An Introduction to Electronics Systems Packaging

Prof. G. V. Mahesh

Department of Electronics Systems Engineering

Indian Institute of Science, Bangalore

Module No. # 07

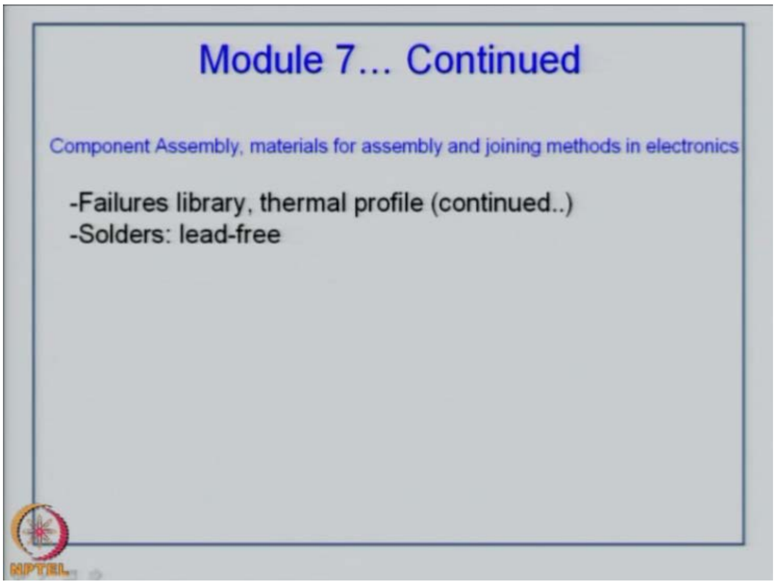
Lecture No. # 36

SMT failure library and Tin Whiskers

.

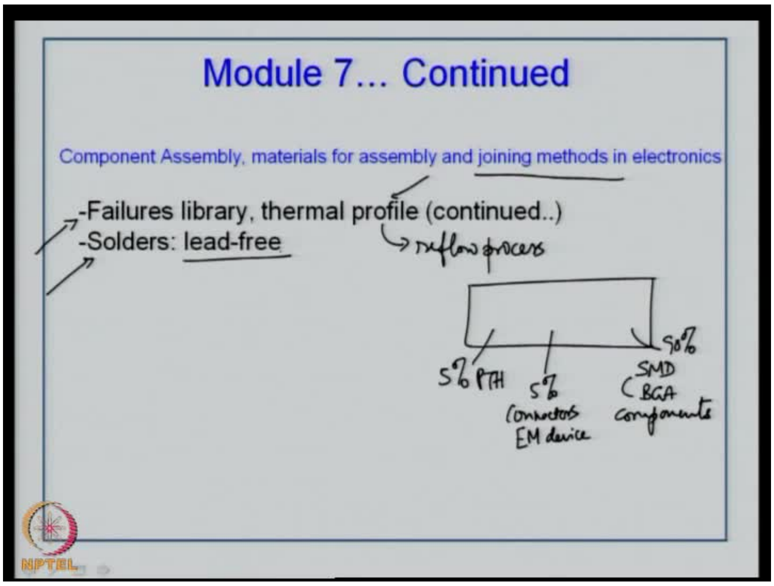
We will continue with the module 7. As you know, we have been covering various chapters in module 7. As a recap, I would like to inform you that we are looking at component assembly, materials for assembly. When we say assembly, we have looked at plated through-hole component assembly, surface mount device assembly and including advanced packages like BGA or CSP, QFN and alike. Now, flip chip if you consider as a chip connection choice not as a package, we have seen earlier how such interconnects can be generated or how you can connect a flip chip on a Printed Circuit Board.

(Refer Slide time: 00:16)



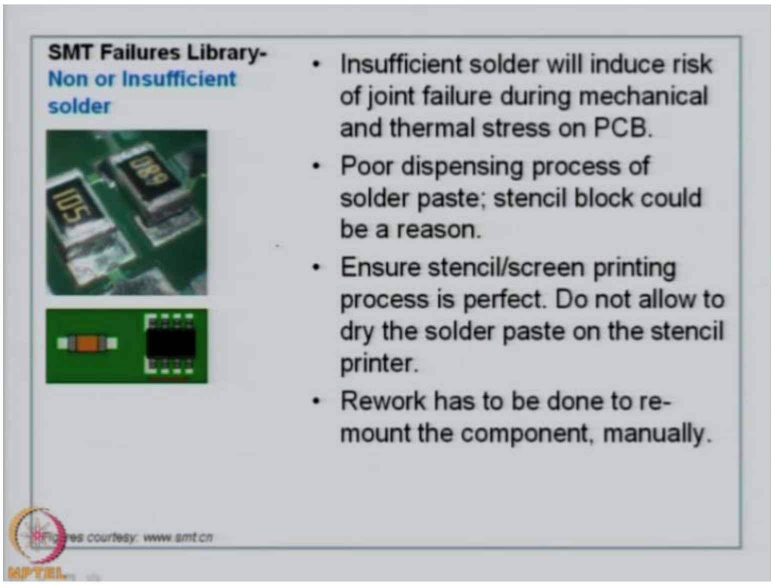
Now, flip chip normally comes into a BGA package. Please remember we have mentioned this quite a few times that if you are using flip chip, it is not a package. It is a chip connection choice. We have seen BGA; how they are assembled on an organic substrate that is the Printed Circuit Board. We have also seen in the few lectures before, the joining methods in electronics. That means, we have seen how to establish a reliable connection in electronics for plated through-hole assembly, surface mount devices and BGA. In fact, we have also seen various video highlights covering these aspects.

