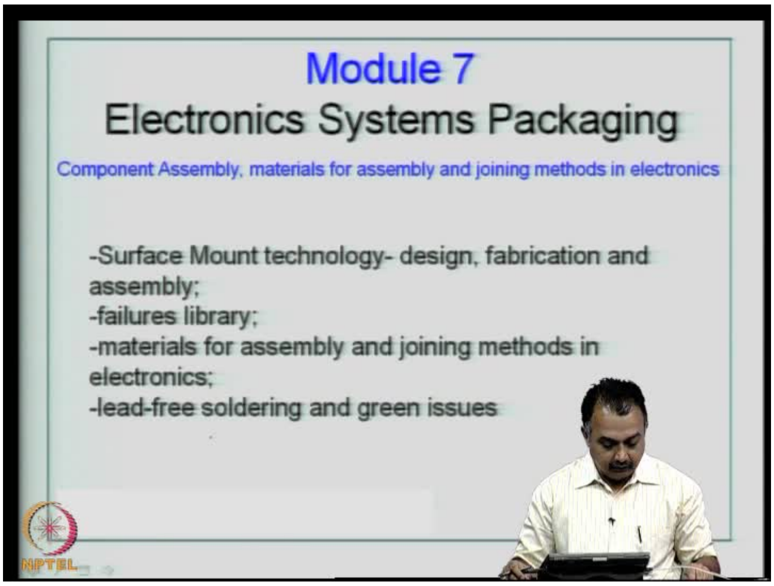
An Introduction to Electronics Systems Packaging
Prof. G. V. Mahesh
Department of Electronics Systems Engineering
Indian Institute of Science, Bangalore

Module No. # 07
Lecture No. # 32
SMD benefits
Design issues
Introduction to soldering

(Refer Slide Time: 00:14)

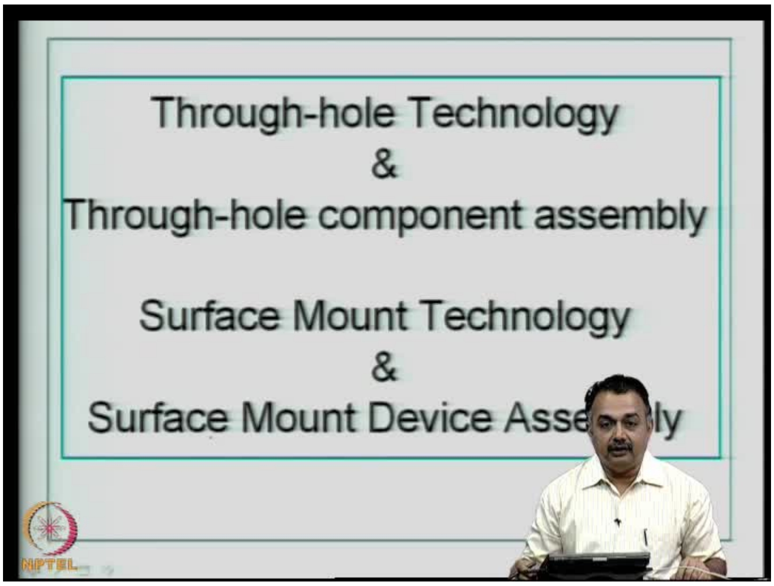


Welcome to this module, Module 7 of this course on Electronics Systems Packaging. This module will cover the following basic topics: Component assembly, materials for assembly that is the board assembly. Here, we are talking about system level Printed Wiring Board. So, I keep emphasizing before every chapter that we look at system as a very important entity in our packaging course. Because right from the beginning, we have been defining what a system is and in specific reference to board manufacturing and board assembly, we are looking at these entities as system level board and system level assembly. We will also talk about joining methods in electronics. When I say joining methods, it would mean the component attachment. If you look at the other modules that we have covered, we have started with a basic definition of packaging, we have moved into semiconductor packaging. Then, we have looked at various packages. We slowly moved into the fabrication area. We looked at the various PWB technologies, including advanced technologies like multi chip module, system in package and so on.

Now, we come to a stage where we have the board ready, we have the packages ready and we are now going to look at situations on how to assemble a reliable board and how to test it thereafter.

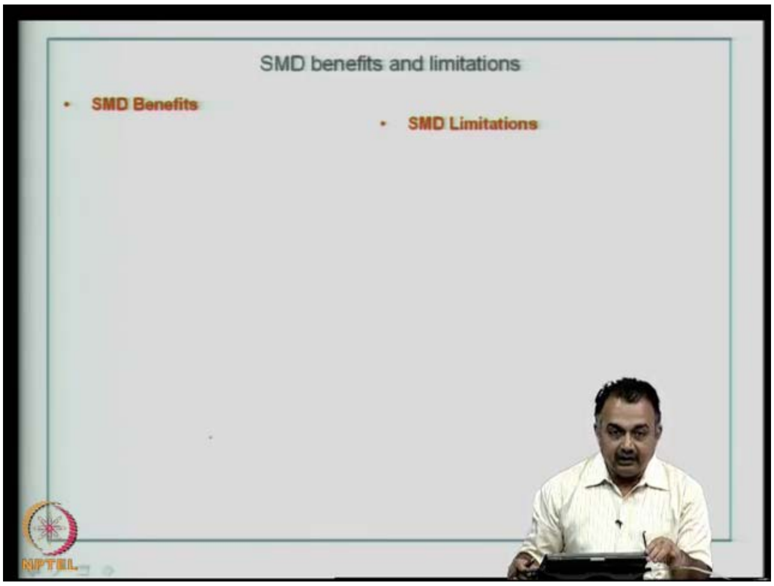
The chapters that will be considered in module 7 will be Surface Mount Technology-definition and basic description, design, fabrication, and assembly; failures library; materials for assembly and joining methods in electronics; and lead free soldering will be an important topic that will be covered. As I have been describing all along, we are moving into an era where we are not supposed to use lead in assembly. We will also look at basic issues in the green environment area, so that we are on page with the legislations that have been introduced in the electronics world.

(Refer Slide Time: 02:59)



Now, there are two things. The first thing is Through-hole Technology. We have understood how a through-hole board has been manufactured and then, once the board is available and once the through-hole components are available, you need to understand how to use an assembly procedure for through-hole components. Similarly, Surface Mount Technology is a technology where we will be using surface mount devices and the assembly methodology that is used will be known as Surface Mount Device Assembly procedure.

(Refer Slide Time: 03:37)



First of all, having seen the various packages, through-hole packages, surface mount, BGAs, CSPs and so on, including flip-chip which is basically a bare die. You are now aware of the various form factors.

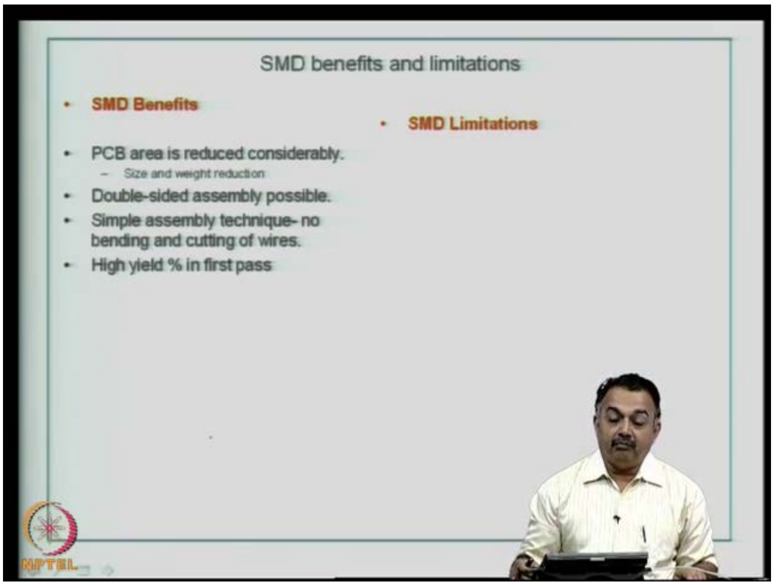
Have a look at this particular sample here that is shown. This is a board where you can see various packages mounted on the surface of this board as you can see here. Now on the other side, there is no interconnect. So, basically what you see here is a surface mount technology established and all the components that are used here are classified as surface mount devices. You can see very small components like small outline IC packages, you can see QFPs and a host of resistors and capacitors, which are very small and sometimes these cannot be handled by bare hands, you need to rely on automated placement procedures to assemble these kinds of very fine pitch components. So this is an example of a board which is a surface mount device.

