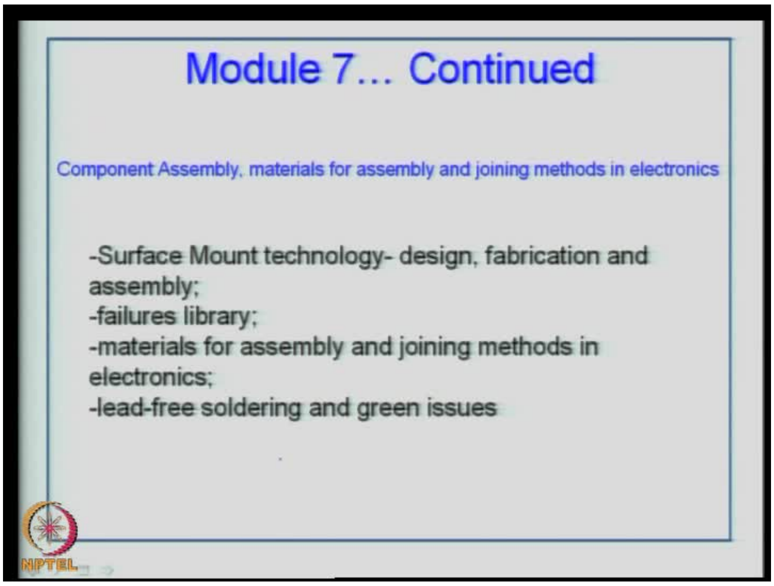
An Introduction to Electronics Systems Packaging

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

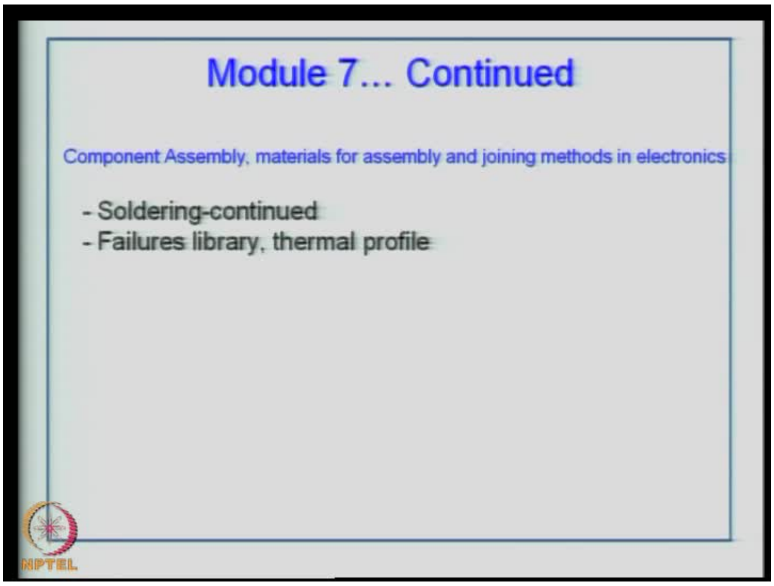
Module No. # 07
Lecture No. # 35
Vapour phase soldering
BGA soldering and De-soldering Repair
SMT failures

(Refer Slide Time: 00:15)



We will continue with module 7 of this electronic systems packaging course. As you are aware in module 7, the focus is on component assembly. The component can be a through-hole, the component can be a surface mount device, or it could be an advanced package like a BGA or a CSP or a flip chip.

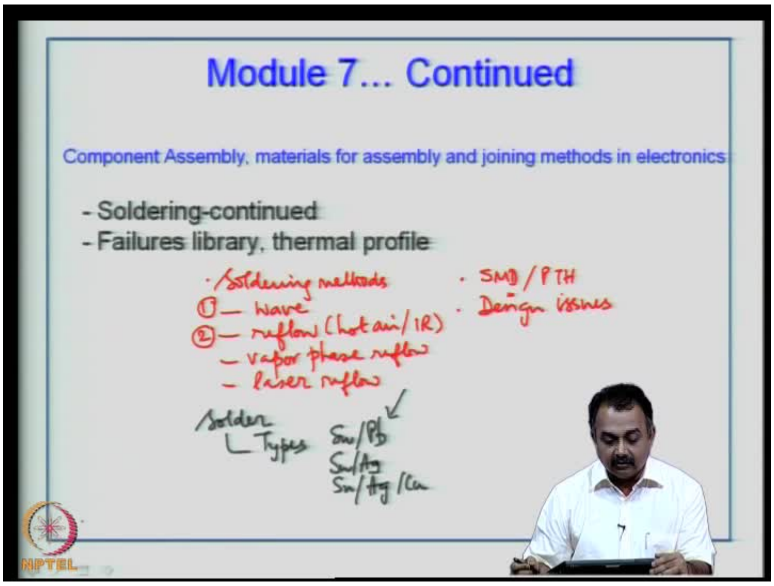
(Refer Slide Time: 01:00)



Flip chip on board is considered a very advanced form of connection although flip chip is in itself a first level interconnection choice. In this chapter, we are looking at materials that are being used for assembly purposes. Typically, the focus is on solder alloy that is used for joining components on the board. This becomes very important because joint failures have been seen in many design and assembly issues. The reason for them could be the joining material, that is, alloy or it could be the nature of the substrate, the nature of the component lead, whether it is a surface mount device or a through-hole component and so on. The failures could also be due to the soldering or joining conditions, whether it is manual or whether it is equipment based or machine based operations. Remember, in this course, we are trying to teach you how you can do these joining methods in your lab in a prototyping unit. At the same time, we are also looking at issues in terms of reliability, in terms of failures that can be seen in large volume production.

So we will continue with this chapter and I just want to give you a brief list on what we have essentially seen. We have basically seen the SMD components and the through-hole components. We have seen the design issues for soldering. We have also seen the soldering methods. We have had a look at wave soldering. We now know what a reflow soldering process is. There is another process called vapor phase soldering and then the last one is called a laser reflow soldering. Vapor phase is also a reflow process. Typically, you will have wave and reflow; and in reflow you have hot air or IR based reflow processes.

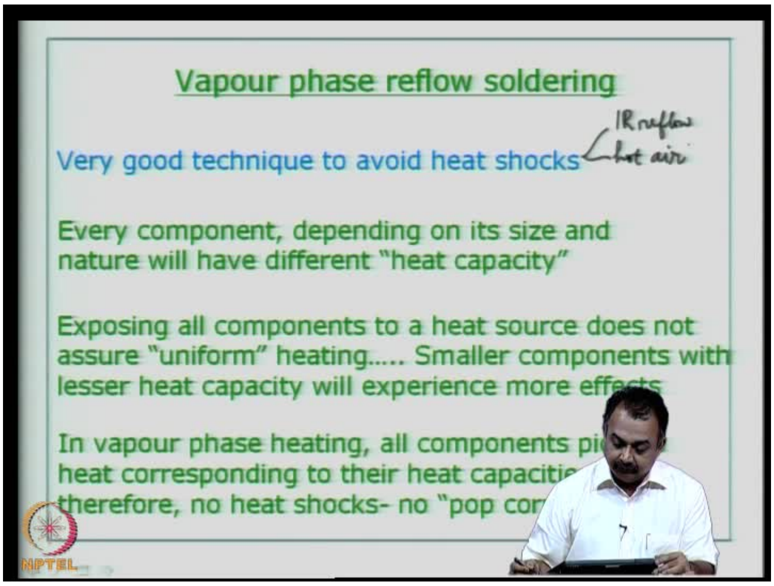
(Refer Slide Time: 03:30)



Then, we have seen the term solder and we know what type of solder materials are being used today. Tin lead was the obvious choice. But today with the elimination of lead, we are looking at other materials like tin silver or tin silver and copper alloys. We have also seen the methodologies for wave and reflow soldering. That is the process steps that you should go about in bringing an effective wave or a reflow solder.