(Refer Slide Time: 01:46)



Today, we will shift our focus to some very important aspects governing, quality assessment of a finished or assembled Printed Wiring Board. The areas that I will cover today will be Failures library and also looking at thermal profile of an assembly process. Typically today, we are using reflow soldering for majority of the surface mount devices. We will focus on thermal profile of let us say, a reflow process; because as I repeat, if you look at a Printed Circuit Board for a mobile application, you will see almost 90 percent of the board are surface mount devices today and it includes BGA components. There may be about 5 percent through-hole components because of non-availability in the SMD sector. You can also have about 5 percent through-hole components in the form of connectors or some kind of an electromechanical device and so on. This is the very general trend today and we look at it from this perspective. We will also look at the current situation of using soldering material for joining components to the interconnect substrate. We will focus on lead-free components because today as you know, there is a huge legislation coming for handling or choosing materials especially in the form of eliminating lead. So, this will be the basis for our lecture today. We have in the previous class, covered a couple of these. We will go through them once again.

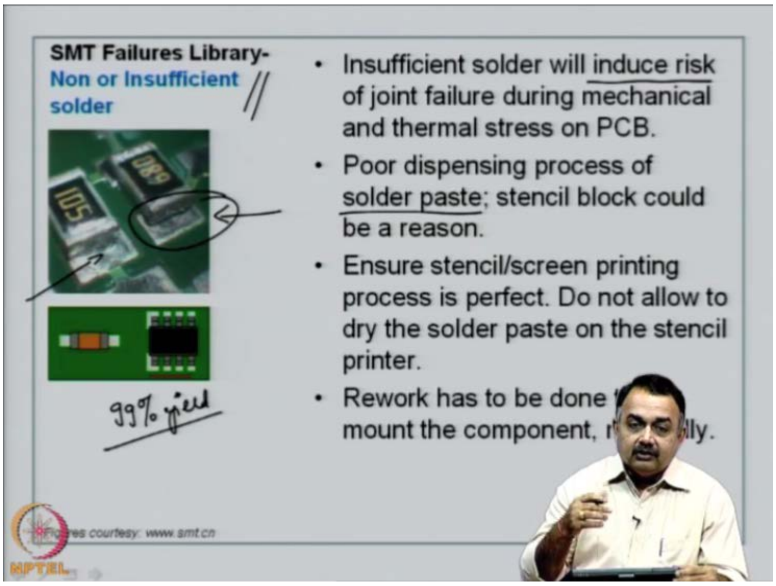
(Refer Slide Time: 04:21)



The Failures library that I am going to list today will cover some of the very common defects that can be observed, when you do a surface mount assembly practice. In the industry, these have to be avoided because you are looking at very high percentage yield. Typically, you want 99 percent yield and the failures can come from various sources. It could be from the component, which means selection of components. It could be from the Printed Circuit Board that has been realized for this particular process. It could be from the process parameters that you have used for assembling the components onto the Printed Circuit Board. So, we will go through some of these very important aspects. Very common defects have been presented here; so that when you try to assemble, you might encounter with these kinds of defects or difficulties. So, you need to be aware of some of the problems.

The first thing which I want to mention is insufficient solder or no solder. As you can see from this picture here, there is a fairly very good interconnect over here; joining method has been well established. But if you look at this particular joint of a chip component, you can see, there is obviously no solder or in some cases you would see insufficient solder. What does this mean? Insufficient solder will induce risk of joint failure. We have seen that an electrical spike could result in a mechanical failure. That means the component could break or it could lead to a solder joint fatigue or solder joint failure. Similarly, thermal issues, heating up of a chip component or an active device could result in a solder joint failure. This kind of failure, what we are seeing here in this slide is a result of a poor process control. You can see during the process, a particular component or a few components, if you carefully analyze, you would see that the solder material here is comparatively less, compared to the other parts of the board. What does this mean? This will induce risk of joint failure during mechanical and thermal stress on the PCB. When a PCB is powered up, the entire board is powered up and there will be thermal cycling. Thermal stress will be built up on the devices and your solder joint typically will have to bear the thermal shock. But because there is insufficient solder, this could result in an open and eventually a failure. How do you control this?