I will also beforehand, show you some of the packages that are available in assembly form. This is a resistor reel and you can see the small resistor packages that are seen here in a row, in a reel or a tape form and these will be inserted into the pick and place machines for automated assembly. This is a surface mount passive device.

Here you can see an example of a very small product, which uses complete surface mount devices, except for the connectors at the edges. So, it can be a mix of through-hole and surface mount device assembly. As a designer, you can expect a mixture of through-hole and surface mount assembly procedures. This particular chapter will try to highlight how you can handle your design right through fabrication and through assembly.

Here again, what we have tried to show you here is a mix of various types of surface mount devices. You can see transistors, then you can see various resistor packs, these are all resistors, preset resistors, regulators, then you have BGAs also coming under the surface mount classification, diodes, LEDs in surface mount format and so on. Various capacitors including tantalum capacitors, fuses, switches, trim capacitors and so on.

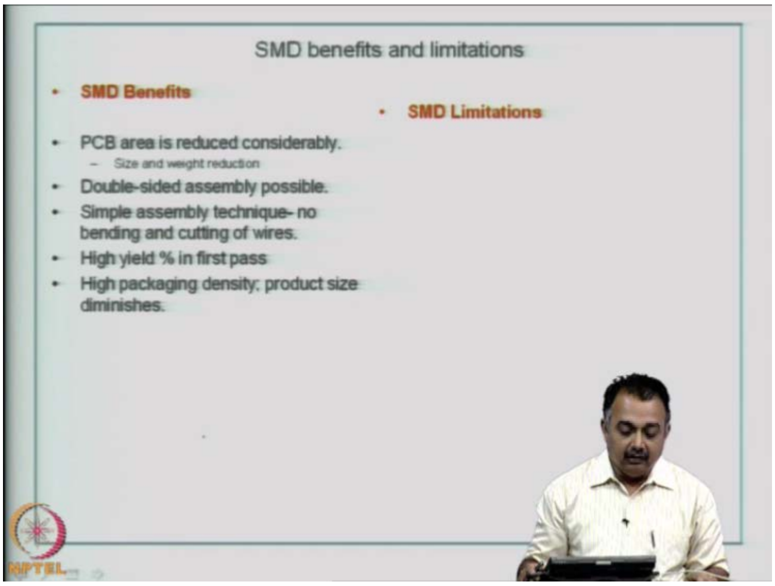
(Refer Slide Time: 03:27)



So, there are a wide variety of components that are available in surface mount format and if you are doing a re-engineering work, from a through-hole assembly into a surface mount assembly, then, you need to be aware of the packages that are available in the corresponding format. To begin with, let us look at the surface mount devices, benefits and limitations. Then you can compare this with the through-hole technology and try to see how a design can be ideally suited for such an assembly.

In a Surface Mount Device Design, PCB area is reduced considerably, because we are doing away with large components. We are doing away with through-hole technology to some extent, because these components do not require vias through which the leads or the pins need to be inserted. They have to be mounted on the surface and soldered on the surface. Therefore, this resolves in a considerable reduction in size and weight. Double-sided assembly is possible. So, you can design in such a way that both the sides can have surface mount devices. Of course, the interconnects can be established, by a simple via connection. These vias will perform the function of basic interconnection between layers rather than looking at the possibility of having a host of through-hole vias, which include component mounting through-holes. This is a very simple assembly technique. No bending and cutting of wires which as you know, needs to be done for through-hole components; high yield percentage in first pass of assembly. You will see that when you go for a through-hole technology, mostly you will rely on manual soldering or manual attachment because the sizes are fairly large.

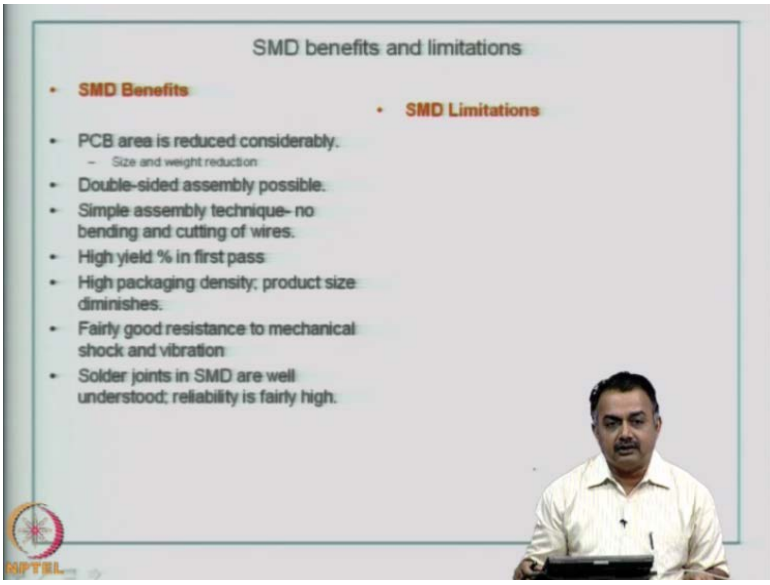
(Refer Slide Time: 09:53)



But in some cases, through-hole technology assembly can also be automated; but it is difficult. At the same time, in contrast, surface mount assembly can be automated, because the components have to be placed on the board at the coordinates or locations where they have to go. You are going to create a system level board with the coordinates or the pads where these packages have to be placed and then you do the assembly. So, in such a contrasting assembly procedure, you are expected to get a high yield percentage pass in the assembly and soldering. Because you are using surface mount device, obviously the packaging density is very high and product size is reduced. If this is going into a product, you can imagine the size reduction compared to a through-hole system level Printed Wiring Board. The devices are fairly resistant to mechanical shock and vibration. You can imagine that the through-holes through which the components are inserted, can experience continuous shock and the reliability of copper plating is at stake.

Compare this with the surface mount assembly. We will discuss this again in this particular module.

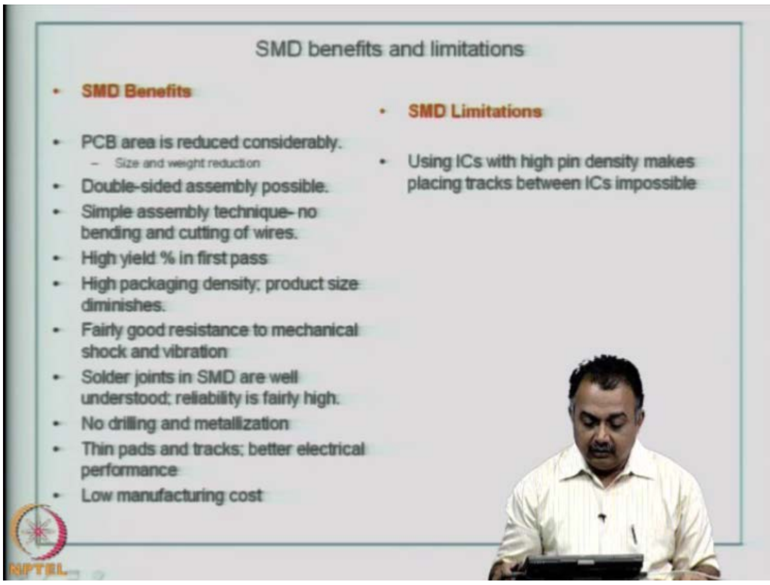
(Refer Slide Time: 10:43)



Solder joints in surface mount devices are well understood. Reliability is fairly high. So, during the last three decades or so, there have been various studies on the solder joint mechanism in surface mount devices. It has been found that even if you use different types of joining material that is the solder material, the solder joints have been well understood. The interface mechanisms have been well understood and the reliability for such boards is very high. The one elimination that has been done in surface mount device boards is that there is no mechanical drilling for mounting the components. When you drill such holes for through-hole components, you can imagine the metallization to be done, the copper to be added and then the reliability gets built up during the copper plating. Of course, it can be at stake also, when the copper plating is not done properly. So, there is a big question mark on the through-hole reliability, in the case of through-hole boards.

