other properties that your design demands. Blind vias are easy to produce using laser drilling than mechanical drilling because you can very well control the depth drilling in laser compared to mechanical drill. Therefore, you can get much better accuracy, tolerance can be achieved, and the quality is also much improved in these cases of blind vias.

This process enables use of conductive paste for stacked vias resembling through holes. That means, you open up a via and then you fill the vias with conductive paste so that it resembles a through hole and it can provide much better rigidity.

(Refer Slide Time: 24:08)



Now, the benefits and disadvantages continues; we will discuss some more points. In the case of a microvia SBU process, we have flexibility in choosing number of SBU layers either at the top or at the bottom of the core to some extent; this is not core symmetric. Compare this particular statement with the conventional multilayer boards. If you use a 2 layer core, then if you want to prepare a 4 layer board, you have to add one at the top and one at the bottom. In a SBU process or a microvia/SBU process, you can choose a 4 layer core, build just one additional layer at the top, or two at the top and none at the bottom, or one at the top and two at the bottom, and so on because, basically you are not adding too much of copper in terms of copper foil. You can do additive plating and add only the required number of copper. Typically, you are talking about signal layers at the top and the bottom of the SBU structure, which means the total amount of copper is very much less compared to conventional board. So, based on your design, you can ensure that your top layers prepared by SBU can have very little copper and that will not add to the issue of warpage.

Use of additive plating on build up layers with just enough copper will enable this kind of a problem to be solved. Test the core before the SBU process. So, once you prepare the core, you need to test it because the removal of SBU layers is still a tricky process. Although, before the curing process, you still can remove the dielectric ink, but after it is cured, it is still going to be an issue to repair the SBU layers. SBU here refers to Sequential Build-Up layers or Built-Up layers. Use of different dielectrics can be restricted. Ideally, you want to use different dielectrics for different layers to improve the electrical performance, but it can be restricted due to compatibility with the laser drilling. So, ensure that problem is not there before you design that.

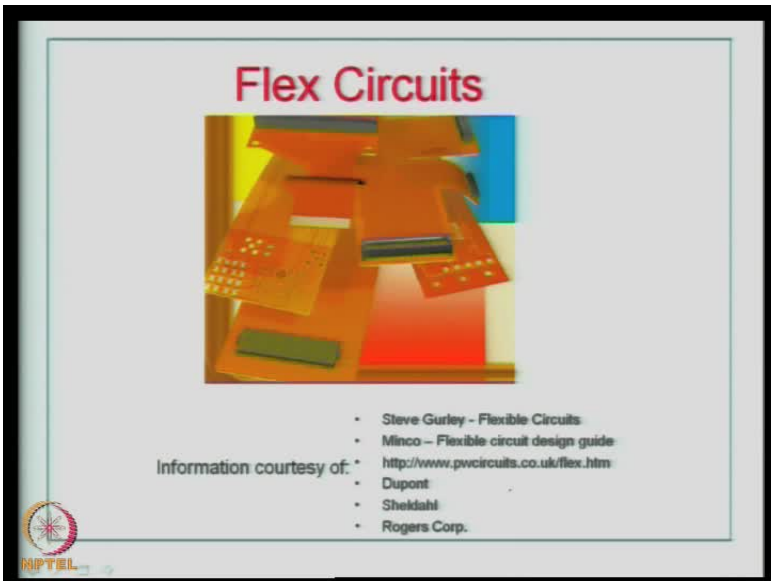
Quality of laser drill is high class. Same hole cleaning procedures that we have seen in conventional is applied here. Different dielectrics can be used for different layers, but again (Refer Slide Time: 26:54) try to link this issue with the statement that I have just mentioned. Increased wiring density obviously is guaranteed. Increased component density is also possible using this process. The dielectric material that we use for the inner layers is the key issue for providing the kind of electrical properties that you require. That makes this particular SBU process unique compared to the conventional board. Its behavior during the thermal cycling testing and all that will give us an idea about the microvia and the conductor reliability.

Finally, SBU obviously eliminates the through holes and the through-hole components. So, you can advance or migrate into non through-hole components; that is, surface mount devices as well as bare chip BGAs, CSPs and so on. Currently, some through-hole components are not available in SMT format. Therefore, only for those, this drilling a through hole becomes essential. So, it depends on what kind of reengineering you are doing for your existing design.

Thermal vias in through-hole format have been well established because the through-hole diameters are very large. The heat dissipation can be much better in a through hole, but in the case of a microvia, where the lands of the microvia are used for the footprint of a BGA. It is being used as a thermal via, but there are some limitations as to how much heat this can take – these microvias as well as the inner layers. So, this is a debatable issue it goes by design to design. SBU has flexibility in the processes and the material choices, which is sort of a conclusion based on all the points that we have discussed so far.

That completes all the aspects about manufacturing and the design for manufacturing issues for all types of boards: conventional boards, multilayer boards as well as High Density Interconnect structures.

(Refer Slide Time: 29:40)

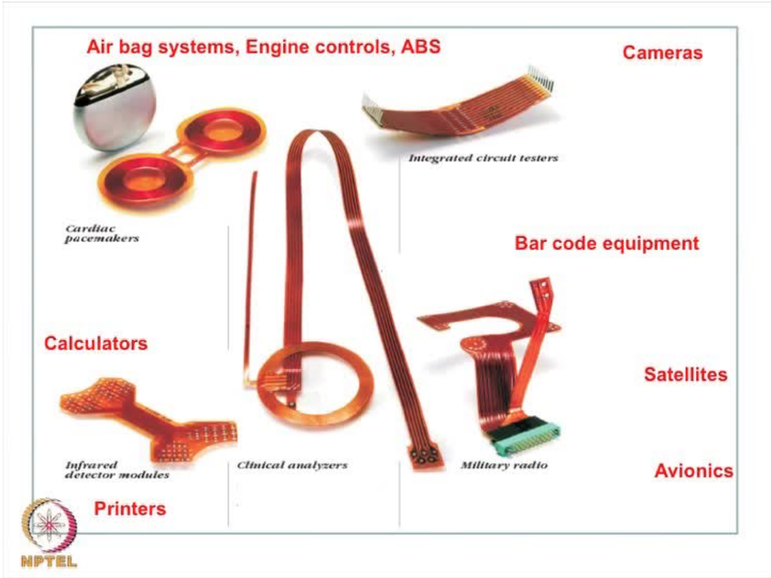


Now, we are going to look into the last aspect of the process technologies in this particular module, which is called the Flex Circuits. Some people refer to it as flexible circuits or flex PCBs. Basically, as the term indicates the substrate is flexible; you can bend it to a certain degree. Why should you have a board that is bendable? Because certain products have less space; the packaging requirement demands that the entire enclosure – mechanical enclosure requires a smaller space; it should be light weight at the same time high performance, but all flex circuits need not be completely flexible. It can be partly rigid and partly flex. So, you can have sometimes a completely flexible base or a circuit or it can be a rigid flex. Some ends as you can see here in this picture (Refer Slide Time: 36:36), some of the ends of the substrate can be rigid and the body of the circuit can be flex. So, it depends on the design, it depends on the mechanical product and the enclosure, where this particular design goes into. Accordingly, you can device a flexible circuit.

(Refer Slide Time: 31:13)

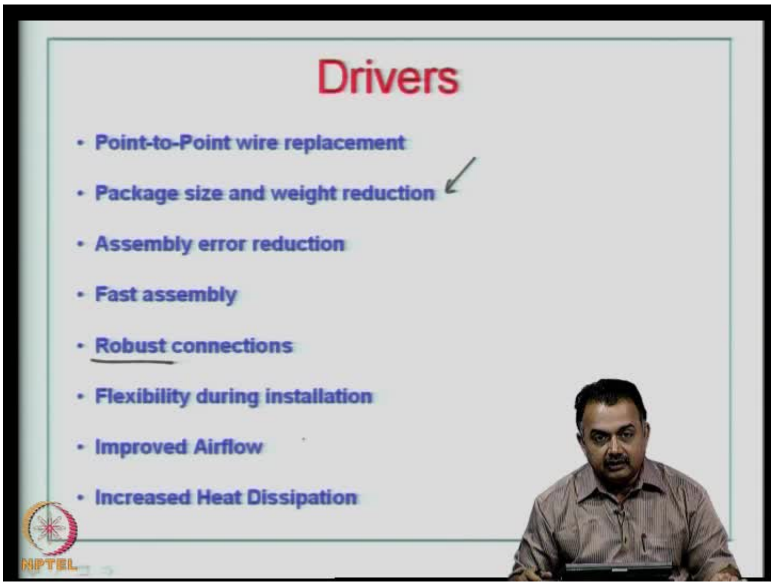
We will briefly discuss flex circuits. Although comparing to rigid substrates, the process technologies are very similar. Flex circuits are typically not meant for high end performance. As the name indicates, flexible circuits are originally designed as a replacement for traditional wire harnessing. Normally, if you look at a flexible circuit, you will see long conductor lines and you will typically have two connectors at the end. Basically, it replaces wires, but they are in a ordered format. So, you will see an array of conductors bonded to a thin dielectric film; it is bent folded or shaped to interconnect multiple planes. So, it will assume a 3-D configuration and you can bend it, roll it, and shove it into a mechanical product. Typically, you will see these kind of structures in cameras, camcorders, and so on. So, servicing is very easy; accessibility is easy; it occupies less space; it is low weight. One would say that these circuits are kind of medium dense or low dense circuitry. So, the names that are used are Flex Circuits, Flexible Printed Circuit Boards, or Flexible PCBs.