(Refer Slide Time: 08:32)

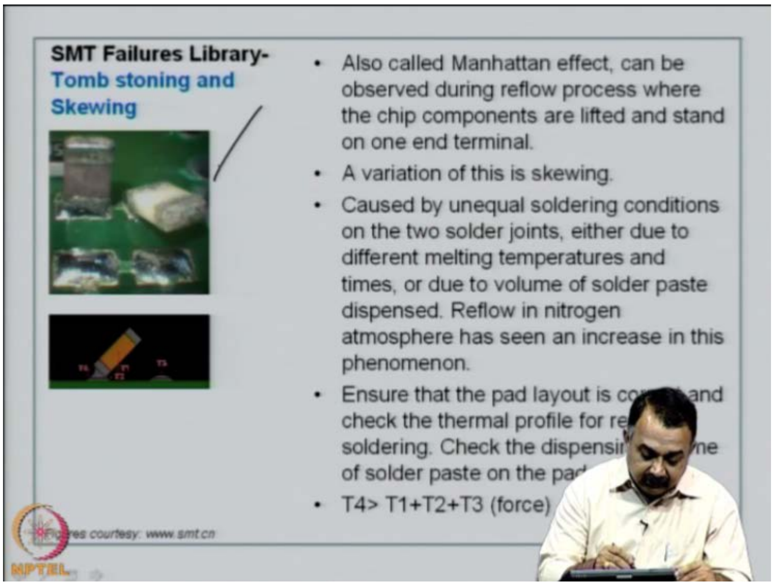


The important area where you need to control is the dispensing process of the solder paste. Remember, you are dispensing solder paste using stencil or screen printing, better done by stencil printing using stainless steel stencils. The stencil could be blocked for some reason; because either you have done too much of printing on that particular stencil or the solder paste could have dried and you have not really cared to clean the openings on the stencil so that the solder paste flow onto the PCB is continuous and consistent. Therefore, stencil block could be one major reason that you should think about when you look at this kind of failure.

So in such cases, you ensure that the stencil or the screen printing process is perfect. How do you do that? You clean the stencil with appropriate cleaning material. Do not allow to dry the solder paste on the stencil printer. So, the technician by experience would not do that once the process is over and if there is a gap between this printing batch and the next, obviously you have to keep the stencil and the work table of the stencil printer fairly clean enough so that the material does not dry.

After that, the remedy is in case you have encountered such a problem you have to do a rework and you have to remove the component and then remount the component manually. Obviously, in a board of 100 chip components and if about 5 or 6 are to be replaced then you have to go in for a manual rework that will be the best process and the best option that you have.

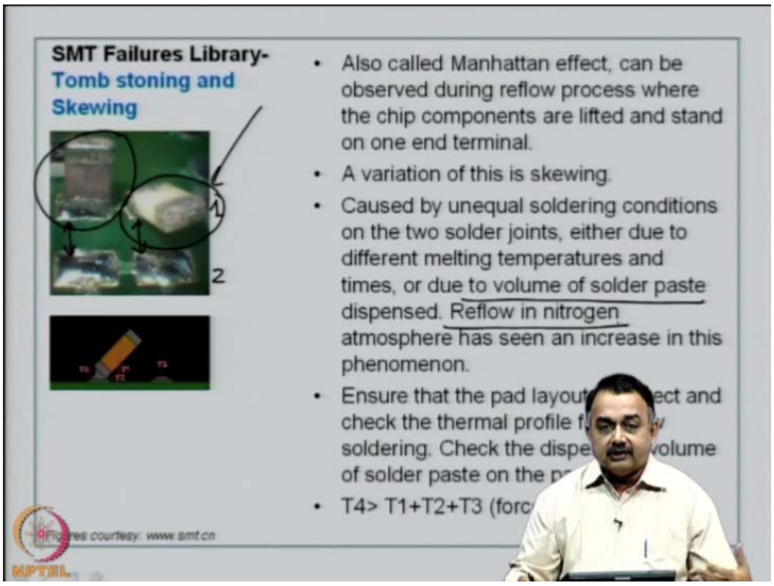
(Refer Slide Time: 10:21).



The next failure that you are going to see is Tomb stoning and or Skewing. Typically you would observe one of these very rarely. You would observe all of these together. It is also called the Manhattan effect. Look at this picture here. You can see that there is a chip capacitor and these are the two pads, copper pads on which the solder paste has been dispensed and your component has been placed prior to reflow soldering process. In this process, you can see that one component has lifted from the pad of the Printed Circuit Board and then it has skewed a bit. In the other case, you can see that it has lifted fully and it is standing as a tomb stone; hence the name Tomb stoning effect or Skewing effect.

This can be observed during the reflow process where the chip components are lifted and stand on one end of the terminal and a variation of this as I told you is the skewing that you see. How is this caused? It is caused by unequal soldering conditions on the two solder joints that the chip component is supposed to be placed on and then soldered perfectly. This has happened either due to different melting temperatures on let us say, pad 1 and pad 2 and also different times that you have given for the reflow process or the time taken for the solder material that you have dispensed on the board to melt. This time has been unequal on the different pads. This could also be due to different volume of solder paste that you have dispensed. If you recall, solder paste is dispensed either by syringe dispensing.