So, at this note, we will continue today's topic. Today, we are going to talk about vapor phase reflow soldering process. This is a very good technique to avoid heat shocks. This could be considered as an alternative to IR reflow or hot air, that is, convection based hot air reflow processes.

(Refer Slide Time: 04:47)



Remember, I keep emphasizing in this particular module that when you do an assembly, where you have a surface mount device that needs to be mounted on a substrate, you need to know the nature of the components that can experience heat shock, thermal shock during the assembly processes. For that, you need to look into the data sheet of the component and they would have specified that the component, in some cases, can experience the thermal shock and therefore, it is not recommended to expose that component above a particular temperature for a certain period of time. That information needs to be gathered by the designer as well as implemented by the assembly personnel.

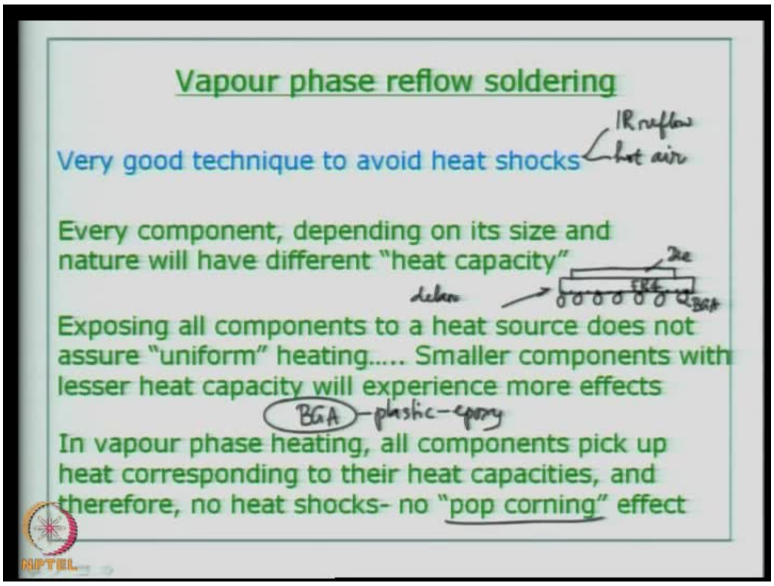
Now, in IR reflow or hot air reflow, the thermal profile has been set and you reach a peak temperature at which the solder melts, reflows and then it attaches itself to the component lead. Now, in this scenario, there are various components on the board and different components will experience different thermal shocks because the materials for each of these components can be different. You can have a plastic package, you can have a ceramic package, and therefore, the heat capacities of these components based on the materials that are used will be different.

So, every component, depending on the size and the material choice will have different heat capacities. We never utilized this property when we do an IR reflow or hot air, because sometimes there is excess temperature above the reflow and it can end up with defect in the component package. Exposing all components to a heat source does not assure uniform heating. Therefore, smaller components with lesser heat capacity will experience more thermal effects.

What are the thermal effects that we can see? There could be a package crack leading to a die crack. There could be failures at the solder joints because of the excess temperature and based on the solder volume, the excess temperature, if the pitches are very small, could cause failures in the joints.

How do you avoid that? Vapor phase reflow is a very good technique because in vapor phase reflow soldering process, all the components pick up the heat corresponding to their heat capacities and therefore, no heat shocks. This, we call as the pop corning effect. As you can see here, the term pop corning effect is used. What is a pop corning effect? If you take a BGA component and if it is a plastic component, basically the plastic material that is used is epoxy and epoxy is known to absorb some kind of a moisture.

(Refer Slide Time: 08:36)

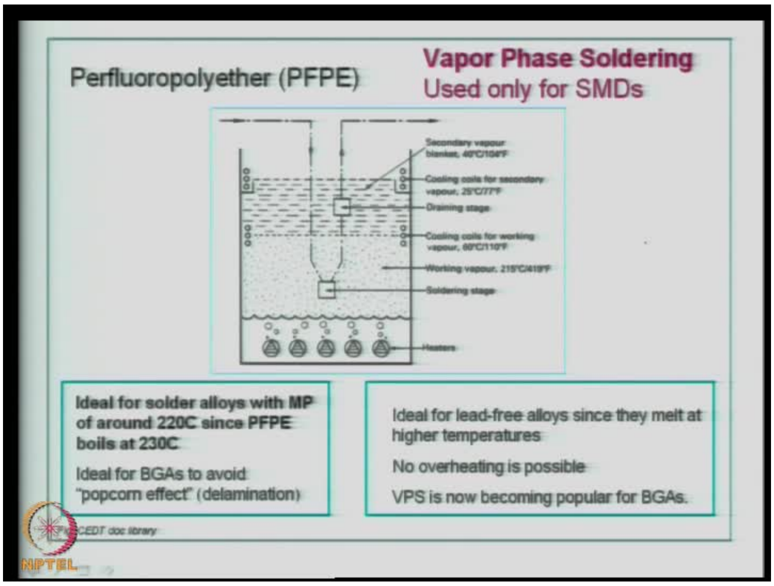


Therefore some manufacturers of packages generally recommend that you preheat the BGA package before it goes for assembly to remove the moisture. What happens if you do not remove the moisture and if you expose it to reflow soldering. For example, this is the BGA package. What I am drawing here is the solder ball, the ball, grid array and this is the substrate, let us say, this is an FR 4 substrate and then you have the die. This is the cross section of a BGA package. There will be inherent moisture because of the nature of the epoxy material. Therefore, a sudden thermal shock that means you do not expect that there is so much moisture entrapped. Now, if you set a wrong thermal profile, the moisture will now evaporate or get removed from this area, that is the FR4 epoxy area, and then the solder ball area, there is entrapment of moisture. So, this thermal profile will now make this moisture burst out in the form of gases and then this will cause delamination.

So, a pop corning effect means you can literally see this pop corning effect. If you carefully observe through a video camera, when you do a reflow soldering process in a package in which you are sure there is moisture. You can see the separation of the solder balls from the FR4 substrate. If you do not notice it, then you will see failures and obviously you have to replace this package.

Therefore pop corning effect is an inherent problem in BGA soldering especially in plastic packages. Based on the plastic material that is used, the encapsulant or the organic substrate that you are using, it is better to preheat, it is better to prepare this package for reflow soldering instead of directly using it from its normal storage position place.

(Refer Slide Time: 11:14)

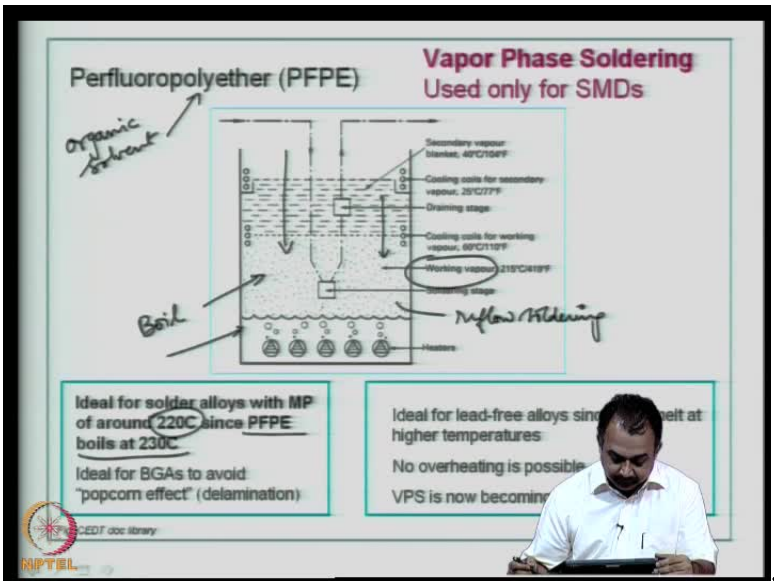


Let us describe what vapor phase soldering is. It is used only for surface mount devices. Typically, as you can see in this figure, there is a container and then you have the organic solvent. So, basically you are going to use an organic solvent. The organic solvent that is used is perfluoropolyether - PFPE. Now, the organic solvent will be heated at a particular temperature, where it can boil. Basically you have to boil the organic solvent, so that you get the vapors. You can see here; these are the vapors. Now, your board, which is picked and placed using machine pick and place process, using a solder paste so that it is tacky cured and the components are in place, is slowly introduced in to the working chamber, where the vapors are ready.