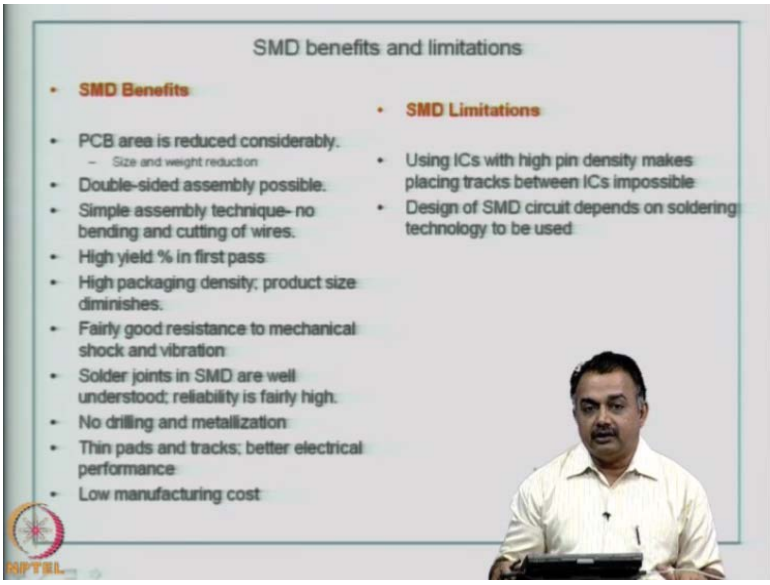
But in a surface mount device, although the buildup technology for the core and the subsequent layers could have vias, which interconnect or connect the interconnects are copper conductors, but you are totally eliminating mechanical drilling or other drilling forms for mounting the devices. Therefore, you are not adding too much copper onto the surface of the board. The packaging density is very high when you use surface mount devices. Therefore, you will match these kind of features with the interconnects; the copper traces that you design for these devices. Therefore, you will have thin pads and you will design tin tracks, copper tracks and subsequently the electrical performance will be very high for surface mount device boards, compared to a through-hole technology board. Low manufacturing cost, because the process automation is well established, the yields are fairly high, because the CAD data that you get from the design for the placement. The vision that you should have as a designer in terms of what assembly process this has to go will go a long way in reducing the cost, if you are thinking of a product that is going to be made in thousands in volume manufacturing.

. (Refer Slide Time: 13.32)



Now, the limitations of SMD on the other side, is that if we use active devices, ICs with high pin density makes placing the tracks between the ICs impossible. Because this has been practiced when DIP packages were used and this was to some extent eliminated when QFPs with fine pitch was used; because you are not able to run tracks between the pins. If you use again advanced packages with low pitch, the possibility of placing tracks between ICs is impossible but that is compensated with other mechanisms of reducing a high density board, like using a micro via and then routing your tracks.

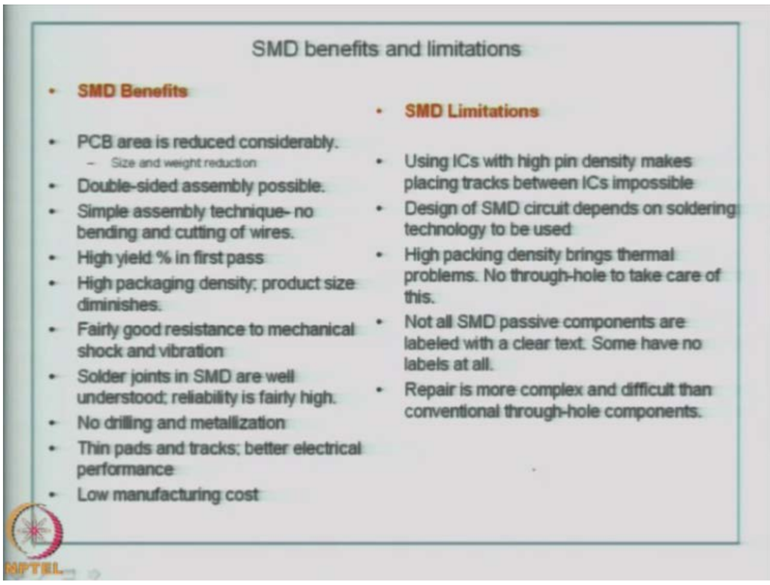
(Refer Slide Time: 14:34)



The design of Surface Mount Device circuit depends on the soldering technology to be used. We are going to discuss in this chapter, the methods of soldering surface mount devices. As a preview, I would like to emphasize here that as a designer you have to be aware of what end process; that is the soldering technology is going to be used for your circuit design and your type of components that you have used, because a lot of reliability is built on the method of soldering. High packaging density brings thermal problem. No through- hole to take care of this. On one hand, we are eliminating through-hole technology based components on the other side we are using packages which are of high density. Some of them will come with problems like thermal handling issues.

Some packages may give out so much of heat and you have to be ready as a designer to design thermal solutions for such devices. If you do not take care of those actions, then although you have converted into a high density system level Printed Wiring Board, you will not have sufficient reliability built into the system which was taken care of earlier in through-hole packages, because a through hole barrel used to be way fairly large. There used to be large amount of copper in these through-holes and they used to act as a heat sink solution. But the design issues for such packages with high density have to be revisited and revised. Not all SMD passive components for your design are labeled with a clear text on the component; some have no labels at all. So, you have to be very careful or you have to instruct the manufacturer or the services people who do the assembly have to be very clearly aware of the notations and the type of reference designations that go along with your passive components.

(Refer Slide Time: 16:51)



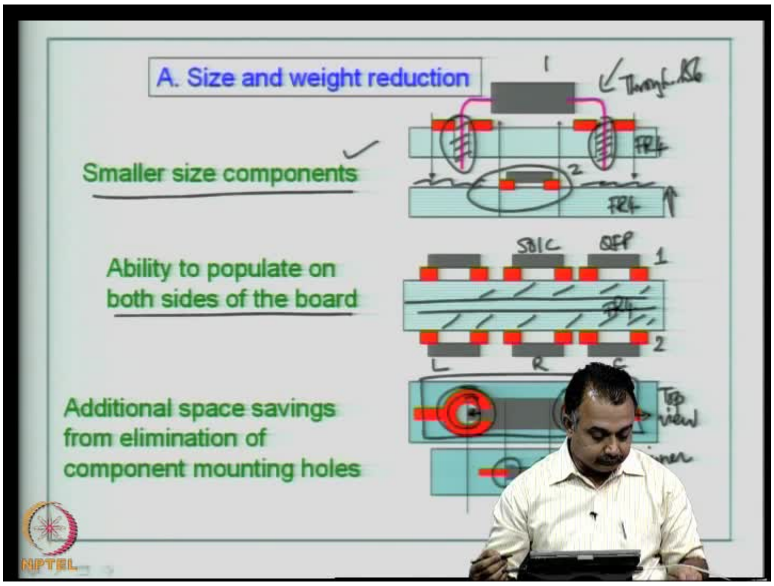
The repair of surface mount devices and the boards are much more complex and difficult to be carried out than conventional through-hole components, because in a through-hole component, you basically flip your board and de-solder the joint material and pull your component out. But in this case, this is a surface mount assembly. These are very low pitch components and you cannot damage your components if they are particularly very expensive and you need to be trained in repair and rework of surface mount devices.