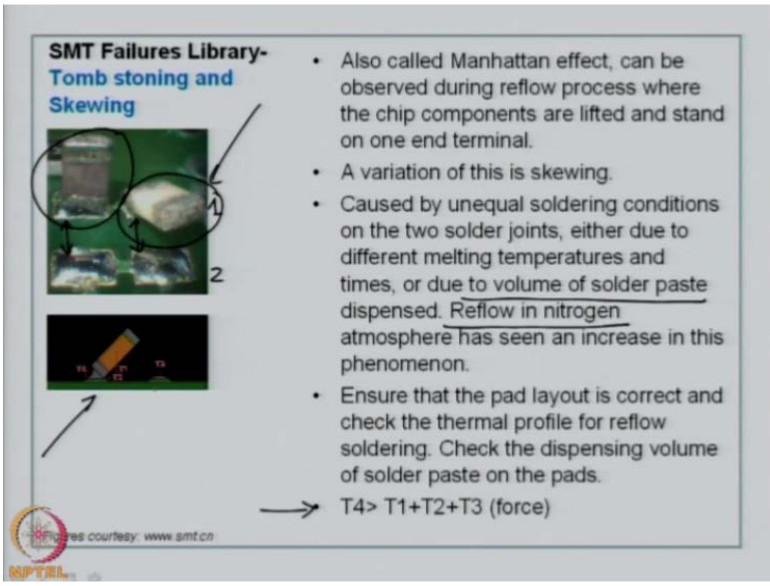
(Refer Slide Time: 13:27).



For small prototyping, you can do manually and that is where the problem comes. If you do manual solder paste dispense using a syringe, you tend to have unequal volumes on the pad. Therefore, this kind of problems could typically be when you do manual syringe dispensing of the solder past. Unequal volumes can come from stencil printing also. We have seen the previous slide that could be corrected by keeping the stencil clean and keeping the pores or the openings on the stencil fairly clean enough after let us say, every 500 prints or 1000 prints to make sure that the correct volume is dispensed onto the board.

In some cases, reflow in nitrogen atmosphere has seen to be the main reason for this phenomenon to occur. Why nitrogen atmosphere? Normally reflow soldering is done in air that means your reflow oven has air, convection based and there will be air circulation and typically the solder paste gets activated well in air. There is certain solder paste material which is recommended to be done in nitrogen; basically because of the composition of the solvent and other components of the solder paste. But then the thermal profile is slightly different for a reflow process in nitrogen atmosphere and that which is done in air. Air acts as a catalyst and you could expect much better activation of the solder paste when you do in air rather than in nitrogen. Therefore, this point of getting different melting temperatures and time of melting could be due to poor inactivation in the zone which is basically nitrogen. Therefore, this could be one of the reasons for looking at these kinds of defects like Tomb stoning and Skewing.

(Refer Slide Time: 14:58).

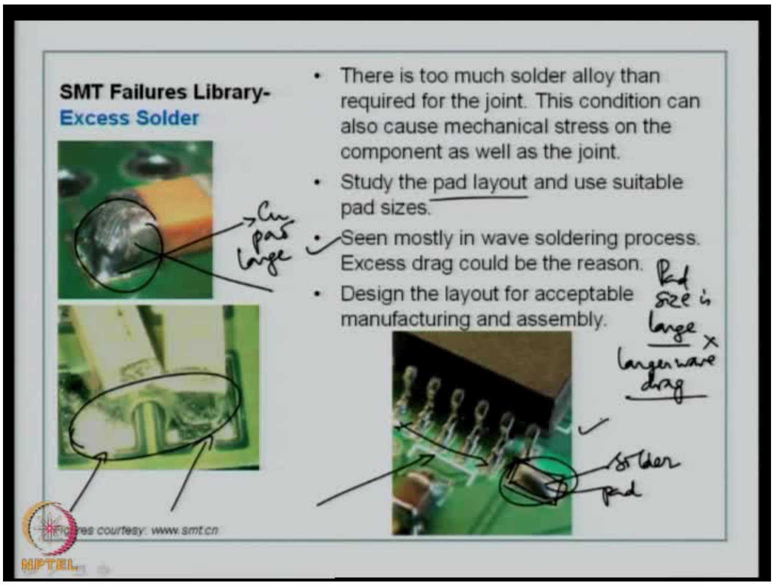


Typically what you see here in the point that I have mentioned at the bottom and if you relate it to the figure here, the force T4 which actually lifts the component from the base of the pad is very much larger compared to force exhibited by the solder paste at this end than the force exhibited by this terminal's stub on the other pad and then the weight of the component. Therefore, what has happened basically here is this solder paste has got activated much faster. The melting temperatures have been attained in a much lesser time compared to this pad and the solder paste. Therefore, this solder paste here by surface tension has pulled the component towards its side and it is not at all attached to this pad. Typically, you will see a very small variation in the melting temperatures and if this entire solder paste here at this point is not molten, then the surface tension is not available or not exhibited by the solder paste to hold the component terminal towards its end. That is one of the reasons why you see tomb stoning taking place. So, having seen this problem how are you going to correct it?

Ensure that the pad layout is correct; especially there are pad designs available for wave soldering and reflow soldering. So, you have to make sure that you use the right pad size for reflow soldering process for the components that you have chosen. Now, it is not necessary to have a very large area for chip components that will lead to a larger solder paste being dispensed and you have thermal profile to be adjusted to suit that kind of a situation. Ensure that you have minimum pad size for the chip component and to make sure that the minimum solder paste is dispensed.