(Refer Slide Time: 32:29)



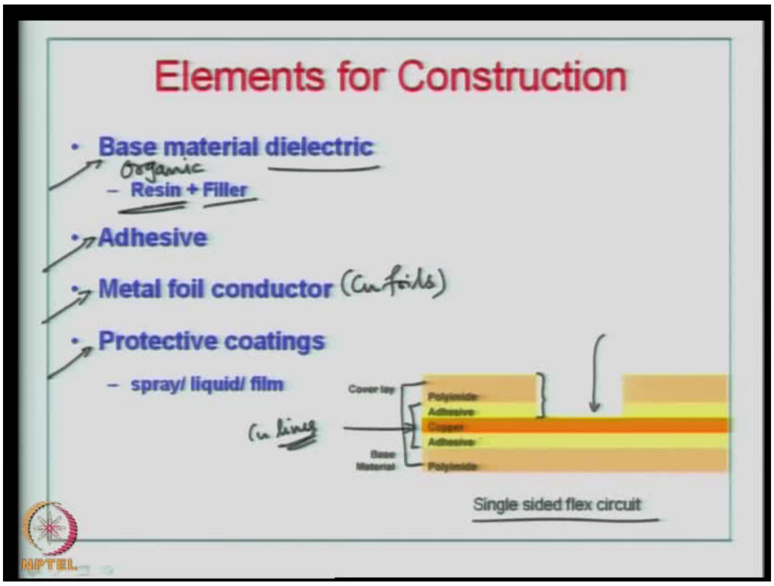
Some of the application areas for flex circuits will be: use in air bag systems, engine controls, ABS that is used in automobiles, then cameras, camcorders, and handheld equipment, bar code equipment, satellites, avionics, printers. In most printers, you will see these kinds of structures connecting to the printer head and the circuit board so that there is a lot of mobility; ideal for such applications where there is some kind of a movement of a mechanical device/electro mechanical device; calculators, because it occupies less space, because it is thin. It is also used in medical applications – biomedical applications like cardiac pacemakers, implantable devices, and so on. So, applications like these are ideal for flex circuits.

(Refer Slide Time: 33:42)



The drivers for flex circuits are point-to-point wire replacement, package size and weight reduction; again very important point – assembly error reduction. Fast assembly because you do not have to deal with bulky wires, but you need to deal with some kind of strips. They are in fact very robust, although they are flexible, thin, and light weight. However, the connections are robust. Flexibility during installation; improved airflow and heat dissipation in devices where flex circuits are used compared to rigid boards.

(Refer Slide Time: 34:25)

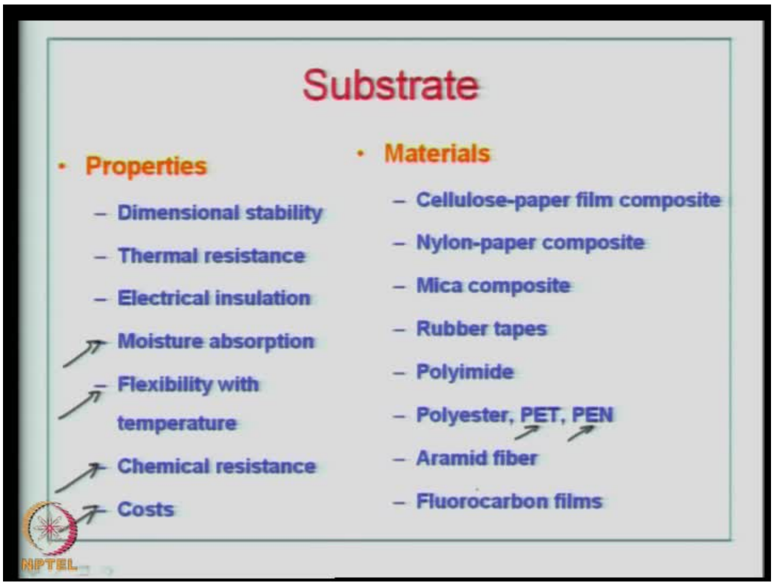


Now, the basic elements for construction; it will essentially require a base material dielectric. So, you need to have some kind of an organic resin, which provides the electrical characteristics that you desire. So, you have a lot of choices of materials to choose from. Filler; you do not use the filler to the extent that you use in a rigid PCB because you need it to be flexible. At the same time, one can say that the filler content can be very low so that you are able to provide the base material that is required for the flex circuit; it cannot be entirely organic resin. So, the filler content can be different, but at the same time, very low in percentage compared to a rigid PCB.

Then, you require an adhesive that can bond the copper foil to the base dielectric material. Here, we are not going to do processes like lamination like we did in a conventional multilayer board. The other part or element in a flex circuit is the metal foil conductor. Typically, you can use copper foils. Then, we use some protective coatings like a spray – conformal coating, liquid, or a film like your cellophane tape or a cellophane film; a dielectric film that can protect the structure of the flex circuit. If you look at the cross section here, what you essentially see here (Refer Slide Time: 36:13) is a copper layer, then you have an adhesive layer, and then you have the dielectric material. You can access by this opening from the top of the dielectric material to the copper.

Essentially, flex circuits are looked at basically a connector-to-connector connection or a point-to-point connection, rather than having a complex circuitry built on the entire body of the flex base. So, you can have the conductor lines. So, this copper represents the conductor lines (Refer Slide Time: 36:54). That is defined by the design – the length, the width, and so on. So, single sided flex circuitry will have this kind of construction. The copper is protected by the polyimide dielectric and the adhesive material.

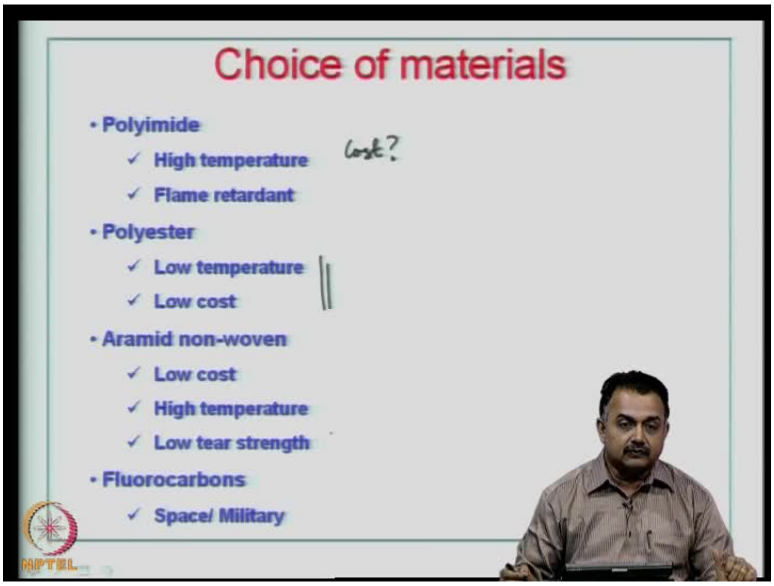
(Refer Slide Time: 37:11)



Now, let us look at the substrate issues – what are the properties and what kind of materials that can be used for flex circuits. The first thing is dimensional stability – very important, thermal resistance, electrical insulation, moisture absorption – very important; just as in rigid PCB, we are worried about the moisture absorption of the dielectric material because it can influence the electrical performance. Flexibility with temperature – although this is flexible at room temperature because there is a limit to flexing the base that we are using, but at the same time, what is the kind of dimensional change that you can look at this material when the temperature is increasing during the operation of product and so on. Chemical resistance – again, issues that we have seen in rigid PCBs. Cost – obviously is an issue because certain products require thinner structures and which cannot be met by the rigid format. So, the obvious choice is to go for a flex PCB. So, the method of manufacturing has to be cost effective if you are producing in thousands.