Now, the vapors will be in contact with your Printed Circuit Board and then based on the heat capacities of the components, sufficient heat will be observed by the components. You can see for example, PFPE boils at 230 degree centigrade. Suppose, if you are using a solder alloy with melting point of around 220 degree centigrade, so you have a window of plus 10 degree centigrade.

Now, the reflow process takes place and all your components are attached during this particular reflow process. This is a process, where you typically do not set a thermal profile in a heater or IR heater. You are not worried about the air circulation in an oven and so on. Basically, you are getting ready in a container with an organic solvent, vapor. Vapor typically condenses back into the same chamber. So, you have a condenser that is usually present in this so that it condenses quickly back and again it is heated. Therefore, you get a constant sufficient area that is available for the vapor to reflow the solder paste material and enable the reflow soldering process.

(Refer Slide Time: 14:00)



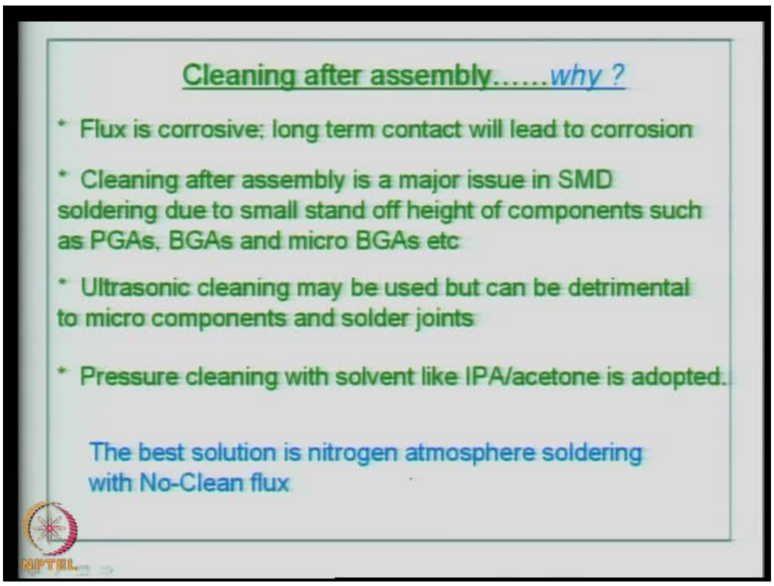
This is very ideal for BGAs to avoid the popcorn effect. In the Printed Circuit Board the assembly process can be done for a few boards depending on the area of the chamber. The reason why we say that this is ideal for BGAs is because you can work at temperatures that are not detrimental to the substrate and the solder ball attachment.

Now, this process is ideal for lead free alloys, since they melt at higher temperatures. Therefore, you can fix the entire process at 230 C and most lead free alloys have melting points around or less than 230 C and therefore, you have a very ideal situation for working with lead free alloy solder phase.

Now, there is absolutely no issue of overheating, which is always possible with the IR or hot air reflow soldering process. That is why we have to work with thermal profile, which we are going to discuss later today.

How, sometimes, even of the best thermal profile, you do not get very good joint? Vapor phase soldering is catching up very fast. It is being tried for mass production and therefore, this is the very good alternative for IR or hot air. In fact, IR is much below in terms of choice compared to convection based hot air reflow soldering. So, this is an ideal situation. This can be used for prototyping and we are simply using the basic concept that various materials have different heat capacities and that could be used for transferring heat from the vapor to the component and thereby to the lead and the solder paste to get a very good wet joint.

(Refer Slide Time: 16:30)

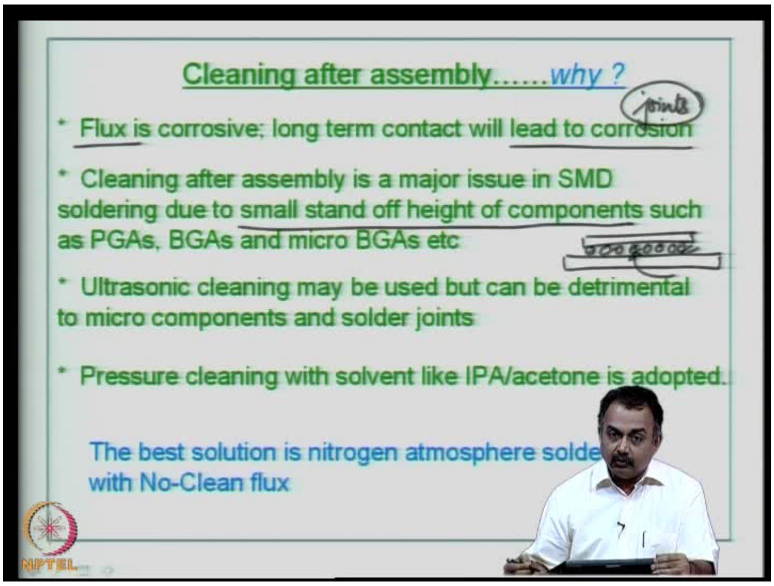


Now, having seen all the reflow processes, let us look at an issue whether we have to really clean the board after assembly. Is cleaning really necessary? Yes, because, as we know, flux is being used in the solder paste, it is corrosive. We have to use flux because we have seen the need for using flux. A long term contact will lead to corrosion. Although flux is used to remove oxides and clean up the copper pads, residues of flux after the reflow process can lead to deposit of various by-products, which in the long term, will lead to corrosion at the joints.

So, you can have joint failure caused by flux residues. Therefore, you can clean the board after assembly. It is the major step because let us say after one year or two years, the board is analyzed and there may be problems due to shorting of tracks or shorting of track caused by debris due to flux and a nearby joint which is also affected due to formation of an electro chemical cell. These kind of issues can be expected, if the board is not cleaned.

So cleaning after assembly is a major issue and especially when you are using small standoff height of components that means components which are low profile and you cannot really find out if there is a flux residue under the component especially in a BGA. This is the board, so you can expect because this is the large area package after reflow soldering you really do not know the kind of contamination that has taken place under the BGA package.

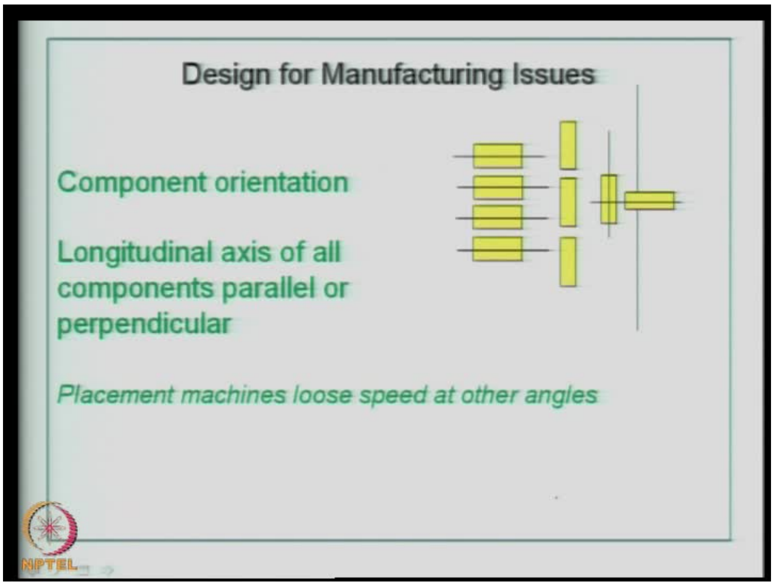
(Refer Slide Time: 18:45)



It could be even in the case of a surface mount chip capacitor. So, we are really concerned about contamination under the package. Sometimes, it is possible to easily remove the contaminants outside of the package area on the board. Ultrasonic cleaning can be used, but sometimes ultrasonic cleaning for a long time could be detrimental because it could affect the micro components, small components and the solder joints due to vibrational shock. Although this is a suggested method, people would love to use pressure cleaning with solvent like isopropyl alcohol or acetone. In fact, IPA mixture with water is also adopted. So, cleaning after assembly is crucial. Only question is how you clean it so that your newly formed joints are not affected, especially by ultrasonic cleaning. There again, the timing is very important.