Now, once that is done from the design stage, the next thing that you have to check is the thermal profile of the reflow soldering process and make sure that the melting temperatures are attained in a fairly quick time without too much of exposure to the component and then you also have to look at the component density on the PCB as a whole. So, thermal profiling becomes very important, to avoid these kinds of defects. Also, you need to understand the components of the solder paste material.

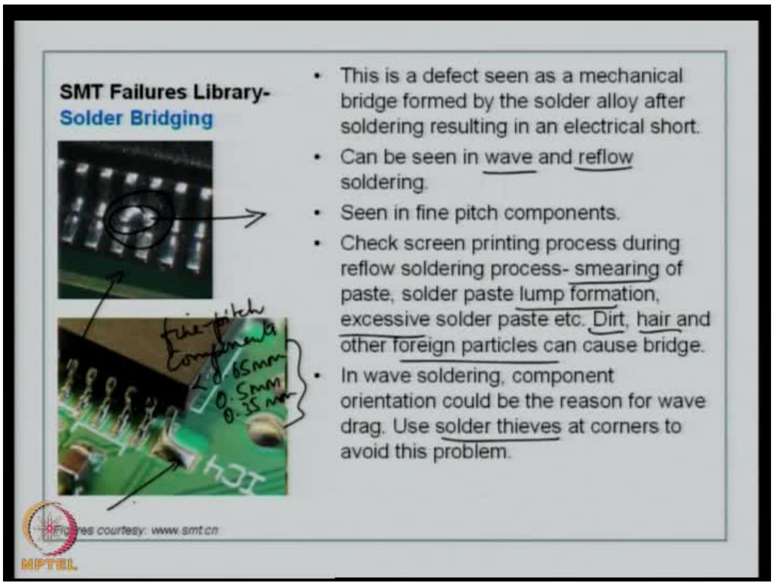
(Refer Slide Time: 17:59).



The third one is excess solder. Here you can see in the picture, different defects. You can see here too much of solder material; that is too much of solder there on the component. There is too much solder alloy than required for the joint. This condition can also cause a mechanical stress on the component as well as the joint. Too much of solder is also a problem. So, study the pad layout; use the pad size correctly and use suitable volume of the solder paste material. If you are using wave soldering, this could be a common problem. What you are seeing is typically if your pad size is very large then there will be a larger wave drag from the wave soldering process. So, you have to avoid that. So, from your design you must understand whether your board is going for wave soldering or reflow soldering and accordingly do the layout. You can see here typically what would have happened here is the copper pad area is very large and it has taken too much of solder during the wave soldering process and that is why you see excess solder.

Now excess solder can also create problems in component swim. They are not attached in their proper coordinates. You can see here in this particular process or figure, you can see a large volume material that is the solder material has been dragged and luckily here there is no bridging between the components. The excess solder has been taken by an additional pad that you have designed in your layout. So, this is a very apt process. A correct process, a correct design that is meant for taking care of such errors or defects during assembly.

(Refer Slide Time: 20:13)



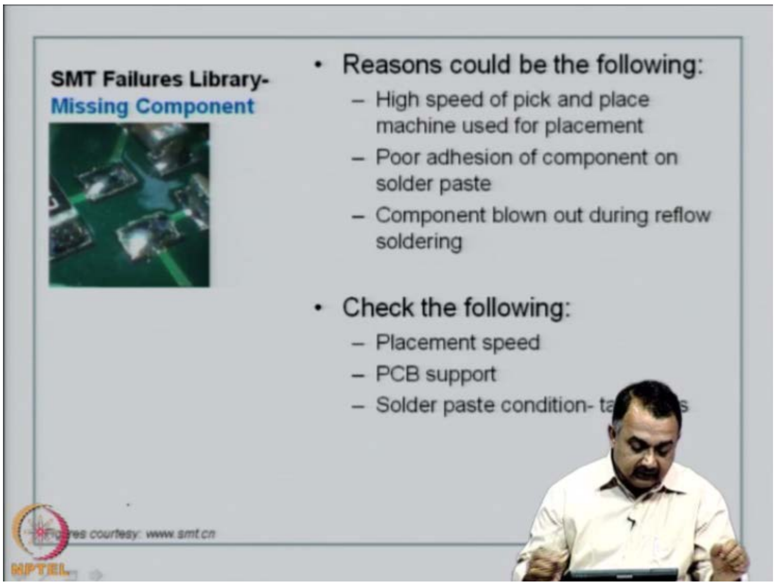
The next failure is solder bridging. You can see in this figure here, there is solder bridging that means adjacent terminals have too much solder and they short each other because of solder present in between pins or terminals. This could be seen in BGA, it could be seen in QFP components and if components are placed close to each other without proper design considerations, then you can see solder bridging. This is a defect seen as a mechanical bridge formed by the solder alloy whatever solder material you are using after soldering and resulting in an electrical short.