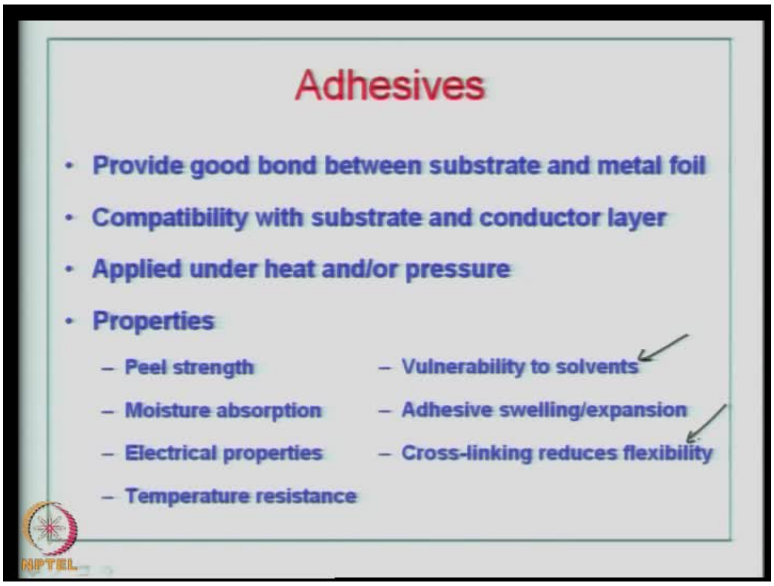
If you look at the materials, the choices are you can use cellulose-paper film composite, nylon-paper composite, mica, rubber tapes, polyimide, polyester, polyethylene terephthalate, polyethylene naphthalate, aramid fibers, fluorocarbon films, and so on. So, a combination of organic films with the filler films will produce the ideal base for a flexible substrate.

(Refer Slide Time: 39:15)



If you use a polyimide, as you know by now, ideally suited for high temperature; it is flame retardant, but at the same time, cost can be an issue. Then, polyester widely used in flex circuits – low temperature, low cost. Aramid non-woven structures – low cost, high temperature, low tear strength. Fluorocarbons ideally used for space and military.

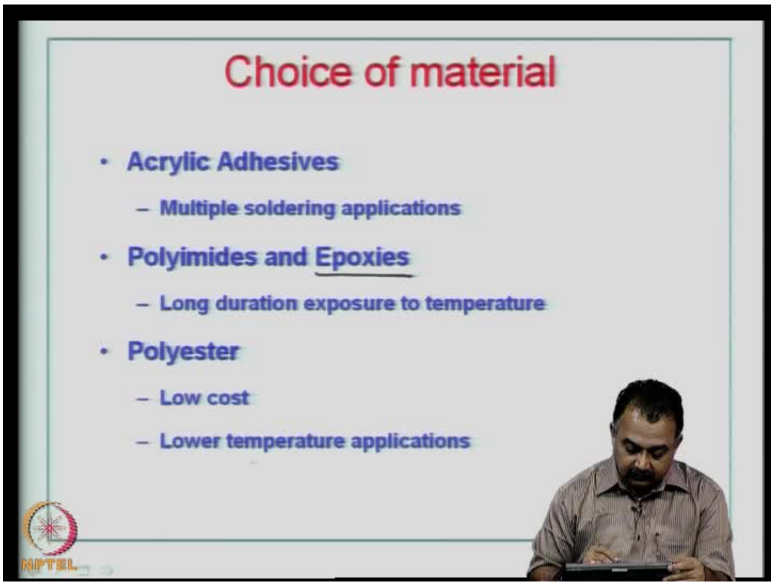
(Refer Slide Time: 39:48)



Adhesives are used to provide a good bond between the substrate and the metal foil; the dielectric and the metal foil. Compatibility should be there with the substrate and the conductor layer. It is applied under heat and pressure. So, it should bond well.

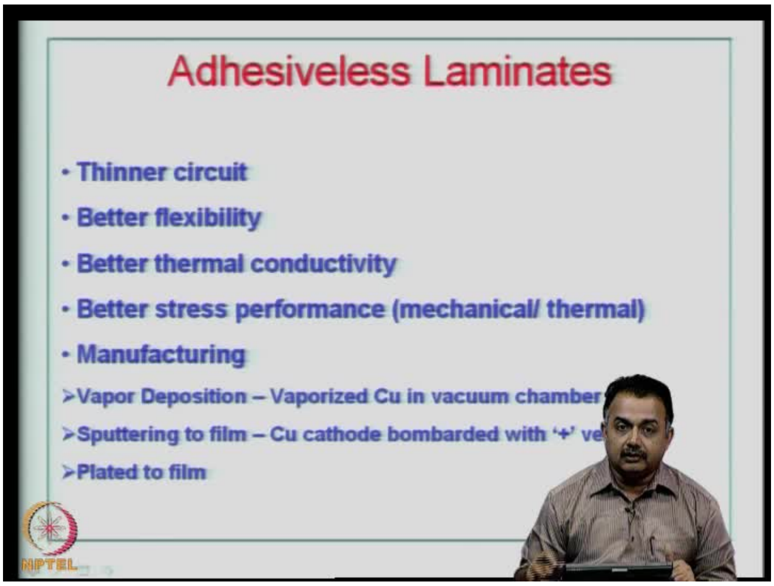
The properties that you look for in an adhesive are the peel strength, moisture absorption because if an adhesive absorbs moisture, it is going to affect the bonding between the foil and the dielectric. Electrical properties, temperature resistance, vulnerability to solvents because if it is going to be dissolving in certain solvents during the processes and so on or during assembly, then it can be a problem with de-lamination between the metal foil and the dielectric. Adhesive swelling and expansion; all these can lead to defects in the entire structure just because the adhesive has got a poor mechanical and physical properties. Cross linking reduces the flexibility. Therefore, use polymers that have less cross linking because we are talking about simple structures that are cost effective.

(Refer Slide Time: 41:12)



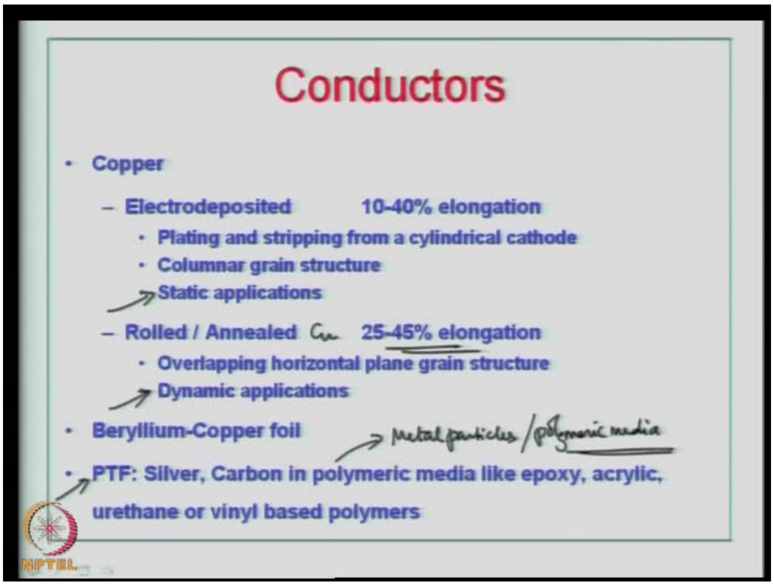
Choice of material: Acrylic adhesives – ideal for multiple soldering applications. Polyimides and epoxies – one would like to use epoxies because they are cost effective; long duration exposure to temperature is possible and you can see less defects with these kind of materials. Polyester is low cost and used for lower temperature applications.

(Refer Slide Time: 41:43)



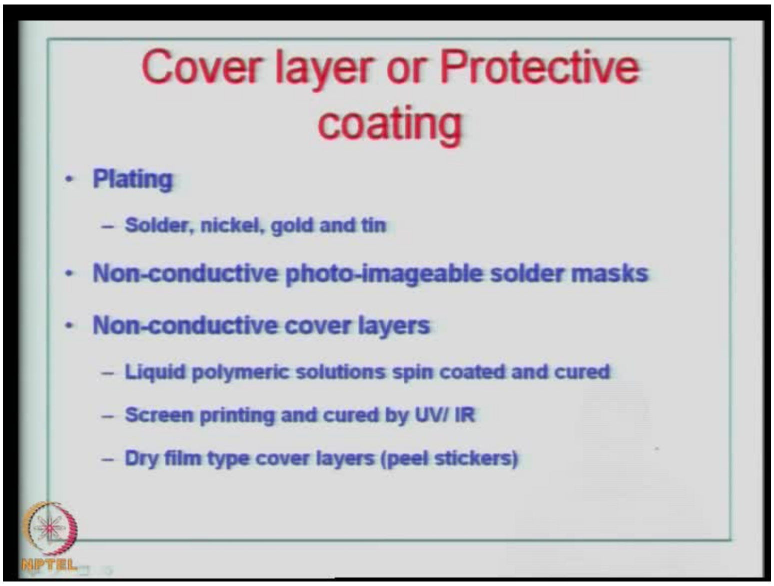
There are cases when we use adhesiveless structures, which mean you can reduce the thickness. So, you get a thinner circuit, better flexibility, better thermal conductivity, better stress performance, and the manufacturing is that you deposit the copper by using vapor deposition or sputtering process or simple electroplating directly on to the dielectric material.