The best solution is working with nitrogen atmosphere. Many people have been using the reflow soldering process in nitrogen rather than in air. So, this could be considered as an option to air reflow and also using No-Clean flux as we have seen when we discussed the flux materials, there is a material known as No-Clean flux. So, if you look at the data sheet of No-Clean flux, what it says is, if you operate the reflow process at a certain temperature, it is guaranteed that there will be no flux residues. So, typically you do not have to clean the board at all, when use a No-Clean flux, but No-Clean flux could be expensive.

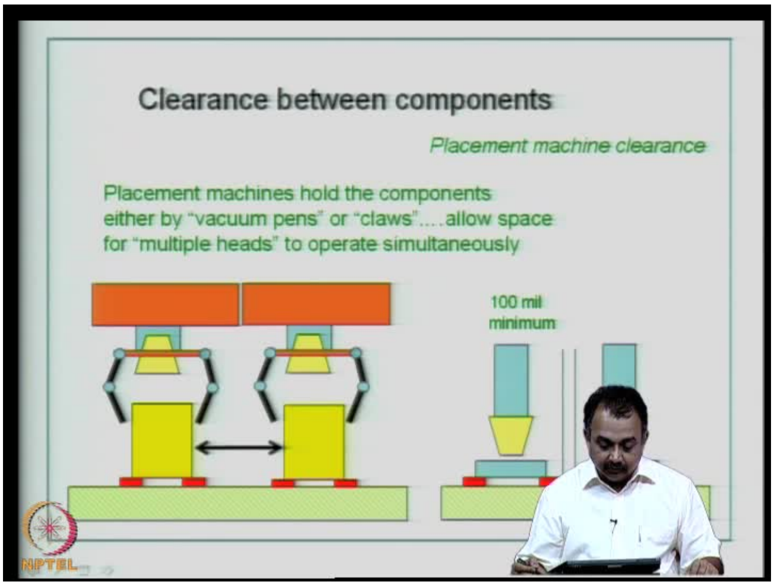
(Refer Slide Time: 20:54)



Now, we look at some design for manufacturing issues. Having seen all the processes, these are couple of points that I am going to discuss, which could go a long way in increasing the yield in large volume manufacturing.

For example, if you are mounting surface mount devices and if you have the components placed at random like this; this (Refer Slide Time: 21:19) is in one direction, this is in another direction; what is going to happen basically is that the placement machines will loose speed, if you work with different angles. It is better to maintain either this state or this state depending upon the population that you have on the board of these smaller or even larger components. It also helps in testing. It also helps in assembly because you can expect tall component's shadow not affecting these smaller components. So, longitudinal axis of all components should either be parallel or perpendicular.

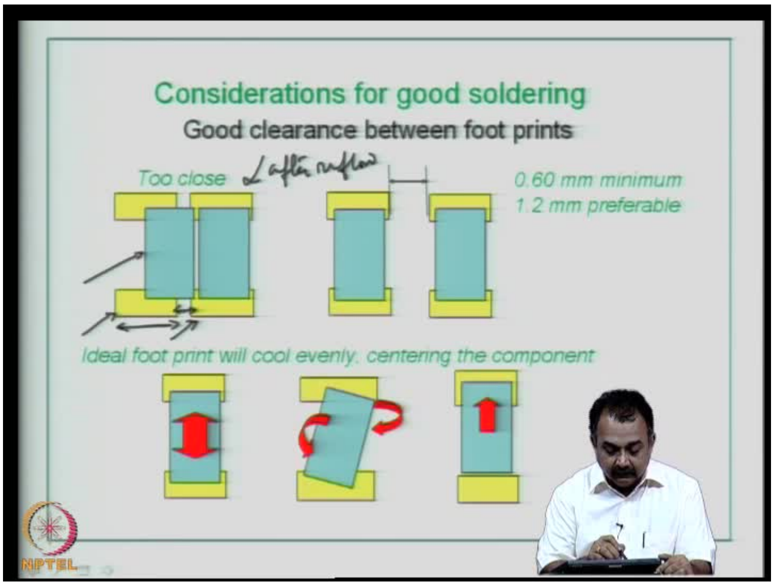
(Refer Slide Time: 22:07)



This is a design issue. If you look at clearance between components, again we are talking about a placement machine clearance. In some cases, the placement machines hold the components either by vacuum pens or claws as you call it and that requires a lot of space, as you can see here and it could be a very tall component. Therefore, you have to give a gap between tall components and it is not advisable to place any components here in the gap.

Look at the kind of equipment that you have; look at the type of component that you are using; and look at what clearance you can give between components so that repair and rework can also be easily done. There could be situations when you want to remove this component and you will have to use special vacuum tools to pull out the component and heating only that particular component for de-soldering.

(Refer Slide Time: 23:30)



Therefore, give a clearance, as we see here, like 100 mils minimum and therefore, it becomes easy for placing or for repairing de-soldering. There are a couple of points, again, that we will discuss before we go further. Considerations as a designer for good soldering: Good clearance between the foot prints. This is the foot print of one device. This is another one. Look at the distance between the two. Is it sufficient for a package like this?

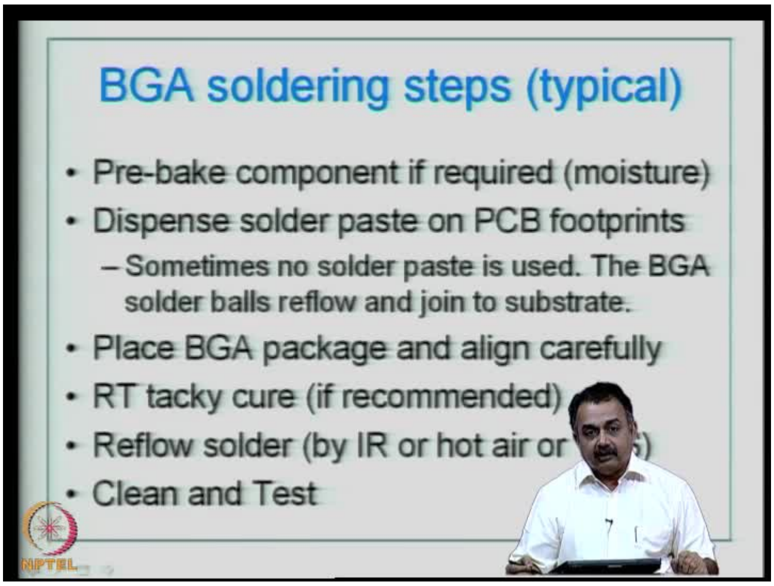
Now, look at the placement. Assume that this is placed at the center of the foot print initially. After reflow, if the solder paste volume is going to be larger than required, you can expect because of the large pad area that you have provided, the component can swim closer to the next component resulting in a short.

So, look at foot print size. Foot print area calculation is very important. Do not use foot prints that are available in the library for all types of devices; that is your chip capacitors. You can modify, edit the component foot print and reduce the size, if required, for a particular component of your choice.

Similarly, this distance that is the width is also very important. You do not want the component to move away from this pad and sometimes the situation is that the component can swim to one side resulting in a no contact on the other side. So, clearances between foot prints and area of the foot print - very important.

Ideal foot print will cool evenly when you do the reflow centering the component. Although we expect self-alignment due to surface tension of the solder alloy, yet this is an ideal case. In some cases, if the solder paste is too much or if the temperatures are uneven, then you can expect this kind of skewing effect that can take place or as I said it could move to one side. So these are situations that you really want to avoid during reflow soldering of chip capacitors or chip resistors. These are all very small points that you can implement at your design stage itself.