It can be seen in wave soldering. It can also be seen in reflow soldering process. So, you have to observe this defect very carefully especially in fine pitch components. If you are using QFPs and if the pitch is less than 0.65 mm and very closely watching 0.5, 0.35 mm, you will see there are chances for solder bridging. Therefore, your thermal profiling and again the quantum of solder paste that you are going to dispense have to be well measured. So, design considerations are clearly very important in manufacturing.

How do you rectify this? Check screen printing process during reflow soldering process that is if the solder paste is smearing, if there is a lump formation, excessive solder paste if there are contaminants like dirt, hair or other foreign particles, if they are attached onto the PCB and if you have not taken care of during placement of the component then there is very good chance that they will attach themselves to the solder paste material and that could be one of the reasons for the formation of the solder bridge. So, make sure that the work is done in a clean atmosphere and you make sure that the technicians and personnel do not work with materials that are contaminated especially the cleaning solutions or the cleaning materials that are being used, the brushes and so on which are normally used to clean the boards and then the solder paste itself should be of very good quality.

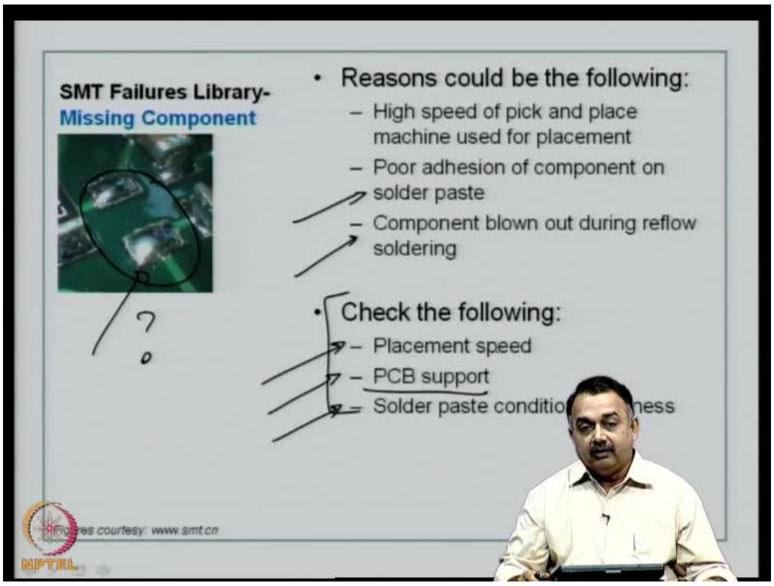
In wave soldering how does this happen? It could be due to poor orientation of the component and that could be avoided by using solder thieves which we have seen in the previous slide. This is typically called a solder thieve, which pulls out the excess solder from the last terminal of a QFP. So, this is about solder bridging. As you can see this is the figure I was referring to.

(Refer Slide Time: 23:40)



Next defect is, missing component. As you can see here, there is supposed to be a component here but that is missing. The reasons could be very simple. When you are doing an automated pick and place, this normally does not occur, when you do small volume prototyping manual placement but normally encountered with high speed pick and place machine that is used for placement of a large number of components and those large number of components could be very light chip components. When you place a component either by the automated machine or manually, you are placing it on a solder paste. Now, solder paste is supposed to have some kind of tackiness to hold the component. So in this case, typically if you troubleshoot, you could probably think that this solder paste does not contain enough adhesive to hold the component. Remember in a pick and place machine, the speeds are very high and normally the pick and place nozzle or the head does not position your component on the coordinates. It actually comes to the coordinate and then actually drops the component and the component by its weight has to sit on the solder paste. The solder paste with its adhesive component should hold the package until it goes for reflow soldering process. In between there is a tacky cure process that is you can do it at room temperature or at very low temperatures in the oven. You can do a tacky cure. But then before it goes for a reflow processing, the components should not move. So, misalignment of component could also happen when you consider these kinds of problems with solder paste material.

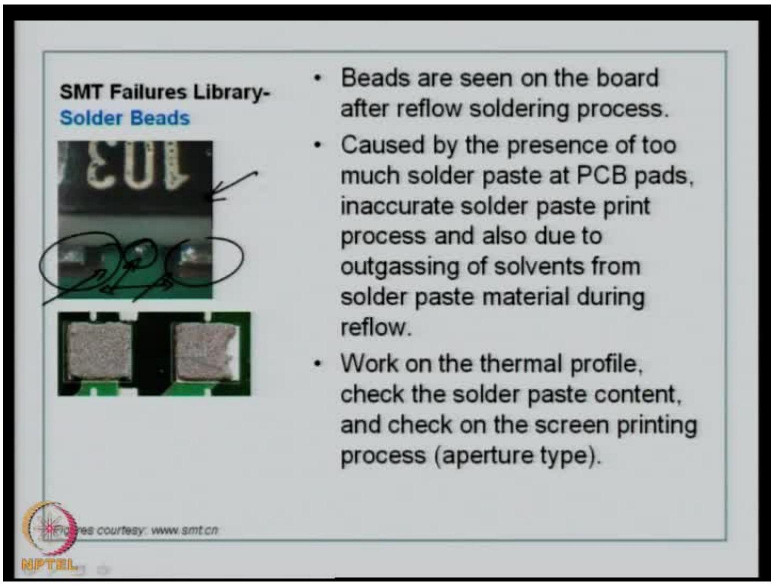
(Refer Slide Time: 26:28)



The other thing is, when you work with automated machines, the component could have been blown out during the reflow soldering. Again in a reflow, if there is convection air, the component is probably displaced because of the air inside the reflow machine. So, again it emphasize the fact that reflow soldering process when you use a solder paste, the solder paste should have good tackiness.