(Refer Slide Time: 42:12)



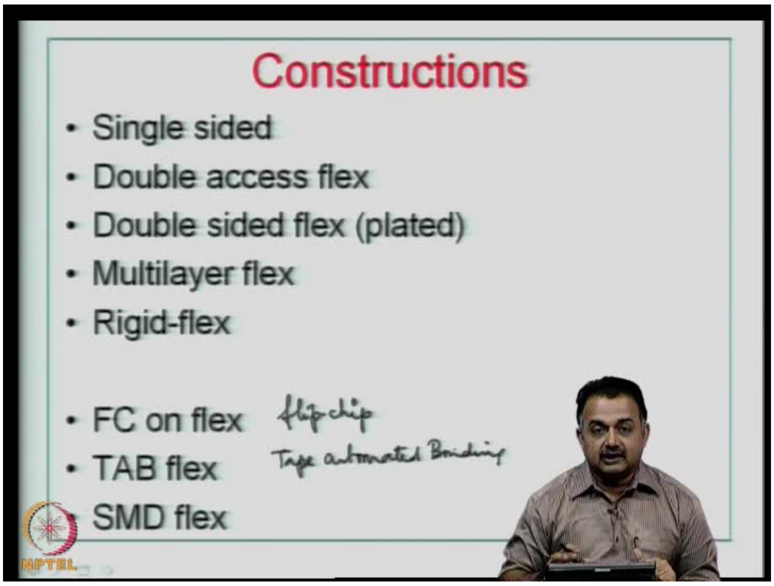
Conductor is essentially copper. It can be electrodeposited. We have seen the electrodeposition process – plating and stripping from a cylindrical cathode; you get a columnar grain structure; static applications. So, typically, the elongation here is 10 to 40 percent. Therefore, that is why we use it for static applications. If you want to use for dynamic flexing, repeated flexing, then we use rolled or annealed copper. Here (Refer Slide Time: 42:47) you can see the elongation is much better, the flexibility is much better. Obviously, the grain structure is different – overlapping horizontal plane grain structure. In some cases, people use Beryllium-Copper foil or in some cases these conductors are realized by printing; stencil printing or screen printing; polymer thick films. So, instead of copper, it can be silver, it can be carbon in polymeric media like epoxy or acrylic, urethane or vinyl based polymers. So, conductors can be realized by electrodeposition foil or rolled / annealed copper foils or by printing using polymer thick films. So, the ideal situation is – printing is very easy. So, decide on which metal particles that you require for a particular application and they have to be mixed in a polymeric media (Refer Slide Time: 43:51) suited for these kind of printing applications.

(Refer Slide Time: 43:57)



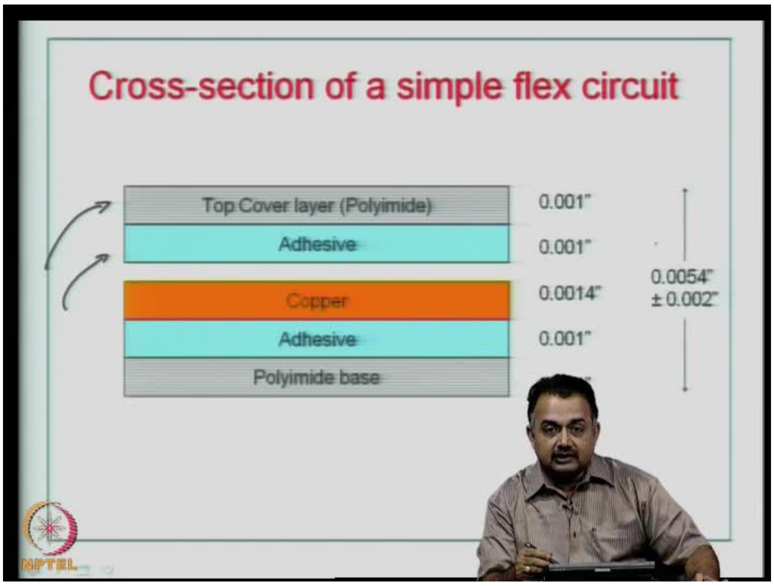
Cover layer or protective coating – either you can protect the copper with solder or nickel, gold, or tin; the similar thing that we have seen in rigid boards. Or, you can use non-conductive photo-imageable solder masks or non-conductive cover layers like a liquid polymeric solutions spin coated and cured, or it can be screen printed and cured by UV or IR, or it can be a dry film tape type of a material or a sticker type of material that can be easily applied and cured.

(Refer Slide Time: 44:35)



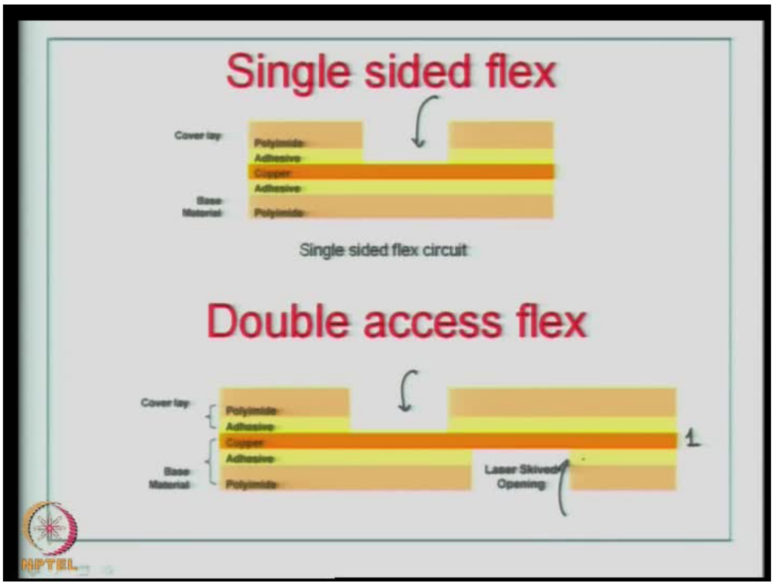
The construction is – you can have a single sided flex, double access flex, double sided flex – plated using a through hole, multilayer flex not very normal, but can be custom built, and then a rigid-flex – a combination of a rigid end and a flex body. Another classification is that you can have a flip chip on flex, you can have a tape automated bonding; that is, TAB, or you can have a surface mount device on flex. So, this is another classification. Basically, you can consider flex as a normal Printed Wiring Board in which you can mount a flip chip and the normal interconnection circuitry, but then you cannot bend it. The obvious thing is that substrate can take the stress and the strain much better compared to a rigid board. So, warpage issues are very important when you consider flex versus rigid for certain applications.

(Refer Slide Time: 45:49)



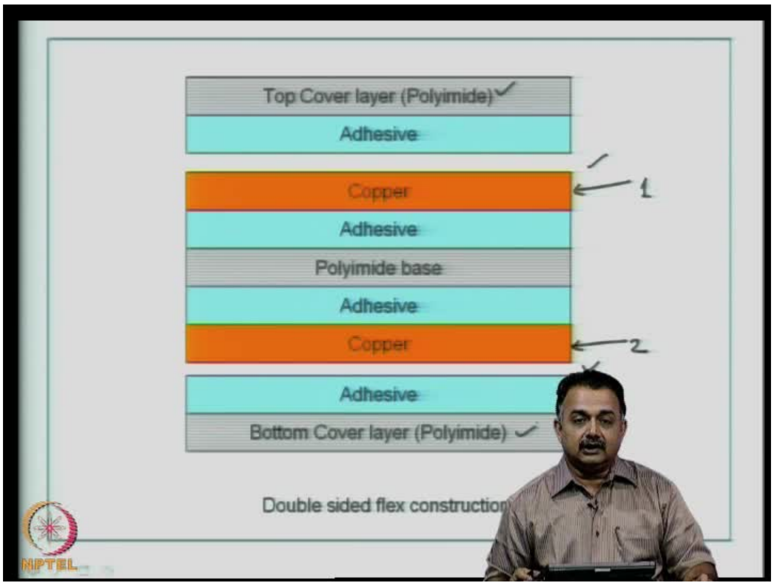
This is the cross-section of a simple flex circuit. Obviously, you have the copper and the adhesive polyimide base. Once the construction is over and the circuitry is realized, you apply the top adhesive and the top polyimide cover layer to protect the circuitry. So, the typical dimensions in inches are given here.