(Refer Slide Time: 26:20)



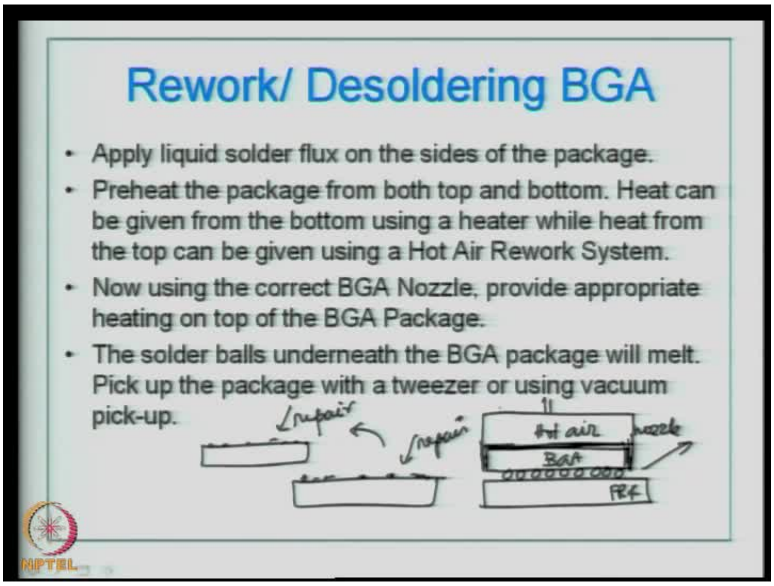
As a summary, we will look at writing down the soldering steps for a BGA. We have seen the various process mechanisms by which you can attach a BGA. I will just list the typical BGA soldering steps. First thing is prebake the component if required, remove the moisture the reason is you want to avoid pop corning effect.

Dispense the solder paste on the PCB foot prints. In some cases, no solder paste is used the BGA solder balls reflow and join to the substrate. In this case, you must have an equipment that can hold the BGA in place. If you do not have such an equipment, then it is obvious that you should use a small volume of solder paste which contains an adhesive and which can hold your BGA until it goes for the reflow oven. So, these are the two important steps.

Now you place the BGA package and align carefully using some kind of a fiducial on the board or if you have an equipment then your CAD data can help. At room temperature you can tacky cure if you are using the solder paste. This is recommended, because your glue will now hold to the BGA solder balls. Before this, your PCB substrate should be cleaned thoroughly that means your exposed pad, it could be copper, but typically it is plated with tin or tin lead or it could be nickel copper surface finish.

Now, you do the reflow soldering process by IR infrared or hot air or vapor phase soldering process. Finally, you clean and test. These are the important steps. There could be variations. These six points typically define how a BGA should be soldered. We are talking about a fresh BGA component being attached for the first time on the Printed Circuit Board.

(Refer Slide Time: 28:44)



Now, there is always a possibility that a BGA can be defective or if you look at the X-Ray after the attachment process of BGA is over, you could see that some of these solder joints have not been attached. There could be a dry solder, there could be flux too much solder paste on certain sides under the BGA which could result in shorting. When you have such a situation, you might have to rework. Pull out the BGA, de-solder the BGA and attach a new BGA after repairing the board as well as the BGA back side that is the solder ball side. That is known as repair and rework.

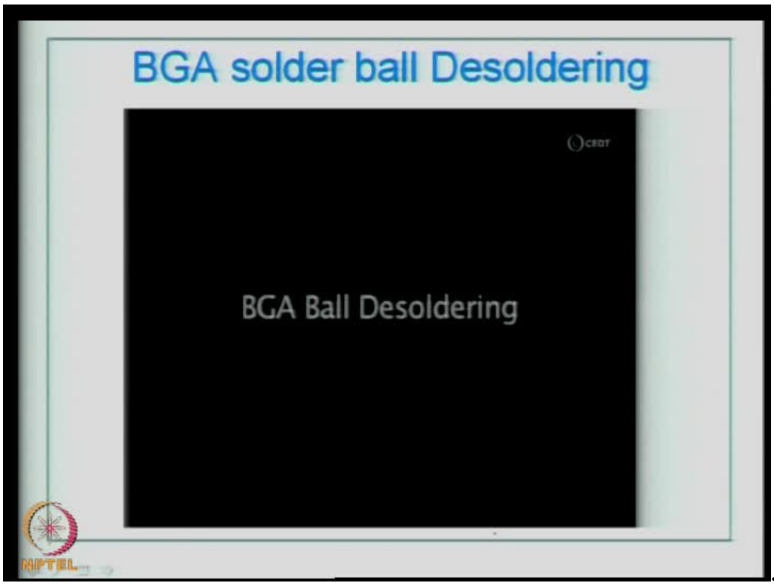
Let us see how this is done. The first thing is apply liquid solder flux on the sides of the package. Imagine a BGA flip chip on one side and the other side you have the solder balls which have been removed. Now, how do you remove the solder balls? You can apply hot air and use vacuum to remove the material and then clean up the area with isopropyl alcohol or acetone to keep it free from the residues from the previous grid that you had.

Now, to rework, apply liquid solder flux on the sides of the package, preheat the package from both the top and bottom. Heat can be given from the bottom using a heater like IR heater while heat from the top can be given using a hot air rework system if you have a prototyping system. Using the correct BGA nozzle, provide appropriate heating on top of the BGA package because we do not want to affect the other areas, the solder balls underneath the BGA package will melt. Now, pick the package with a tweezers or using vacuum pickup.

This applies to both. If there is a board and there is a BGA package. This is the BGA package. This is your FR4. This is to be repaired. What basically you can do is apply a hot air around the BGA package on top of the BGA package totally and then you remove it. When you remove it, what you get is on the PCB those pads will be having some debris of the solder ball and whereas, the BGA package if I show you the top side they will also be non-uniform because you are basically trying to remove the BGA from the board.

Now you have to repair the BGA. You will also have to repair the board bonding sides and then only you can go for a new re-balling process. This is a task that everybody who is working in hardware, who is working in board design and assembly need to be aware of today because we are using a lot of BGAs.

(Refer Slide Time: 32:30)



What I will now do is show you a video that shows about BGA ball de-soldering. This has been done at the CEDT packaging lab. I will explain to you the process steps as we go by. What you are seeing is a BGA package. Assume you want to because there are a few bad interconnects, you want to pull out the solder balls from those areas.

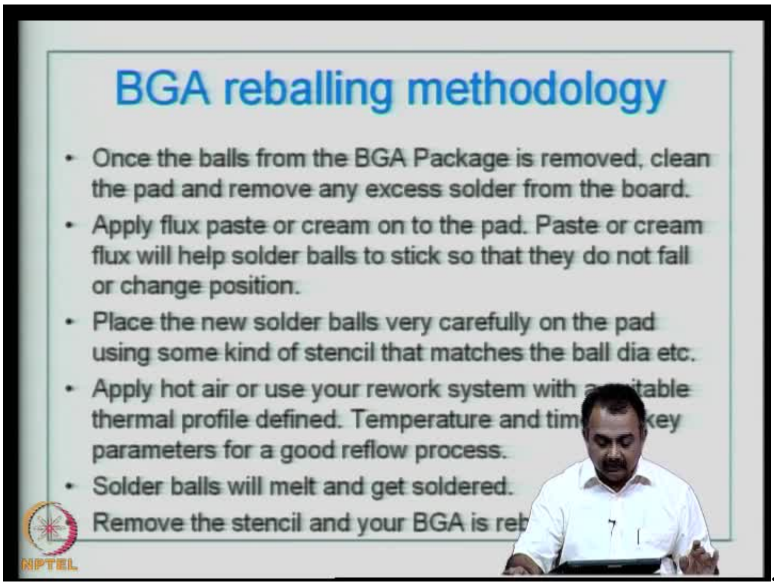
We will now show you how to remove the solder balls which are defective. In this particular video, irrespective of whether it is a defective ball or not, we are now going to remove all the solder balls assuming there are a lot of defects. What we are trying to do is the BGA package is held firmly on a frame. There is a hot air nozzle. You apply hot air at appropriate temperatures that is suitable for the FR4 substrate and to melt this solder alloy that you are using. So, you need a knowledge of that. Then, you have here the vacuum pick up of the molten solder. What you are seeing now, hot air is blown and simultaneously quickly we are removing the molten solder by a vacuum syringe technique. In this way, very carefully without affecting the board surface you are now removing the solder balls from entire area that you need to rework.