Some class of components has not yet been available in surface mount format. They are still available in through-hole format especially some power devices. Therefore, you might still be using them. There is a chance that your boards or re-engineered boards cannot be 100 percent surface mount device format. You have to live with through-hole components in those cases. In some cases, the custom-built surface mount devices have been very expensive. Therefore, you would go for through-hole technology boards.

There is no harm in using a through-hole technology mixed with surface mount devices. The only thing is you need to be very clear about what kind of assembly procedure your board will have to go through and your board should have received minimal thermal shock during the entire process of assembly and finally amortization of old installations.

The technology changes very fast in surface mount devices format. The assembly equipments are changing very fast; the throughputs are increasing day by day with new equipment. Therefore, an industry will have to think about the capital investment and how much it can sustain. So, amortization of old installation and the time taken for you to make profit using your old installation are all economic issues that need to be considered.

(Refer Slide Time: 19:04)



Now, let us get into some of the technical details into what we have just described in the previous slide. One of the important topic is when you think about surface mount device is that there is a size and weight reduction basically, because you are using smaller size components. So, if you look at the through-hole technology here, this is a through-hole figure you can see the through-hole component. The component is inserted into the through-hole and this particular hole has to be plated to provide the electrical interconnection. You can of course build double sided boards here and compare it with a surface mount device.

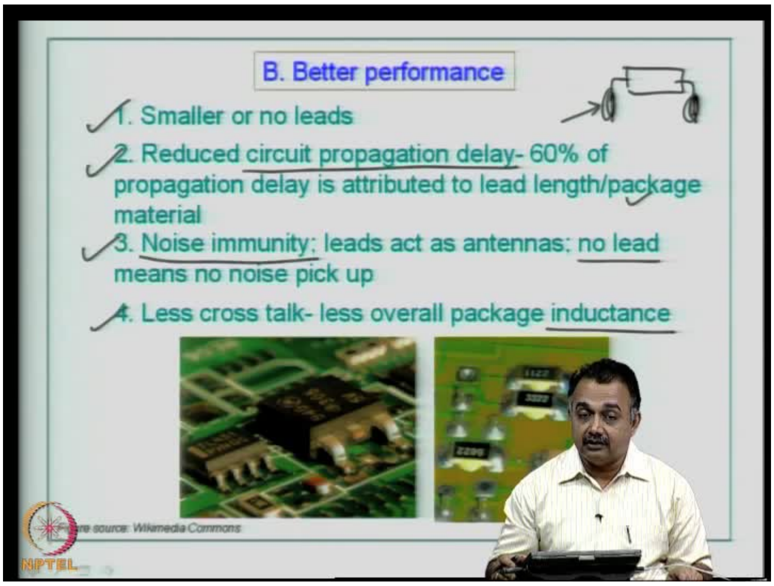
This is a surface mount device and this is the substrate like an FR 4 for example. Now, the components are placed on the surface and you can see that if assume that this component and this component are performing the same functionality, you can see there is a dramatic reduction in size and weight and this helps to utilize other areas on the board for other active and passive devices. Obviously, the density of this board is very high compared to a through-hole technology board.

Now, the other point which I want to emphasize here is that the ability to populate on both sides of the board. This is layer 1 surface mount, this is layer 2 surface mount and this is an FR 4 material. You can have a core here, you could have a core, and you could also have various multi layer generated through various technologies that we have seen. The top and bottom layers obviously are built keeping in mind that you are going to do a double sided assembly completely with surface mount devices. It could be a QFP, it could be a small outline package, it can be a capacitor, it can be a resistor, and it can be an inductor, all of them in surface mount format. This again will give you a classic view that surface mount technology helps in size and weight reduction, improves packaging density, improves reliability, helps you in reducing feature sizes of the board, and ability to pack components together.

The third point in this topic is additional space saving from elimination of component mounting holes. You can see here, this is a top view that you are seeing. You can see there is a drilled mechanical hole and the component is getting inserted through these holes. You can see that the space occupied by this component including the size of the hole, the annular ring, and then the track getting out from the hole area to the next interconnect will take up a lot of space. This is a lot of space compared to a surface mount device.

Here, which you see, there is no through-hole and basically you require two pads where the surface mount device is going to be placed. Because the device is smaller, the tracks also become thinner and therefore, this again emphasizes the point that the density increases once you move from through-hole technology to a surface mount device technology. A surface mount device technology enables you to think about how you can fabricate a system level Printed Wiring Board and how you can look in the market for various active and passive devices that can give you higher performance and that can match your thought about product miniaturization.

(Refer Slide Time: 23:31)



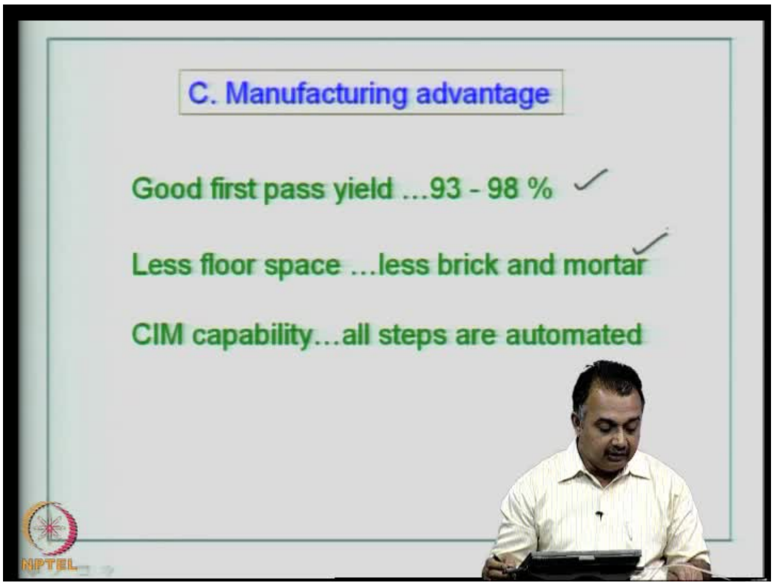
The second point in favor of the surface mount devices is that it gives you better performance. Because the first point is that it has smaller leads or in some cases no leads at all very close to the solder joint. So, you can imagine that the electrical performance will be definitely enhanced compared to a device which has long leads, because the space occupied by the leads and the reduction in electrical performance due to inductance or due to material losses or the signal propagation delay which is caused by the huge package size gives you lower performance compared to a surface mount device.

So, that is why I say that the second point here, which I mention, is very important for electrical designers. That is reduced circuit propagation delay. 60 percent of the propagation delay is actually attributed to the lead length or the package material.

The third point is noise immunity that means the long leads act as antennas and if you have no lead, it means there is no noise pick up, which means the signal quality, for example, in analog circuit or a digital circuit will be very different compared to a large size through-hole component in contrast to a smaller lead surface mount device.