(Refer Slide Time: 46:13)



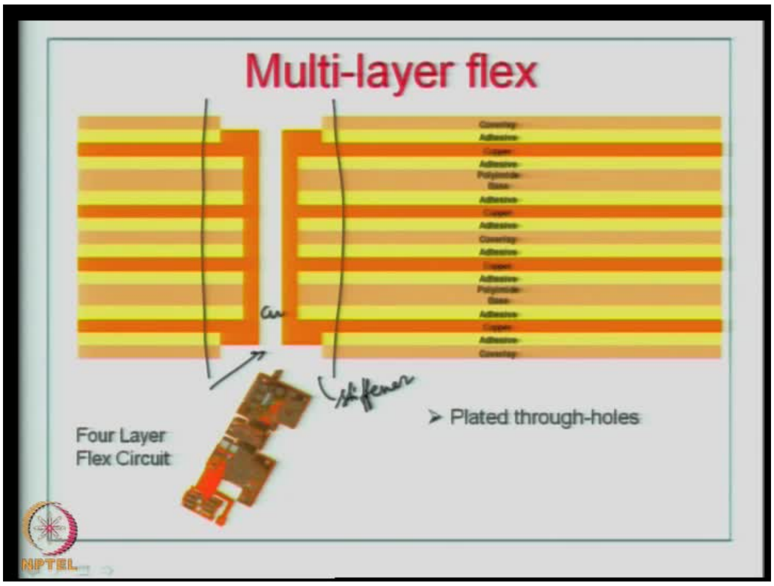
The cross-section of a single sided flex will be as shown here. We have discussed this before. You can access it from here. Now, a double access would mean that there is a single copper layer, but you can access from the top as well as from the bottom. After you bend the board, you can have access from both sides. That means you can access both sides of copper if the flex circuit is bent. So, you provide laser opening for those kind of access that you require for testing or repair.

(Refer Slide Time: 46:48)



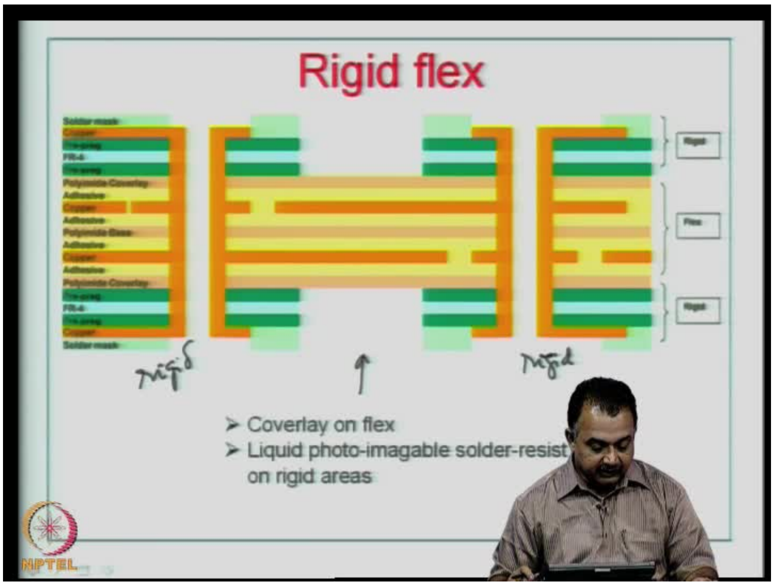
A double sided flex construction will look like this. Obviously, when you say double sided, you have two copper layers: layer 1, layer 2. Then, you have the core base. Then, the adhesive used to laminate the polyimide base to the copper structure. Once this circuitry is realized over here, it is now protected with the top cover layer of the polyimide and the bottom cover layer of the polyimide. So, this is the… take in to account all the thicknesses of the individual layers and you will get the final thickness of the flex circuit.

(Refer Slide Time: 47:29)



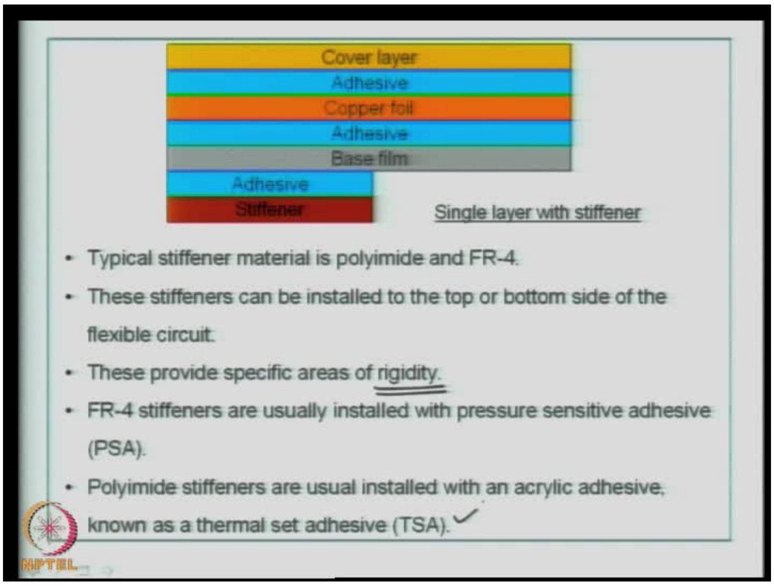
A multi-layer flex in principle can be generated with the same elements of construction, alternating between copper adhesive cover layer, copper adhesive cover layer, and so on. You can drill a mechanical hole here like this (Refer Slide Time: 47:49), which means in this area, you have to provide some kind of a stiffener so that you can drill a mechanically drilled through hole, and then this can be plated with copper, and necessary interconnections can be provided to the inner layers, or simply it can connect all the layers depending on the design. So, this is a modification in the flex circuit build up.

(Refer Slide Time: 48:20)



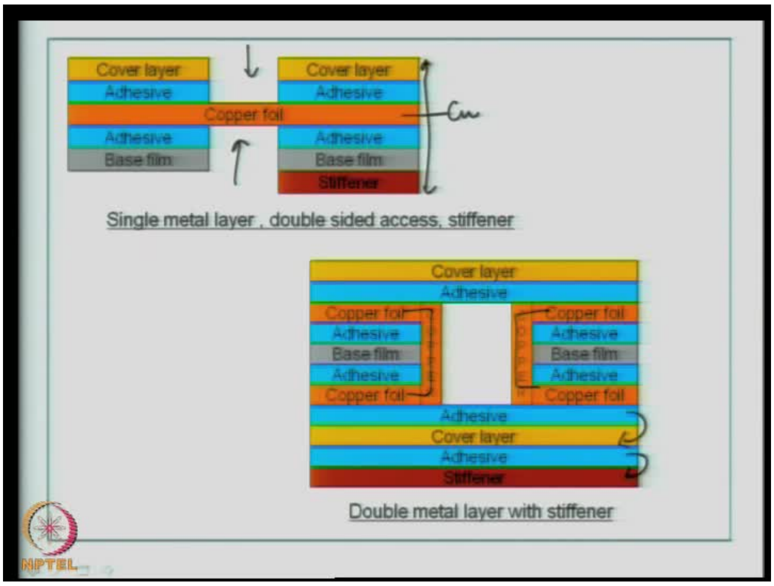
A rigid flex will again look like this. Part of the… so, this will be the flex area, where you can bend the circuit board. These are the rigid areas that you can see. Again, here, some of the areas are rigid. So, based on your number of components, obviously it is a low component density process; the flex process, but based on the requirement of which areas need to be stiffened with the stiffener and mechanically drilled through holes created, you can design a rigid flex board.