Now, remember in a BGA you can work on smaller areas. You need not exactly unless of course, all the BGA balls are damaged in that case you have to remove the entire solder balls. So, you can see here now these areas have been removed with solder ball and what you are seeing is the exposed pad on the Printed Circuit Board. Now, you can see entire BGA surface has been removed with the solder ball you can see here.

So, this is the best technique you have to do it manually, if you want a good performance done and if you want to make sure that your substrate is not affected. You can use the same chip after re-balling the package with new solder balls. Currently for this technique, there is no typical automation because you want to carefully look at every package about the de-soldering and the re-balling process.

Now let us look at how we do a BGA re-balling. Once the balls from the BGA package are removed, clean the pad as I said before. Clean the pad with appropriate solvent and remove excess solder from the board so your package is also cleaned and your board is also cleaned.

(Refer Slide Time: 35:50)



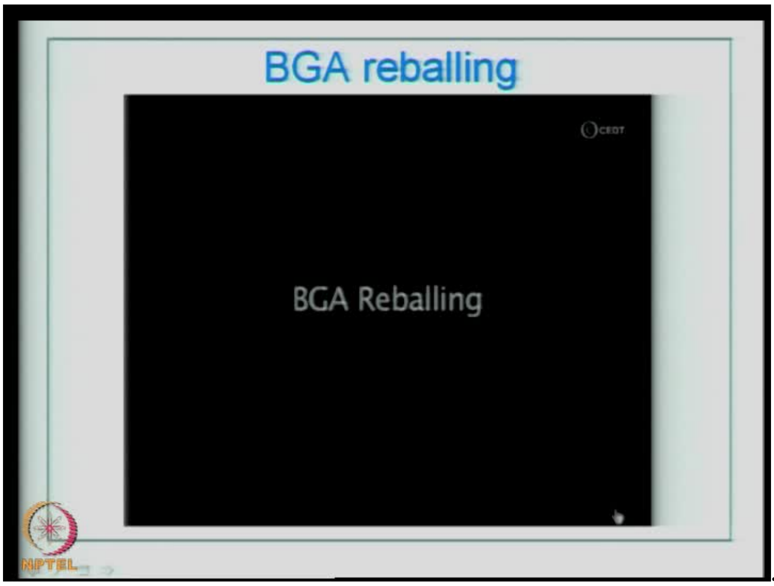
Now apply flux paste or cream on to the pad. Paste or cream flux will help solder balls to stick instead of using a thin flux liquid, so that they do not fall or change the position when the solder balls are attached.

Place the new solder balls very carefully on the pad using some kind of a stencil that matches with the ball diameter that you intended to have. That information is very crucial. You cannot use any ball size. You must have an idea what originally was the ball size because you are working on a definite pitch. Different ball dias will affect your pitch of the component.

Apply hot air or use your rework system if you have with a suitable thermal profile defined. Here again, you have to ramp up your heating very slowly. Remove the flux solvents and then soak it for some time and then again ramp it to a peak reflow temperature where the solder alloy will melt. There will be good fusion of the materials and then you slowly cool to room temperature.

Temperature and time are very important for a good reflow process. You have to work on these issues. Again, I repeat temperature, time, knowledge of the solder material, knowledge of the substrate, that is the glass transition of the substrate are all essential information. Solder balls will melt and get soldered. Remove the stencil and your BGA is re-balled now.

(Refer Slide Time: 37:35)



Let us look at how a re-balling is done. What are the essential starting materials for a BGA re-balling? You can see this is the cleaned BGA substrate which is de-soldered. You have a stencil here. You have the flux creme in syringe. You have a brush. You have the new solder balls and this is the equipment or the place holder where this template and the package will go and you will do a reflow when by capping this and heating it. So, these are the starting materials.

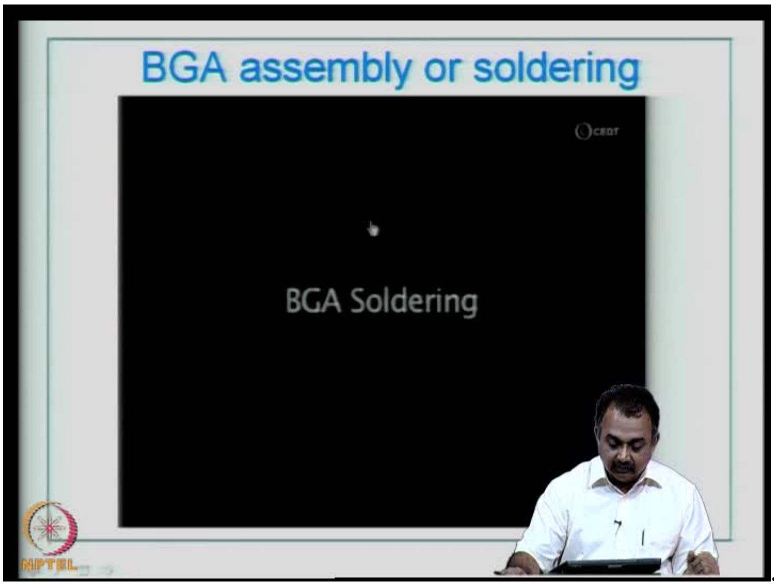
Now, let us see how this re-balling is done for that package which was de-soldered. First use a flux creme which is somewhat consistent, which is somewhat tacky. So, using a brush, apply creme on to the cleaned areas of the BGA package where you removed the old solder balls. This should be a very thin layer sufficient enough to act as a flux and also to act as an adhesive for the BGA. Now you are placing the BGA. You are now placing the template. You can see in this holder you are placing the BGA substrate as well as the stencil. The stencil contains the holes pertaining to the BGA metrics of the right dimensions. Once you have done this, you can see the exposed pad area.

The next job is to apply vacuum and keep it in place. This will help the BGA balls also to flow in to place. Now, we are seeing new solder balls of the same quality of the same diameter being poured on to the stencil area and you can see the entire metrics is now ready with new solder balls. You are now closing the lid. This is a stainless steel material. It can be stand higher temperatures. It can protect your BGA substrate.

Now, use any equipment that you are working with, which can set a thermal profile from room temperature to the peak reflow temperature, let us say 220 or 230 degree centigrade depending upon the alloy. So, you have to gradually heat it up. You cannot set it at higher heating grades. You can see a thermal profile here, very slow heating and then reaching the peak reflow temperature. Once it is done and it is slowly cooled you can see slow cooling taking place to room temperature.

Now, you can lift the cover or the lid and you can see the BGA has been re-flown. Now, you can remove this stencil that was intended. This stencil is not affected by these temperatures. It is basically a pure dielectric epoxy glass. Now, you can see a wonderfully done BGA re-balling. The quality is excellent. There are no defective solder balls. We have used the right temperature and timing to re-ball this package. This package is not wasted. You can reuse the package.

(Refer Slide Time: 41:33)



This is a very good demonstration of how a BGA can be re-balled. This BGA is ready for soldering. We will now see how this BGA is soldered on to the board. Package to board interconnection. What you are seeing here is the board. We can see provision for other components. This is the board. This BGA package which you have repaired is now vacuum picked and is now being tried to align on to the foot print on the board. Make sure you have some mechanism, you have some fiducial on the board to align this BGA on to the board. After this, you can set the reflow thermal profile for this particular board.