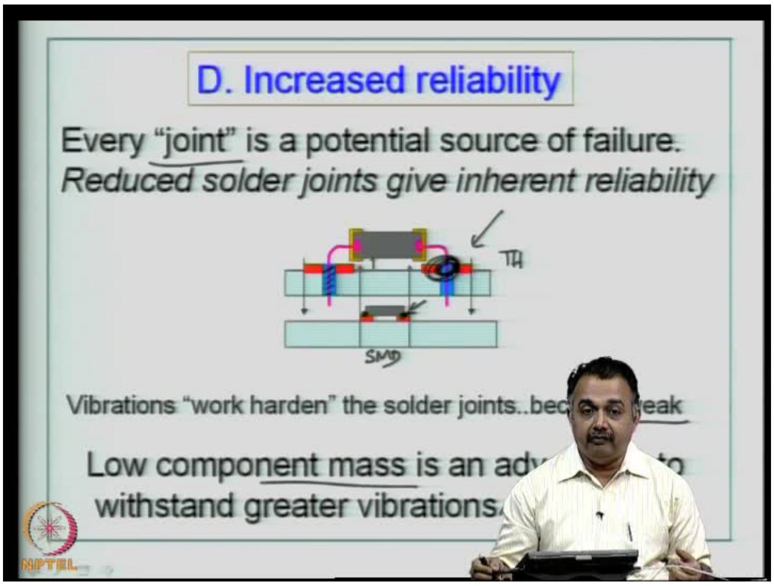
Then the final point here in this particular section is that there will be less cross talk that is because of low inductance and less overall package inductance, again because of no leads or smaller leads or the volume of material is much less, the inductances is low. Therefore, crosstalk which is an important parasitic that electrical designers have to encounter will be somewhat reduced. Therefore, as a designer you have to think about converting your through-hole designs into surface mount devices and if you are using surface mount devices look at packages that can enhance your electrical performance and of course that is not ending at the design stage, it ends up in the quality or the type of suitable material based system level Printed Wiring Board that you make. Finally, ending up at how or what method of attachment and what material you're using in the soldering process.

(Refer Slide Time: 26:25)



The third point is the manufacturing advantage that you have when you go to SMD, a good first pass yield, sometimes even up to 98 - 99 percent, the throughput in very high in each of the equipments that we are going to discuss shortly. Less floor space compared to a through-hole technology assembly process. Here, a good prototyping unit can basically rely on 3 or 4 equipments and then complete the manufacturing services. Then this particular technology has the advantage that it can merge well with your CAD process. Therefore, the computer integrated manufacturing is very much adaptable for surface mount devices, because all the processes that we are going to discuss can be automated. It involves less of human intervention and then the CAD process that you are doing especially the assembly part goes a long way in providing input data for SMD process.

(Refer Slide Time: 27:38)



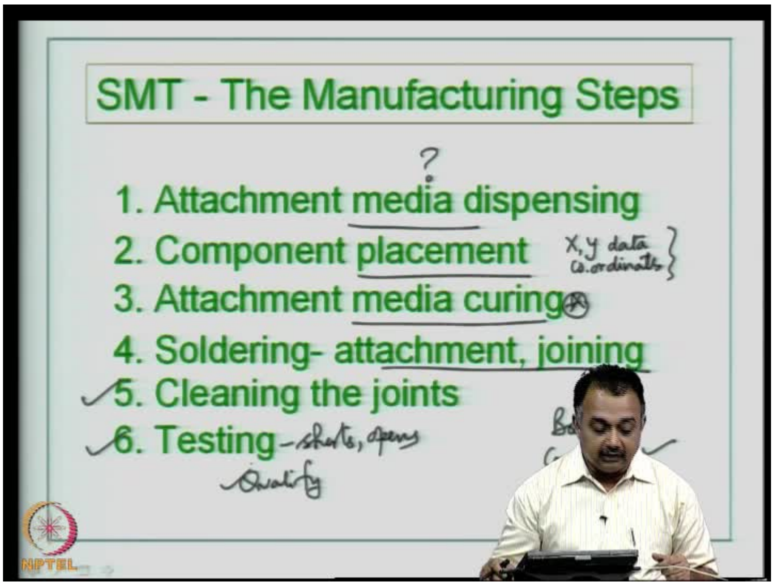
Increased reliability is another issue. Every joint that is your solder joint at the assembly is a potential source of failure, because as I have explained earlier, if your drop your mobile phone, let us say 5 times, 6 times, 10 times; one day it is going to fail. So, as an engineer if you look at what is the kind of failure; it is basically that mechanical shock has been transferred into a crack at the solder joint which has now resulted in an electrical open, a failure. So, a mechanical shock is finally getting into an electrical failure. So, the joint that attaches or joins the leads of the component with the surface of the board via the soldering process is very important and at the same time that is the potential source of failure. So, reduced solder joints give inherent reliability.

If look at the through-hole device, there is going to be a chunk of solder here, this is the pin, the device, and then the hole is filled with copper and finally, the solder material. A large volume of solder is getting added to the board. Such a large volume of solder obviously will in some sense is going to be a huge potential source of failure. You can expect failure, because there can be material behavioral patterns over a period of time based on the solder material and its interaction with copper on the board. Therefore reliably, reliability built over a period of time has to be simulated or measured experimentally.

In the case of surface mount devices, you can see again illustrating that the area is small and since it is assembled on the top, the pads are small, the devices are small, and therefore, the volume of solder or the attachment material will also be less and the vibrations work hard on the solder joints and therefore, they become weak. This is true with a through-hole component. Imagine a through-hole, here it is a through-hole component and there can be vibrations in an equipment or part of a product in its application. Because you have long leads with long surface area in contact with the copper and then the solder material and the through-hole during vibrations over a period of time this can work harden the material and they can crack or induce fatigue in the solder joints. Therefore, they can become weak and can yield resulting in a failure.

In contrast through a through-hole, you have the surface mount device, which has got low component mass and therefore, this can be considered as an advantage to withstand greater vibrations. There has been a question over the suitability of surface mount devices for applications where it can experience vibrational shock. It has been proved that surface mount devices and surface mount assembly can qualify very well in those application areas.

(Refer Slide Time: 31:16)



We look at the SMT- Surface Mount Technology manufacturing steps. Here you remember the board is ready, the system level Printed Wiring Board is ready, the components are identified, components are known, all you are now going to do is use these and do an attachment process which is now known as the SMT process.

The first thing is attachment media dispensing on the board. If you take a board like this, imagine this board was manufactured, the components are ready for this process, and how do these components get attached onto the board. If you look at the table here, the first process was attachment of the media or dispensing the media. What is the media here? The media is the one that holds the component onto the board. We are worried about what kind of media we are going to use.

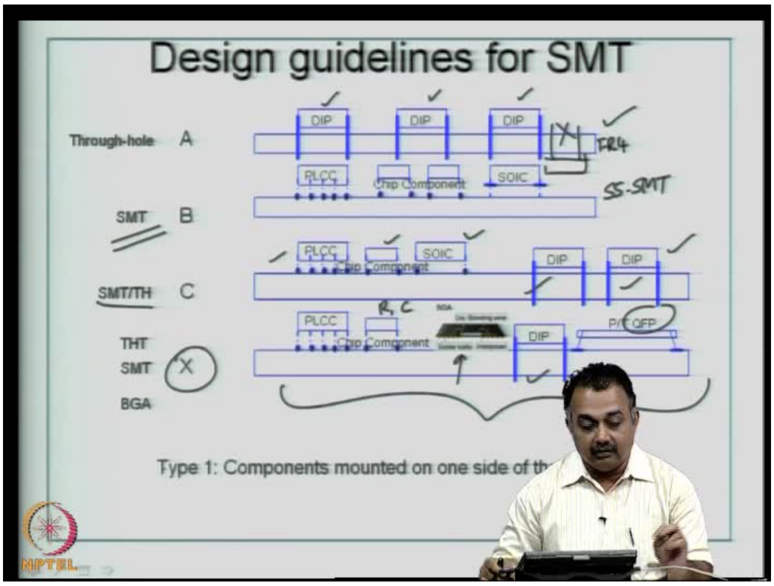
Then, the next one is the component placement. The right components are to be placed in the right spot. Your CAD will give you the x y data or the x y coordinates of the various components to be placed on the board. Use that information. You can use manual placement or you can go for an automated placement, if it is a very large board and if it is for volume manufacturing. When I say manual placement, it is very suitable for prototyping.

Then the media that you have attached has to be cured. This curing process will ensure that the component is firmly attached in its position. Then, you introduce the solder material. This will help in attachment of the component and provide the electrical connection. This also is known as joining process or joining method.

Once the joining process is over or the soldering process is over, you clean the joints because the media as well as the solder material that you are introducing could have extraneous materials, could have some impurities like oxides and other metal particles that can be there when the individual materials are manufactured. Once the joints are cleaned, then it can be tested for shorts and opens. Then, you can qualify this particular board. The testing process will qualify the board for marketing.