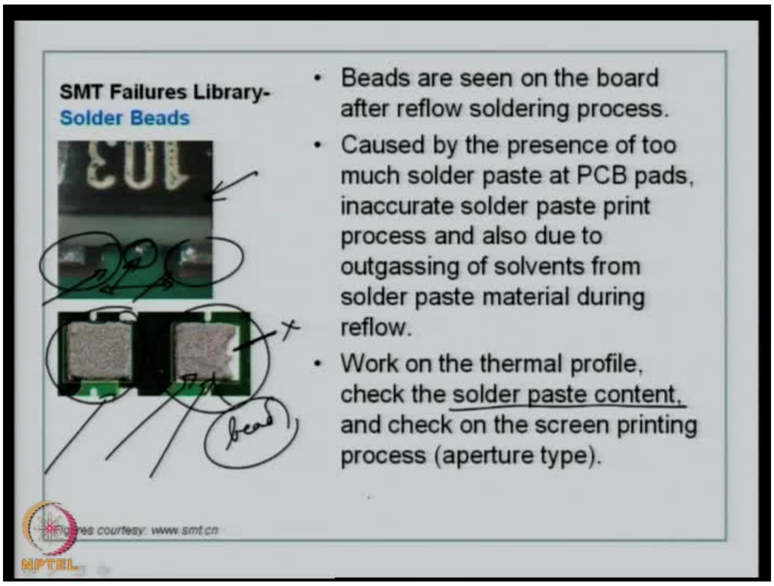
So, how do you take care of these kinds of defects that should not be constantly seen in your boards? For automated workplaces look at the placement speed, look at the PCB support and look at the solder paste condition. All these things I have just now emphasized, especially the solder paste, the PCB support comes from the basic equipment loading and the placement speed, you have to take care of in the pick and place machine. All of these can be controlled well; it could be monitored well by an experienced technician. If you are looking at good ethics of working and good environment for working, these could be avoided.

(Refer Slide Time: 27:02)



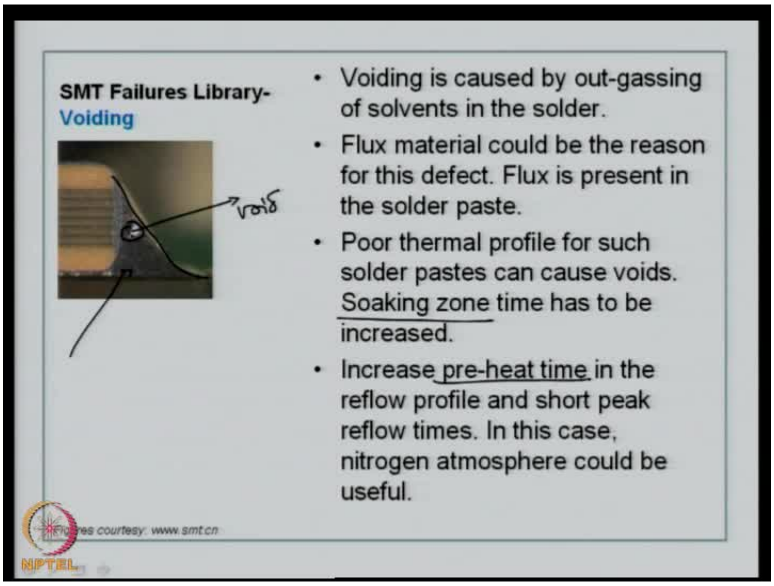
The other library illustration that I would like to show is a defect known as solder beads. Here you can see, there is a package and you can see the footprint and in between you can see there is a solder bead. Now, could this be a defect? Yes, if this pitch is very small and if there is a bead that you have noticed or not noticed and you think it is not going to harm the package or the board, then you are wrong because this could be a very important seat point for corrosion. It could damage the package. It could result in short over time because of some kind of a whisker formation or it could contaminate the surface of the board and form an electrochemical connection bridge between adjacent pads or tracks and so on. There is a solder mask on the board, but a lump of a hot solder bead which is typically tin lead or even if we take lead-free materials they could be harmful in terms of decomposition and if the board is subject to certain kind of a thermal stress or thermal cycling they get decomposed and the degassing or the solvents or the other chemicals from the solder alloy could damage the surface of the board eventually leading into some kind of a short between very close tracks on pads.

(Refer Slide Time: 30:21).