You can see there is a nozzle that is being specially utilized for this BGA package which means hot air is specifically targeted around this particular BGA package. You can get these different nozzles for different sizes.

Now, the thermal profile heating has started. There is a preheating first, then there is a soak zone, where the entire system will be soaking so that all the components of the solder paste if you are using a solder paste is removed. Then finally, it reaches a peak reflow. Situations are different when use a solder paste or without a solder paste.

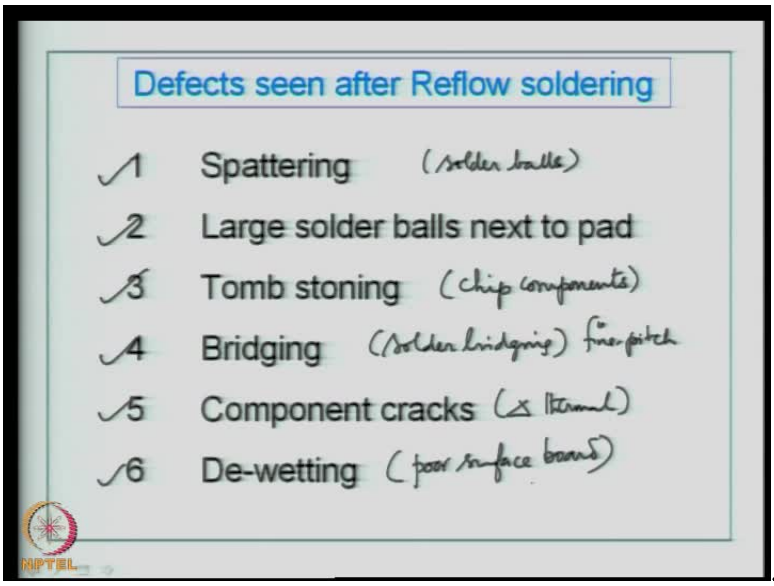
So, in any case, you can see this particular equipment has got IR heating from the bottom. How does it help? This helps in keeping the board active so that it easens up this reflow soldering process because hot air is applied only on the top. Board sensitization to this process is done by IR. So, it is a combination of IR on one side and hot air on top of the package. Use a thermal profile that suits your BGA package and you can see here very nice solder ball connections done on to the board.

Now, you can remove the nozzle. Your BGA connection to the board is ready. Everything works with slow cooling. You cannot do a sudden cooling, because the solder joint reliability also depends on the right temperature of reflow as well as slow cooling. So, the solder joint failure is dependent on these qualities of reflow process.

So, we have seen these three videos, where we saw de-soldering then re-balling and then soldering process. Whatever be it, even if you do a good reflow process or a wave soldering process or a vapor phase process, you might end up seeing some defects. Defects cannot be ignored and defects are possible even of the best equipment and process. We have to watch out for defects, that is why quality control in assembly becomes very important.

Some of the defects that we can see I am going to talk about reflow soldering especially is Spattering. Spattering means spreading of the solder balls especially the solder balls on the board after the reflow process is over. This can be due to various reasons.

(Refer Slide Time: 45:26)



We will see that. You can see the solder balls, micro solder balls spread all over the board in some cases due to wrong temperatures, due to poor quality of the solder paste, too much of solvent in the solder paste, and so on and also poor binder in the solder paste.

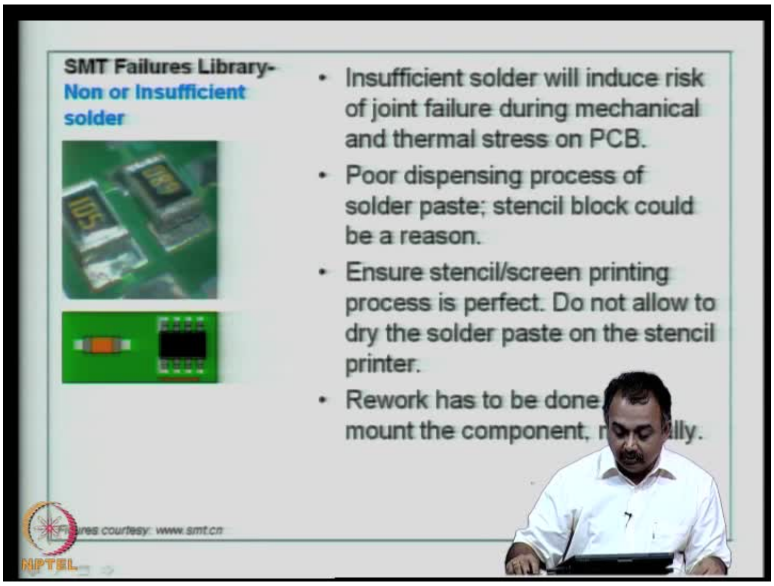
The next one is large solder balls next to pad. This is also known as solder beading. There are certain issues. Why these are formed? We have a very important effect usually seen in typically chip components called Tomb stoning. In chip resistors, chip capacitors you will see Tomb stoning.

Bridging is a very common defect. Solder bridging that especially happens with low pitch or fine pitch components. Therefore, the quantity of solder paste applied for solder bridge for attaching a fine pitch component is a key parameter.

Component cracks due to thermal shock is an issue. Then De-wetting, basically because of poor surface preparation of the board. So, these are the issues or defects that one can see. There could be defects that you can actually notice at the solder joints. It could be a dry joint, it could be a voiding, it could be a trapped bubble or an air or a moisture leading to a void. So, those can lead to joint failures over time.

So, we have to carefully look for high quality processes. Defects, if any, have to be a rectified and we have to zero in or troubleshoot where exactly in the process line, the reason for the defects, the process which has caused the defect.

(Refer Slide Time: 47:52)



We will now look at failures library. I will show a series of slides which could be typical cases of defects that can be seen after reflow soldering process. The first one is known as non or insufficient solder. Obviously when you say no solder or insufficient solder that means there is no proper electrical connection. As you can see from this figure, in this case, there is insufficient solder may be on one side of the terminal, but this is a very good joint, but the adjacent one has got problems.

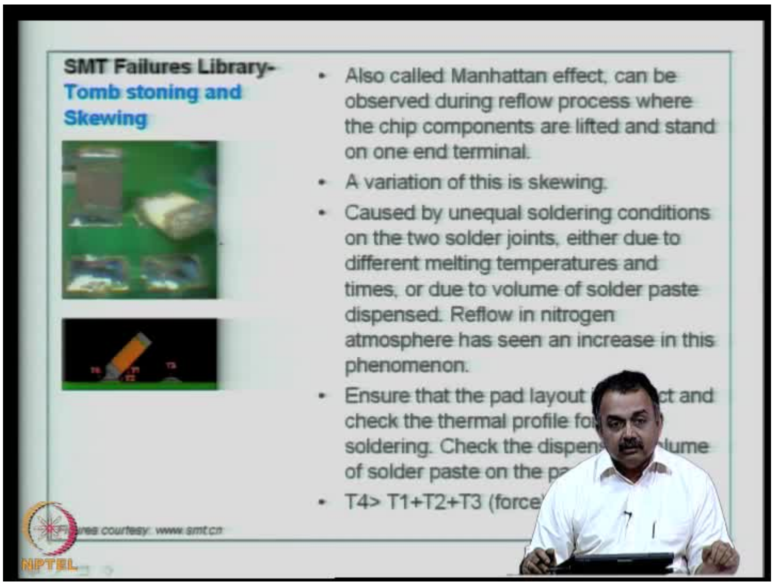
What are the issues with this? Insufficient solder will induce risk of joint failure, because of low volume of solder during mechanical and thermal stress loads on the PCB. If you notice this, you can better remove that chip component and do a manual soldering process.

What is the reason for this poor dispensing process of the solder paste? That means in that particular area there is a stencil block. This stencil that you have used is blocked. The area needs to be cleaned. If you repeat this and you create the same defect. If you have seen this for example, in 50 boards then definitely the stencil need to be re-cleaned and reworked so that you can start afresh the printing process.