(Refer Slide Time: 49:07)



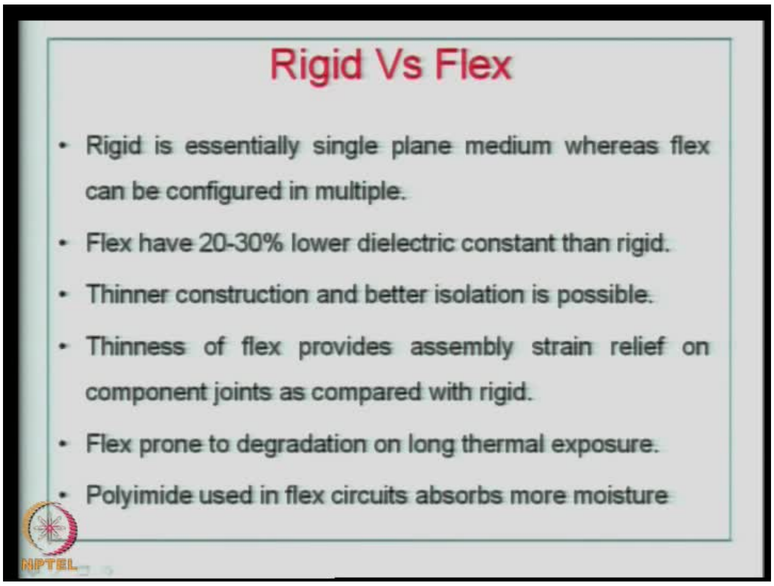
Typical stiffener material is polyimide and FR-4; that is, epoxy material, flame retardant, glass epoxy material. These stiffeners can be installed to the top or bottom side of the flex circuit. These provide specific areas of rigidity. These can hold all the layers together, registration is maintained, and at the same time, it provides an opportunity for mechanical drilling and creating interconnections to the inner layer. They are usually installed with pressure sensitive adhesives. Polyimide stiffeners are usually installed with acrylic adhesive known as thermal set adhesives.

(Refer Slide Time: 49:59)



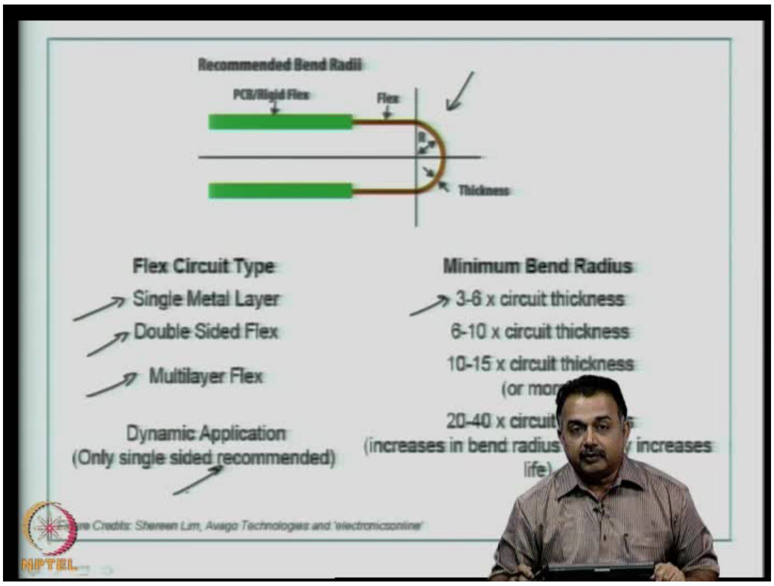
Now, look at the structure here. Essentially, this area is provided with a stiffener, but it is a single metal layer that means single copper layer, double sided access, and stiffener provider. Now, in this structure here, you have double metal layer that means you can see – this (Refer Slide Time: 50:21) is the copper structure, two metal layers: adhesive and cover layer polyimides at the top, and here you have an adhesive and cover layer. Since you are providing a stiffener, you are providing another set of adhesive layer along with the stiffener. So, the choice of stiffeners is the key requirement that goes with the design and the requirement in the product by itself.

(Refer Slide Time: 50:54)



Rigid versus Flex: If you want to compare, rigid is essentially single plane medium, whereas flex can be configured in multiple. Flex have 20 to 30 percent lower dielectric constant than rigid. Thinner construction and better isolation is possible. Thinness of flex provides assembly strain relief on component joints as compared with rigid. Therefore, if you look at the component failure at the solder joints in a flex as well as in a rigid circuit, you will get different results. So, it depends on having the component density in a rigid board can be very high, whereas in a flex circuit, you have to carefully look at what kind of component density you can maintain because there can always be a strain at the component joint in the case of flex circuit. Flex is prone to degradation on long thermal exposure. Polyimide used in flex circuits absorbs more moisture. So, these are some of the points that one can immediately look at in terms of advantages of rigid circuit, or base versus a flex circuit, or a flex base.

(Refer Slide Time: 52:20)

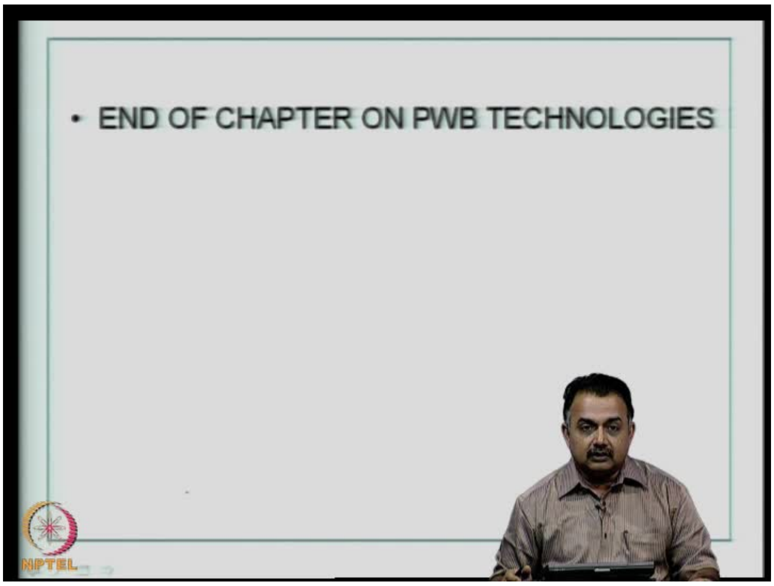


One of the important issues for a designer is to look at the bend radii of a flex circuit. Now, how much can you bend? Typically, it can be a static application or a dynamic application for a flex circuit. If you look at the picture here, you have the bend described here and the radius and the thickness also indicated for a typical flex circuit bending. Now, if you look at the flex circuit type - single metal layer, the minimum bend radius is 3 to 6 times the circuit thickness. So, look at the copper conductor thickness and identify the area of bend in the particular flex board and provide the interconnects in your design so that you achieve minimum bend radius of 3 to 6 times the circuit thickness.

In a double sided flex, you can have 6 to 10 times the circuit thickness because in dynamic applications, you can have situations, where if you do not use the right amount of or the right type of copper, then you can have cracks in the copper, you can have cracks in the resin of this dielectric material if the right resin is not chosen. If the right bend radii is also not chosen, you are going to end up with de-lamination and so on. So, as a designer, you have to look at what kind of circuit you have designed and what kind of bend radii you can provide that finally can be whether it is a static application or a dynamic application before it goes into a product.

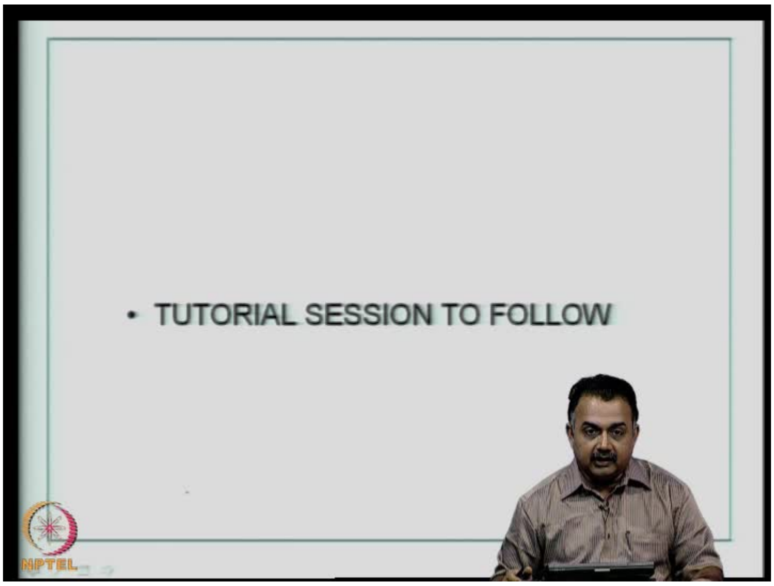
In a multilayer flex, 10 to 15 times the circuit thickness or more. In a dynamic application, obviously it can be 20 to 40 times the circuit thickness; increases in bend radius normally increases the life of the product. So, in a dynamic application, obviously single sided is recommended. These are some of the design rules or design guidelines that one can find that is being practiced in the industry as far as the critical issue of bend radii is concerned. So, look at what type of board you have manufactured – single metal layer, double sided flex, multilayer flex board, and whether it is for a static application or a dynamic application. Accordingly, identify the bend area that will provide the bend radius for the entire board.