In a product typically, you might have 1 2 or 3 such boards and then they are interconnected through connectors and so on or it can be a sub rack assembly and then it will be part of a huge system. So, individual boards will be tested, individual components will be tested, in some cases, sometimes electrical measurements will be made for specific components while it is powered up. Therefore, all such electrical tests will be carried out before the board is released for further processing.

(Refer Slide Time: 34:58)



We will have a look at the design guidelines for surface mount technology. As a designer, you will be encountered with various questions and process details that you have to think about when you encounter different types of components. These are some examples that I can give.

For example, in the top figure you see here, it is known as through-hole where you have a substrate FR 4 for example, very simplest to remember. There can be many other substrates, as you know. Then, you can mount these through-hole devices, DIP packages, for example. Many other examples can be given. For simplicity, we will identify through-hole components with the DIP packages. DIP packages are as you know large sized components, dual rows and you can have many pins but the leads are long. You can see here the assembly is done on one side that is components are mounted on one side of the board. When you have such a configuration you will hesitate to mount or it is forbidden to have through-holes from the other side. This is not permitted. You will do assembly only on one side when you have complete through-hole components.

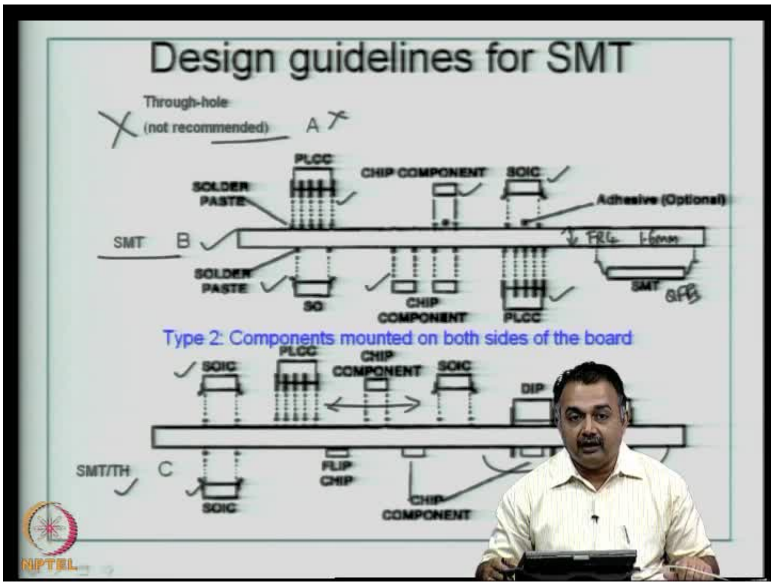
The second example is surface mount technology, where you see on the surface, you have different types of surface mount components; small outline IC, chip component, resistors, capacitors, then you have the PLCC that is ceramic carriers, all of them are placed on the surface. On the other side, there are no components placed. This is a single sided surface mount technology board.

The third one is a combination of surface mount technology and through-hole. The assembly is done only on one side of the board not on two sides. You have in some sections of the board, you have the through-hole technology, and on some sections of the board, you have surface mount devices. These are the surface mount devices. You can have a very good mix of assembly procedures or components and devise your assembly procedure accordingly. As I said, the joining process is going to be done at some temperatures. Because the material has to fuse, has to melt the solder material and therefore the component should not get adversely affected by repeated thermal shocks. So, this is permitted.

Then, the third one is that you can have a through-hole technology on one side of the board; you can have surface mount devices like your plastic or thin PQFPs, quad flat packs, chip component resistors and capacitors, for example, PLCC, you could also have BGAs and advanced package like a BGA and all of them can be done on one side of the board. I am describing situations for a designer where you can consider assembly on one side of your system level Printed Wiring Board using different types of components. In reality, you are going to experience these kinds of situations. You can go ahead if you feel you are in these kinds of entries.

Through-hole surface mount exclusively, a mixture of through-hole and surface mount. Then you can have through-hole, surface mount and BGA. When you use these kinds of complexities as defined in this, then you have to really think about the thermal shock that your board will undergo. As we are now going to see the soldering process and temperatures, then you have to really look at removing some of the components from one side of the board into the next side so as to minimize the thermal shock.

(Refer Slide Time: 39:38)



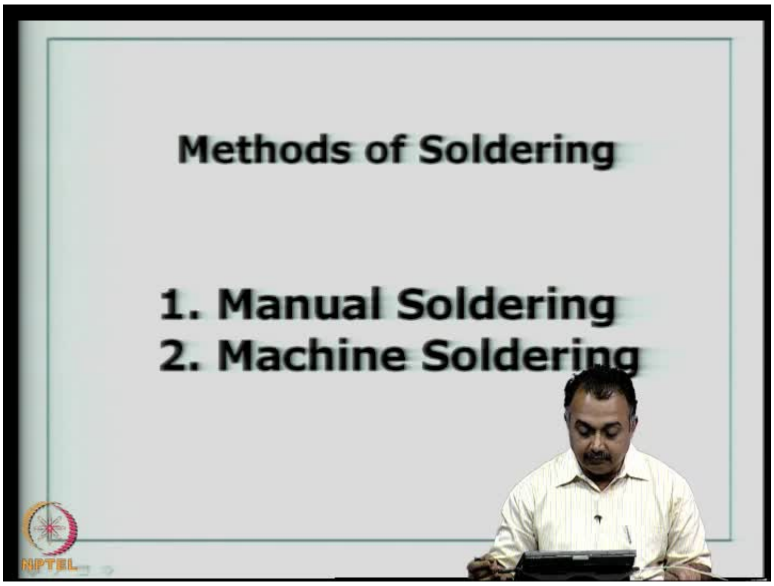
The next slide will obviously be how to look at opportunities or avenues where you can mount the components on both sides of the board instead of having everything on one side of the board. Having everything on one side of the board also increases the board area. If you plan at your CAD stage, you can reduce the board size by having double-sided assembly. The first thing that I want to mention is that you cannot have a through-hole, double sided board. Not recommended. So, type A, if this is the classification that is ruled out.

Then, the second one is very simple. You can have double-sided surface mount that means on one side you can have PLCC, chip component, resistors, capacitors small outline ICs, then on the other side, different types of QFPs, for example, quad flat packs again PLCCs, chip components, small outline packages. Even if you mount them exactly below the other component on the other side, the process is not going to affect the other side components, because this thickness of FR 4 material typically let us say 1.6 mm dielectric material is going to take care to some extent. The processes of soldering or attachment will really make sure that the peak temperatures for solder fusing and attachment will not affect the components that have already been mounted on the other side. So, this is accepted.

The other possibility is you can have surface mount technology on one side and also surface mount technology can mounted on the other side as you see here, small outline

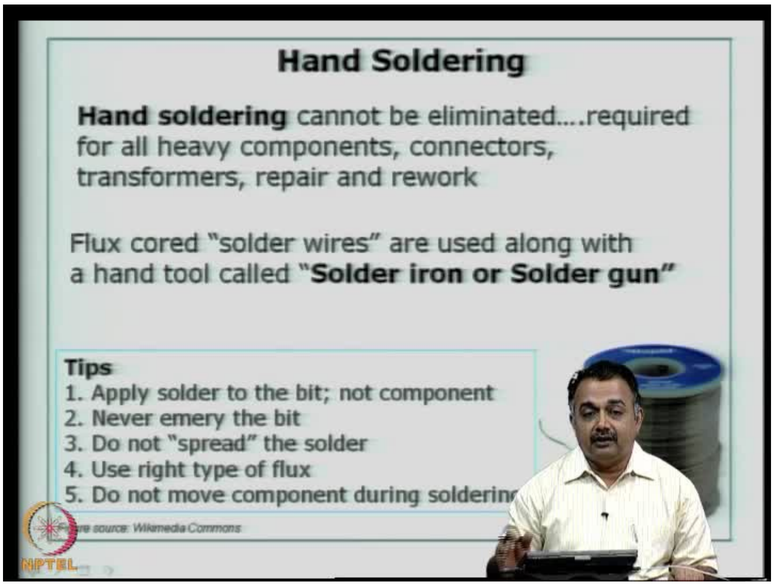
IC, small outline IC, all these are surface mount. You can have DIP packages on one side. Do not have DIP packages on the other side. When you have such a combination, the only problem is when you have these DIP packages; it poses a restriction for you to mount very large surface mount devices in these areas. Typically, you can use chip components and smaller SMD resistors, and capacitors in these areas so as to occupy or utilize the real estate. The second option becomes a bit tricky but with experience one can really design such kind of hybrid boards.