So, that is why when you dispense the solder paste by syringe or stencil before the components are placed, you need to have a look at the printing quality because that will give an indication of how much volume the solder paste has been dispensed.

Ensure that the stencil printing process is perfect. Do not allow the solder paste to dry on this stencil printer for a long time. Use small volumes. Finish off the printing and then mix a new paste or material. If there is a drying up on the screen or on the PCB before reflow, then you have problems. In the manufacturing, I think they take care of these kind of issues. Rework has to be done to rebound the component... But you have to do manually, because there may be just one or two such kind of defects on a PCB normally.

(Refer Slide Time: 50:50)



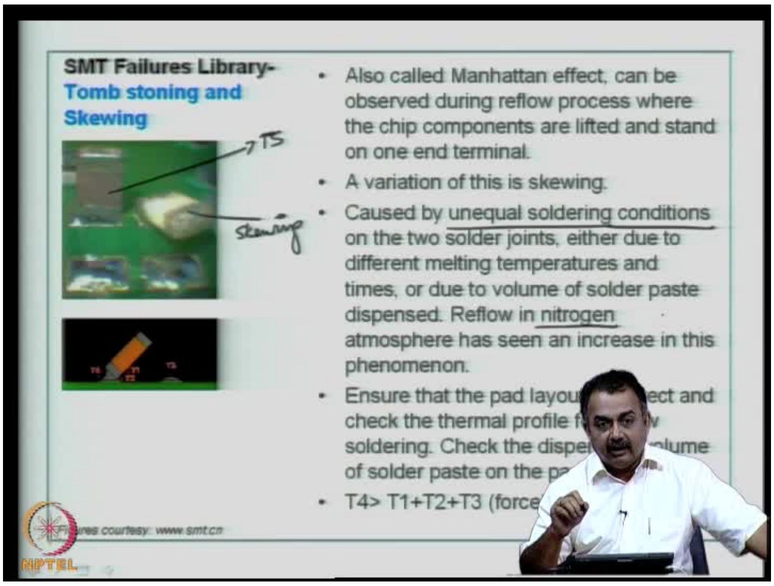
The next failure that we are going to see is Tomb stoning and skewing. What you see in this particular case is tomb stoning and this is skewing. This is also called Manhattan effect. It can be observed during the reflow process, where the chip components are lifted and stand on one end terminal.

There are couple of video clips available on the web which you can try to find out and see how exactly this happens. A very interesting situation and variation of this Tomb stoning is known as skewing. Here the component does not lift completely to one side whereas, here it stands up like a tomb stone. Therefore, it is known as the Tomb stoning effect. The other one basically lifts and slides to one side. Both are defects and you have to repair it.

How is it caused? It is caused by unequal soldering conditions on the two solder joints. Although the joints are too close to each other, the condition here could be different compared to the condition on the other joint.

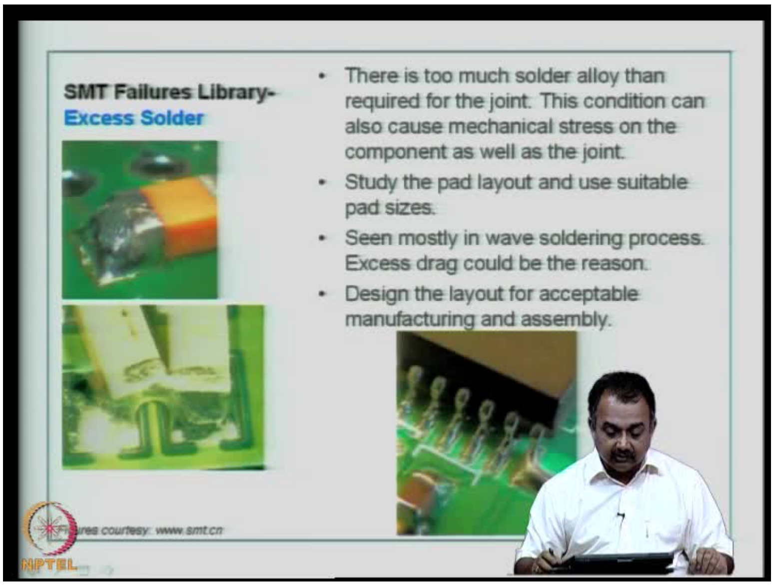
The solder paste, when it is dispensed on both sides, could be unequal. The temperature that each of the terminals attain during the reflow process could be different; the atmosphere during the reflow process, whether it is nitrogen or air, could be a reason for these kind of defects.

(Refer Slide Time: 52:47)



Ensure that the pad layout is correct, so that there is sufficient drag of the solder. Check the thermal profile heating. Make sure this spacing between components are okay so that they get equal thermal energy during heating and check the dispensing volume of the solder paste, when you print. Essentially these terms, Tomb stoning and skewing happens because the force at this terminal is larger than the force exhibited by the component on either of the terminals and at the center. T 4 here is greater than T 1 plus T 2 plus T 3, the individual forces at different sides of the assembly of a particular component. So, these are situations which can be rectified to eliminate Skewing and Tomb stoning.

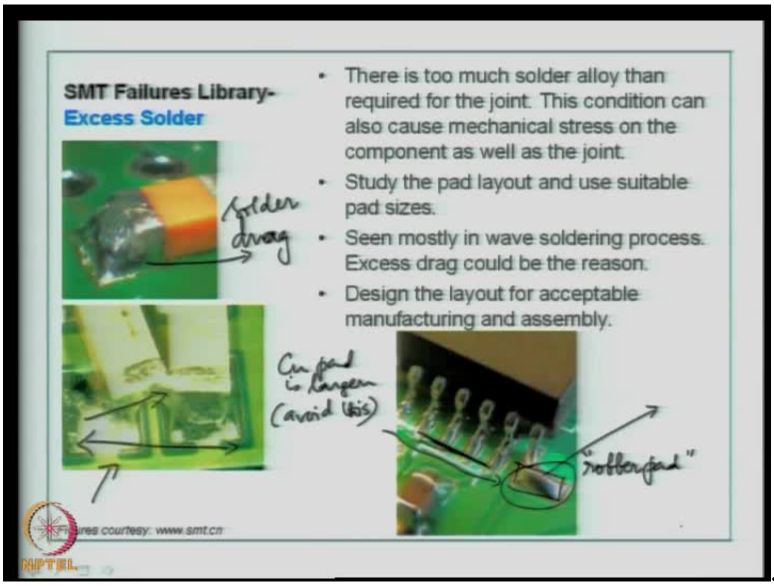
(Refer Slide Time: 53:50)



The third defect is excess solder, which means there is too much solder alloy, as you can see, than required for a particular joint. Because the size of the package is very small, you do not require too much solder. At the same time, you cannot have deficient solder too. This condition can be detrimental to the board because it causes mechanical stress on the component. Because there is too much solder, it can heat up the component, it can cross stress on the component material as well.

Study the pad layout and use suitable pad sizes. It is seen mostly in wave soldering processes because the drag is too much on a particular solder. Drag is excessive on one side means your copper pad area is larger; so, you have to avoid this. You can see in this picture, how the solder drag or too much solder makes the component shift from its base. We have seen the design issue regarding this and it ends up in a short. So, you have to repair both these components manually.

(Refer Slide Time: 55:20)



Now, excess drag is an important issue during wave soldering, Design the layout for acceptable manufacturing and assembly. For example, you can see here, this is a QFP assembly which has undergone wave soldering, but the designer has taken care of by having what is known as a robber pad. What it does? Basically during the wave soldering process, there is a drag of the wave, but at the end terminal there is an extra pad provided which accumulates all the solder material and it avoids solder bridging in any of these areas. So, this is the design issue that can be effectively utilized.

We will see such defects in the next class. We have seen three defects today. One is no solder, one is excess solder, and the other is Tomb stoning and skewing. I will review this chapter in the next class again and then we will look at more defects. We will also look at lead free alloys; and towards the end in this module, we look at green issues for packaging.