(Refer Slide Time: 55:13)



This concludes the chapter on Printed Wiring Board technologies.

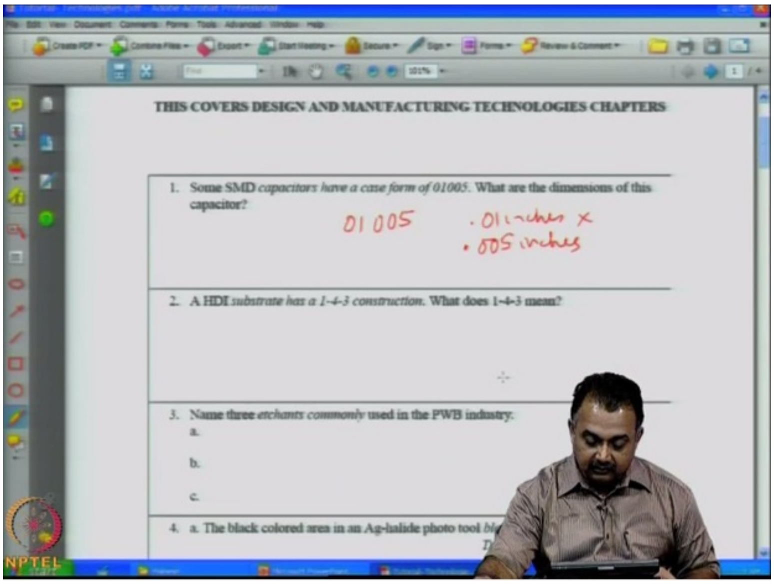
(Refer Slide Time: 55:21)



We will now look at briefly a tutorial session. This tutorial is aimed at giving you some kind of a review on this particular chapter or module on Printed Wiring Board technologies.

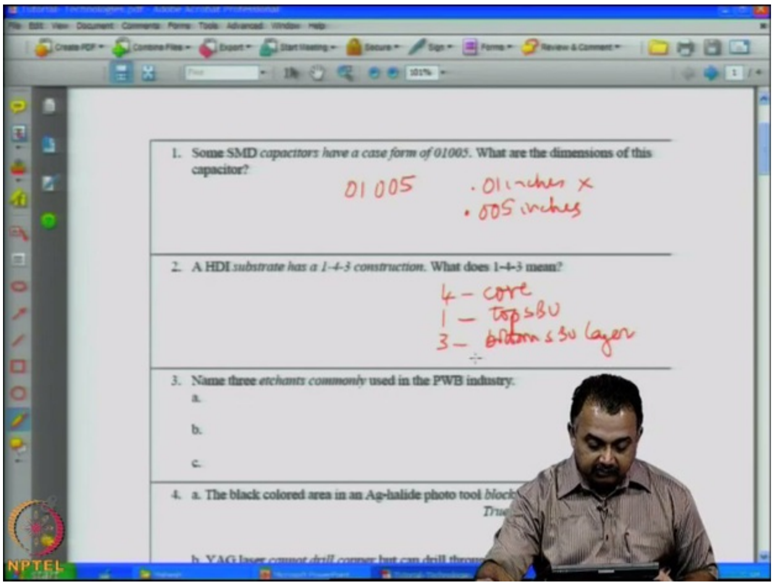
Now, this can be attempted by you after going through all the nine lectures on the Printed Wiring Board technologies.

(Refer Slide Time: 55:55)



Basically, if you look at some of the questions here, the first question is - some SMD capacitors have a case form of 01005; what are the dimensions of this capacitor? This will give you an idea about the shape, the size, and the criteria for using SMD components. Here, 01005 indicates that the capacitor has a dimension of 0.01 inches by 0.005 inches. So, this indicates the case form of the capacitor.

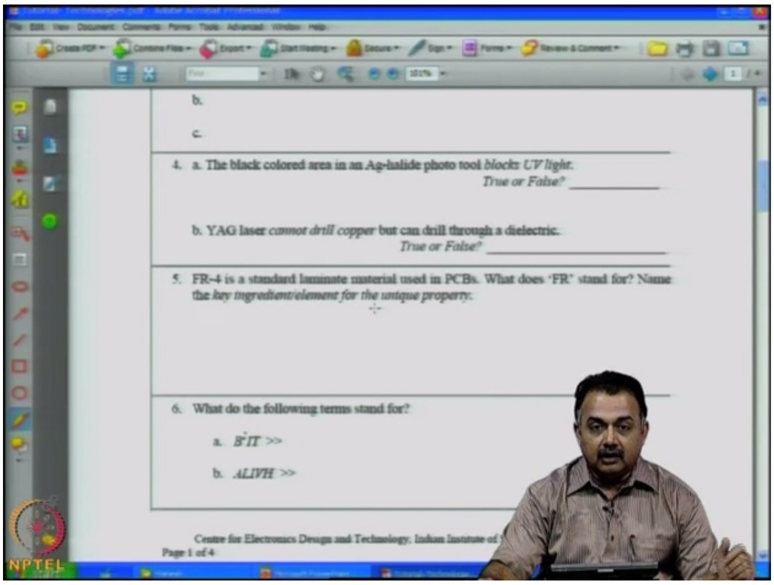
(Refer Slide Time: 56:40)



A HDI substrate has a 1-4-3 construction. What does 1-4-3 mean? We have seen that 4 indicates the core's layer that is used and 1 is the top SBU layer and 3 indicates the bottom SBU layer.

Name three etchants used in the PWB industry. You should be able to look at three common etchants that are used in the PCB industry; typically, ferric chloride, cupric chloride, and alkaline ammonia.

(Refer Slide Time: 57:24)

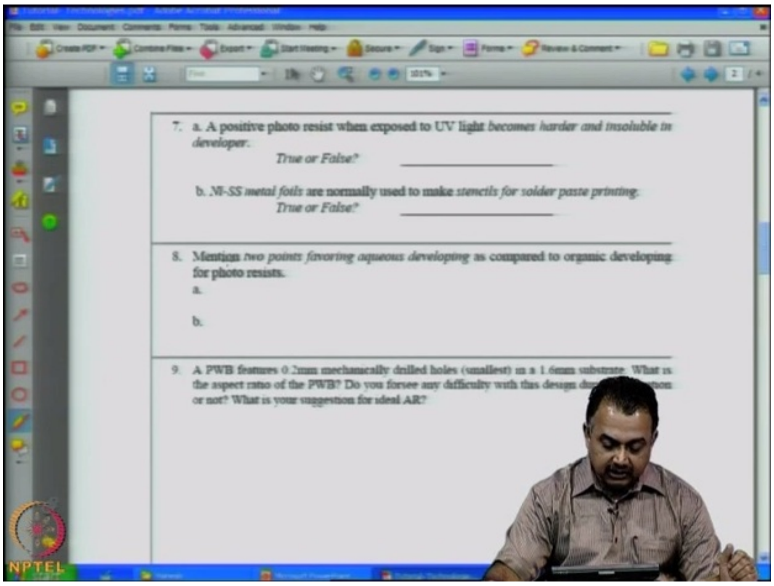


The next question is – The black colored area in a silver halide photo tool blocks U V light. True or false? Basically, if you look at the photo tool, which is black in color, it blocks the UV light. So, it is true. YAG laser cannot drill copper, but can drill through a dielectric. Ideally, YAG laser can drill through copper as well as the dielectric material.

FR-4 is a standard laminate material used in PCBs. What does FR-4 stand for? FR stands for flame retardancy or fire retardancy. Name the key ingredient or element for the unique property. The key element in the case of FR-4, we normally use bromine currently, but today it is replaced by other materials since bromine is banned in the industry today.

What do the following terms stand for? B squared I T – B squared is Buried Bump Interconnection Technology. ALIVH – ALIVH stands for Any Layer Interstitial Via Hole technology. A review questions like these are very interesting for you to have an understanding of what you have gone through in this chapter.