(Refer Slide Time: 42:27)



Let us go to methods of soldering process and in this process we will describe how individual processes can be thought of by a designer in producing boards that are of high reliability. The first one is manual soldering; the second one is machine soldering.

(Refer Slide Time: 42:52)



The case of hand soldering, many of you would have done hand soldering in your course lab or otherwise or working on boards as a hobbyist. You would have used soldering guns or solder iron and used hand soldering method. You would typically heat up your soldering rod or iron and then try to fuse your components along with solder wires and then make an attachment or a solder joint as you call it.

Hand soldering typically even for volume boards cannot be eliminated, because let us say if you are having a volume board for a desktop computer and 99 percent of the components are surface mount BGA, CSP and so on and then you may have one connector, a very small connector which you want to do by hand soldering. You can do it. It is highly reliable and it is recommended that in such cases where you have one or two components, which are of through-hole technology; then you have to proceed with hand soldering and there are experienced technicians who can produce a very good quality hand soldering solder joint. These are required for all heavy components, connectors, transformers, and typically you will also hand soldering for repairing and reworking your SMD devices.

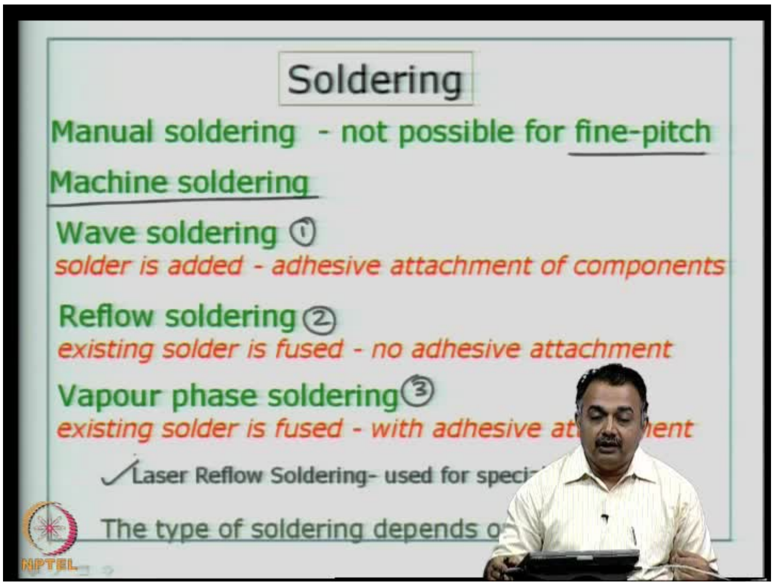
Even for the BGA component, sometimes you will have to use very specialized hand soldering guns or solder irons to remove the components and reattach or rework on them. We will look at those kinds of hands on videos as we go by.

You are going to use solder wire. You are going to use hand soldering. These will be flux coated or flux cored and they will be used along with a hand tool called solder iron or solder gun and the tips typically we are not going to discuss hand soldering in this session much.

But in this particular slide, I would like to describe that apply solder to the bit not to the component, never emery the bit, do not spread the solder, because you can establish solder bridges if you do not set the right temperature and if you use too much of solder wire material, use the right type of flux on the board before applying the soldering wire or in today's soldering wire most of them are coated with flux material and therefore, you do not have to apply flux separately and do not move the component during soldering process, observe the right temperatures that do not affect your board surfaces because we have seen what a Tg is the glass transition temperature.

If you are using FR 4, typically the glass transition temperature will be between 140 to 170, so you need to know as a manufacturer or assembly person, what is the Tg of a material and work within the limits. Obviously, you need to know the melting temperature of the solder material, because today you have different solder wires based on different materials. Therefore, Tg on mind and the soldering temperature in mind you need to work at the right optimum temperatures.

(Refer Slide Time: 46:30)



Manual soldering is not possible for fine pitch. Never even think about manual soldering for fine-pitch and you need to rely on automated equipment.

In the case of machine soldering, there are different types. The first one is the wave soldering, which is still active today. But when through-hole technology was very prevalent, wave soldering was an important method of attaching the through-hole components. I will describe the wave soldering process a bit in detail for those who are still involved in through-hole technology.

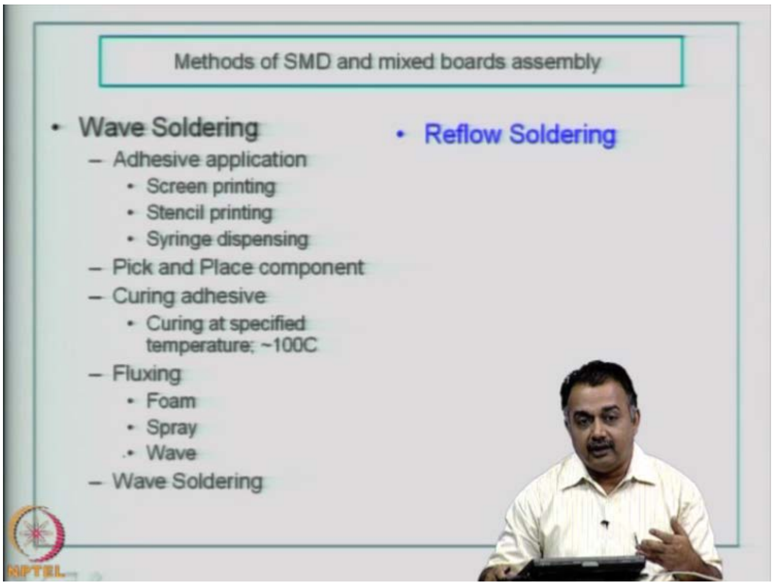
Reflow soldering is a process typically used for surface mount devices, a very popular form today used globally by industries. In reflow, you will see there are different methods and here again the key to reflow soldering process is that the temperatures are very important and an understanding of the substrate material, the peak temperature of reflow, and the component status, that is some components can have some restrictions on temperatures at which they can be used. Please read the details in the component data sheet about what temperatures a particular component can withstand.

Then the third method is vapor phase soldering. In the first case of wave soldering, solder is added to the board. You attach the components on the board by means of an adhesive; it is not a conductive adhesive, it is a basically a glue nonconductive glue and then solder is added to those areas required to establish the solder joints. In the case of reflow soldering, existing solder is fused that is why the term reflow is used. You are attaining the temperatures where the solder melts again. So, it is known as reflow. The material flows and then fuses with the solder, with the component leads on the surface mount devices. So, there is no adhesive attachment. The solder material itself will have some kind of glue that will take care of positioning of the component in the right place.

In the case of vapor phase soldering, again the existing solder is fused with adhesive attachment. In addition that the existing solder is fusing with the leads, in order that the components do not move from its coordinates, we use an adhesive attachment. Because this is going to enter into a different kind of a soldering process where you are utilizing the vapor phase of a solvent and then trying to use the heat from that material to reflow or attach the components.