(Refer Slide Time: 58:54)

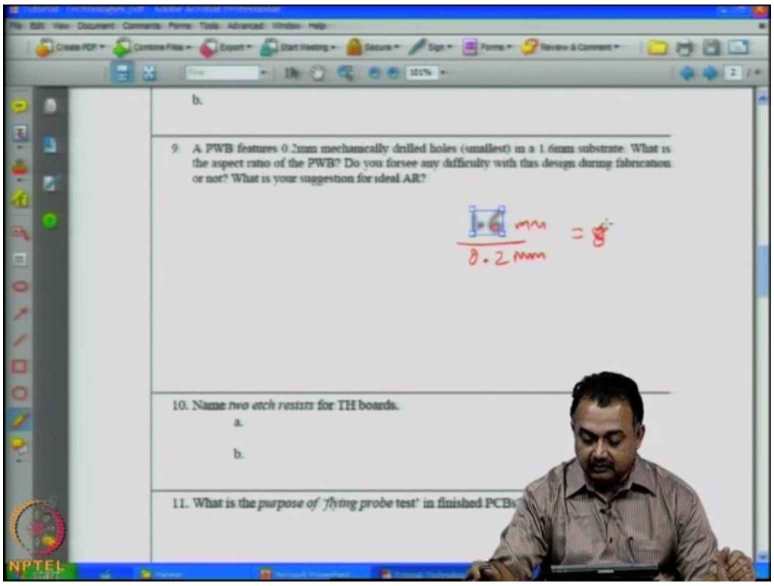


A positive photo resist when exposed to UV light becomes harder and insoluble in the developer. When a positive photo resist is exposed to light, it actually becomes soluble in the developer.

Nickel stainless steel metal foils are normally used to make stencils for solder paste printing. True because, for stencil to have a longer life, we use stainless steel impregnated with nickel; so, this is true.

Mention two points favoring aqueous developing as compared to organic developing for photo resists. Aqueous is environmentally friendly; more yield is possible with aqueous developing; it can be prepared in your own lab or site. Organic developing – most of the organic solvents are banned today because of hazardous nature post by the organic solvents.

(Refer Slide Time: 59:56)



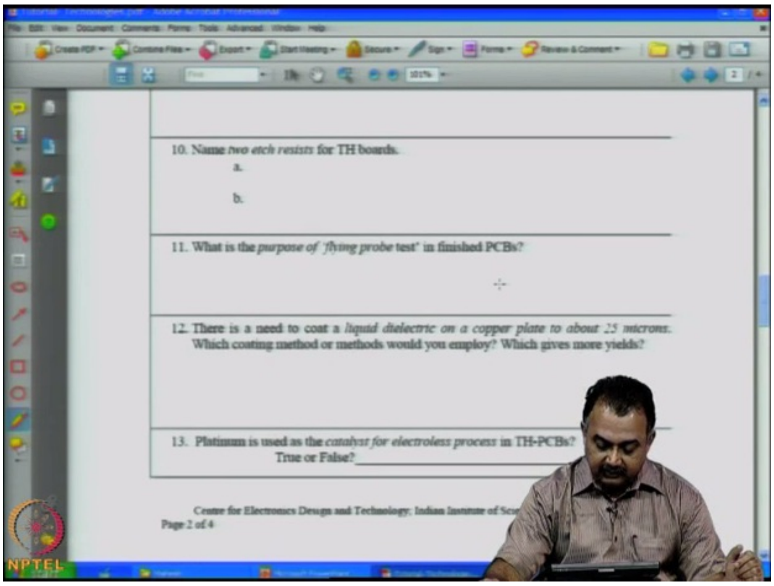
Now here there is a question about the aspect ratio. A PWB features 0.2 mm mechanically drilled holes in a 1.6 mm substrate. What is the aspect ratio of the PWB? I think we have discussed this in detail. So, 1.6 mm is the thickness and the drill size is 0.2 mm. So, the aspect ratio is 8.

Is there a difficulty in processing this kind of a board? Obviously, it is difficult because on a 1.6 mm substrate, you are talking about drilling a 0.2 mm hole. Unless the electroless plating is perfect, it is very difficult to achieve high yield with this aspect ratio.

Name two etch resists for through-hole boards. One, can be a metal resist like gold, nickel gold; the other can be a photoresist.

What is the purpose of flying probe test in finished PCBs? It is basically two – look at opens and shorts in your finished board so that the information can be passed on to the customer.

(Refer Slide Time: 1:01:12)

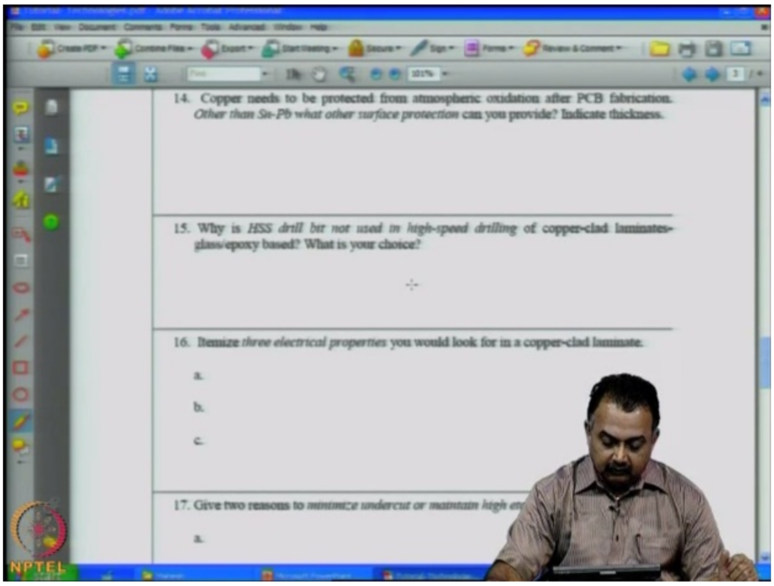


There is a need to coat a liquid dielectric on a copper plate to about 25 microns. Which coating method would you employ? For 25 microns, ideally you can use curtain coating or spin coating. Spin coating with good consistency can be achieved.

Platinum is used as the catalyst for electroless process in through-hole PCBs. True or false? It is actually palladium. So, platinum is not the material that is used.

This kind of a review enables you to have look at the chapters that we have seen. A couple of more questions:

(Refer Slide Time: 1:01:52)

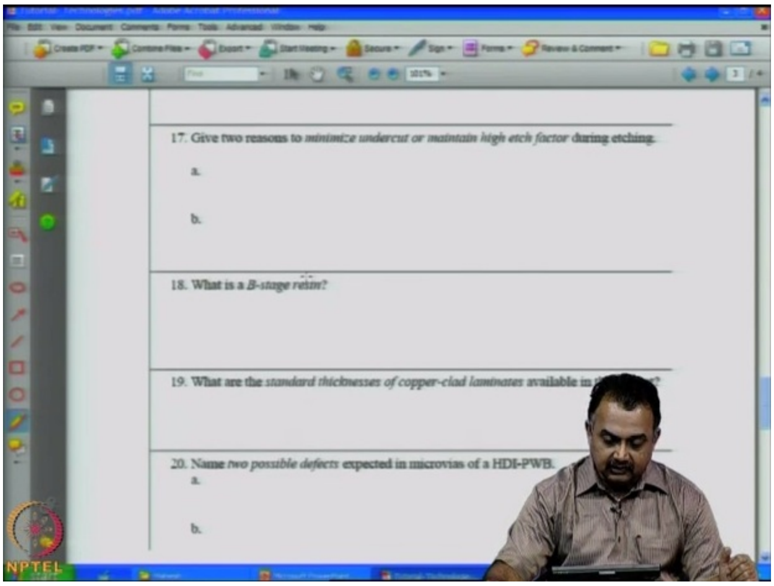


Copper needs to be protected from atmospheric oxidation after PCB fabrication. What other protection can you give other than tin-lead? You can use typically tin coating or the… lead is not being used today. You can ideally use nickel-gold. Nickel to the extent of 5 microns and gold to the extent of less than 1 micron can be used.

Why is HSS drill bit not used in high-speed drilling of copper-clad laminates? Because of the material; we use tungsten carbide. High-speed steel will break if we use in the very tough glass epoxy substrate material. This is because of the high speeds that we are working with.

Itemize three electrical properties you would look for in a copper-clad laminate. There are so many things that you can list – warpage, dielectric constant, moisture absorption; All of these are physical properties, but in terms of electrical properties, it can be dielectric constant, low loss tangent, and then the high voltage breakdown strength that you require.

(Refer Slide Time: 1:03:09)



Give two reasons to minimize undercut in your design or maintain high etch factor. You have to maintain the solutions constantly, look for the elements in the right proportion and provide tolerances in your design.

What is a B-stage resin? Your prepreg is a B stage resin.

What are the standard thicknesses of copper-clad laminates? 0.8 mm, 1.6 mm, and 3.2 mm are the standard thicknesses of copper-clad laminates.

Finally, two possible defects in microvias – we have seen that one is delamination or poor adhesion and the other thing is shrinkage of the dielectric material. So, this is the end of the tutorial and end of this module on High Density Interconnect technologies and a complete spectrum on Printed Wiring Board technologies.