In very few cases like prototyping, Laser Reflow is used typically used for special devices where it cannot be accessed or it is very difficult to access using the above three processes. Special devices with 3D kind of boards and so on, you can use some kind of a Laser gun and produce the heat energy that is required for solder reflow. The type of soldering depends on the design. Now, as a designer I have talked about the design for manufacturing. You need to understand for your component choice what kind of soldering process you need to specify to the manufacturer, because the type of soldering process as I said and I emphasize again, the temperatures are different, the materials could be different and you try to aim at reducing the thermal shock in your board. So, if you have no other choice but to have different flavors of components like BGA, SMD and through-hole, then you have to really think about minimal thermal shock.

(Refer Slide Time: 51:02)

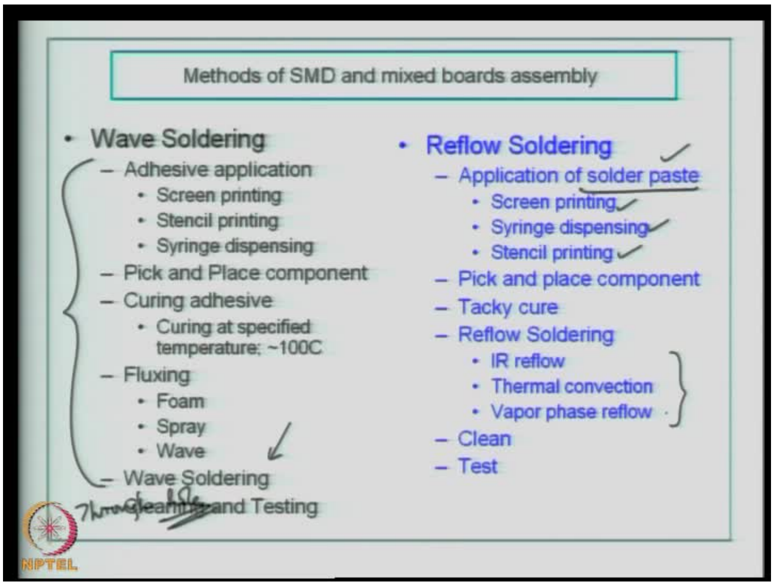


Now we will look at the methods of SMD and mixed boards assembly. The first one is wave soldering. We will see the process steps and then we will look at reflow soldering process steps.

In the wave soldering process, the first step is adhesive application. Typically a nonconductive glue, an epoxy material can be used. That is the very simplest economically viable material that can be used. There are different other adhesives. But the cheapest for the industry is epoxy based. It can be applied by screen printing process. We have seen screen printing earlier. Stencil printing by using stainless steel or nickel stainless steel stencils and as you recollect these stencils are prepared by electroforming or by photochemical milling, so that you can get very fine lines and packed sizes.

The third one is syringe dispensing. The advantage of using syringe dispensing, by using normal syringes that is used in medical applications is that you can use various capillaries. You can use hand dispense or automated syringe dispensing equipments that are available. So, you can utilize them. The next process after the adhesive has been placed is that you do a pick and place of the component based on the design that you are working with. It could be active devices in plenty, passive devices, electro mechanical components, transformers, and connectors and so on. So, it could be manual or it could be automated.

(Refer Slide Time: 53.15)



Once the placement is done, then you cure the adhesive. Curing is done at specified temperatures that are again based on the adhesive that you have used. Typically for epoxies, you will work with around 100 degree Centigrade or 90 to 100 for about 10 minutes that will keep the component in place. The reason for keeping the component in place is that the next process is the soldering attachment. Solder is added to the board and during that process your component should not deviate from their coordinates.

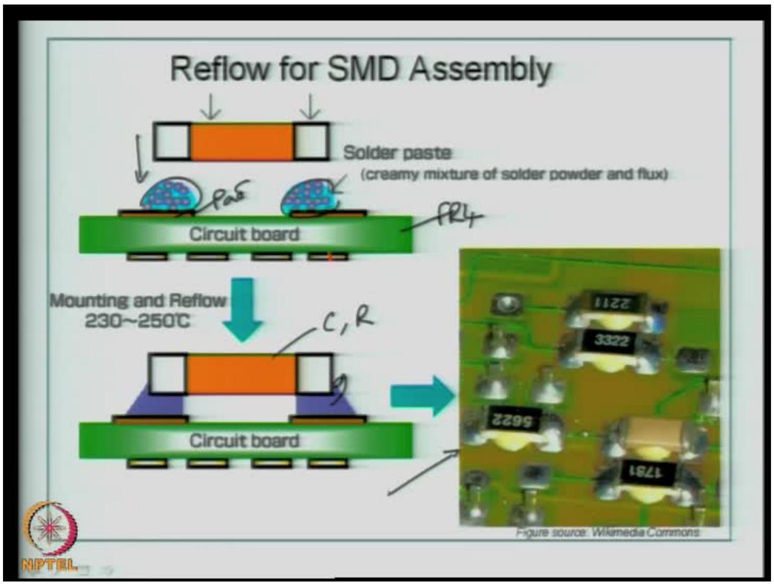
Before that, a fluxing is done. Fluxing can be done by foam fluxer, spray fluxer or a wave fluxer. In typical wave soldering equipments, the first process could be fluxing wave and then it goes to a solder wave. During the wave, as the name indicates, there will be a wave of the solder and this wave will get in contact with the component leads and due to the wicking process the solder gates attached to the component leads and a solder joint is established. This process typically is very ideal for through-hole boards. But today, if you have the capacity to monitor, you can also apply wave soldering for SMDs. With the advent of free flow soldering process, most people do not use wave soldering for SMD process. Then finally, you have cleaning and testing in as in any process.

We now look at the process steps for reflow soldering. The first one is application of solder paste. You see here in comparison, you are adding this solder from the wave; here you are dispensing the solder paste. So, solder is attached. There is no glue attachment and this is done by screen printing, syringe dispensing, or stencil printing. The solder paste is basically a media which contains very fine particulates of the solder material. It could be tin lead; it could be lead free that is, tin-silver-copper, or tin-indium or tin-silver.

There are different flavors of solder material and each of them have different peak reflow temperatures that you have to be aware. So, simply choosing a material will not help. You need to understand the components. So, this paste will contain particulates of the metal powders that you intend to use. Then, you will have a binder material similar to an adhesive so that helps in keeping the component in place when you dispense the solder paste. Then it will have a media like an epoxy; so that you can use processes like screen printing or syringe dispensing and stencil printing to flow the material. So, a right combination of metal particulates, the media like epoxy and a binder will comprise the solder paste material in addition to flux. Flux is already included in the solder paste unlike in the wave soldering process, where you have an opportunity to flux the board. Fluxing is basically a process to remove the oxides and to provide a wet joint to the board. So, this is an essential material. But flux again creates problems because unless you clean the board after the soldering process is over, you will end up with a lot of oxides which can interfere with the tracks, because they will induce corrosion on the board.

Then, you pick and place the component. It can be manual or automated. Now, you do a process called Tacky cure, because the ingredient in the solder paste, that is the glue will need to cure. At this stage of time, the solvents and the epoxy are not going to be affected but the adhesive will bind the component onto the surface of the board. Then you do the reflow process by either of the methods it can be an infrared reflow process, thermal convection process or a vapor phase reflow. We are going to look at each of these processes in detail. Once this process is over, you can clean and test the board. Cleaning is essential. If you have used fluxes that can spray oxides or other material onto the board after the reflow process is over. Wave soldering and reflow soldering process are the key processes in defining the reliability of your surface mount board.

(Refer Slide Time: 58:01)



We will now look at a typical pictorial description of a reflow process. What is meant by a reflow for a SMD assembly? You have a circuit board, typically let us say FR 4, and then you have the pad that is defined. Then you dispense the solder paste by any of the processes that I have described. Then you introduce the device, surface mount device. Then you mount and reflow the board so that the solder paste melts and it fuses with the leads of the device. This is a typical example of a capacitor or a resistor format and then you can see the kind of joint that you get. This is described here in this picture; well defined wet joints produced by reflow process. We will look at in the next class further points on wave soldering and reflow process.