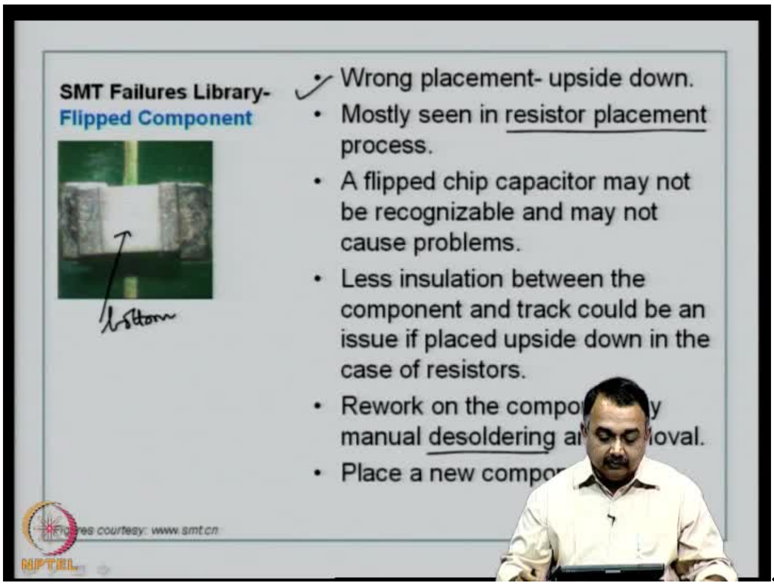
Beads are seen on the board after reflow process. How has this happened? The solder paste has the solvents in the solder paste. They have boiled or evaporated because you have set a thermal profile which is absolutely wrong. You have done a very fast preheating and then it has out-gassed. All the material has sputtered and during this sputtering process, the solder bead has been thrown out close to the package or under the package or close to a copper pad. So, out-gassing of solvents due to poor thermal profile from the solder paste material during reflow is one of the main reasons for solder bead formation. Now again, I have to repeat that you have to do a very good thermal profiling. I am going to show later in this lecture, what is a thermal profile and how do you take care of and study thermal profile for different segments. In this case, you must know what is the content in a solder paste, what kind of solvents are there, do you do gradual heating to remove the solvents. Here you can see in this picture, a solder paste has been dispensed on pads and even before attachment of the component, you can see they are not adhering onto the copper pad. You can see bead formation already, because at room temperature or during tacky cure, you can see that the solvents have gone and the solder paste has disintegrated. It is a material that has lost its shelf life or during the synthesis of the solder paste, the media has not been properly utilized in holding all the solder paste material that is the conductive material together. That is a very key issue in the synthesis of solder paste. So, thermal profiling plus the poor quality of solder paste could lead to solder beads.

(Refer Slide Time: 31:31)



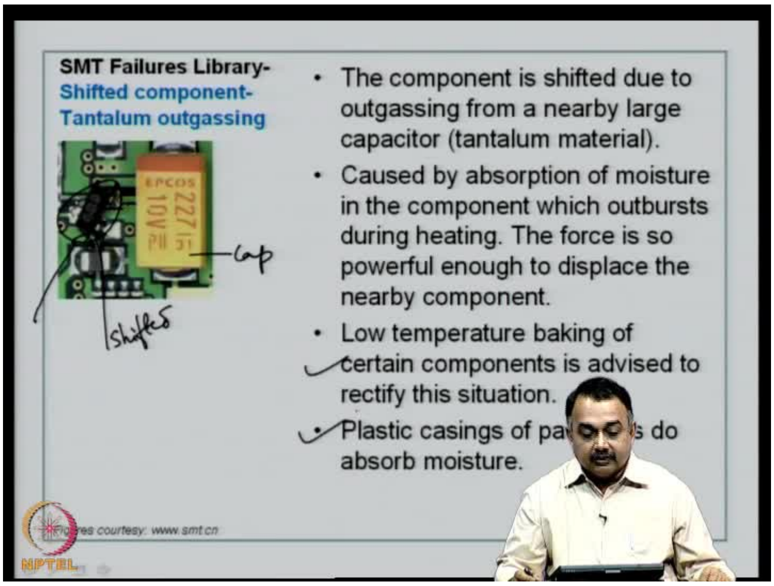
The other defect is voiding. You can see here, this is a void in a very good otherwise very good solder joint. Voiding is caused by out-gassing of solvents in the solder. You can assume that this is a very good joint formed but unfortunately there is a defect which has been caused by out-gassing of the solvent in the solder paste material and that has created a void. Flux material could be the reason for this defect. Flux is present in the solder paste as you know and poor thermal profiling for such solder paste can cause voids. Here again, you should not approach the reflow temperature fairly fast. It has to be very gradual. You have to look at the substrate material Tg. You have to look at what solvents are being used, what binder is used, and what flux material is used and accordingly slowly remove these solvents or evaporate the solvents very gradually. If you give a very high heat rate of heating in the thermal profile that you have generated then the solvents could just bubble out and the resulting out-gassing could create voids. Soaking zone as we call it in the thermal profiling has to be increased. You have to activate the board very slowly or you can keep it activated for a longer time in the soaking zone before you go to a reflow zone. Increase the pre-heat time of the board so that your copper pad and the substrate are nicely activated and then it goes into the soaking zone where the solder paste material and the flux material are activated gradually. In such cases, if you think that air is a deterrent and you can probably work with nitrogen atmosphere; so that you can control these kinds of issues.

(Refer Slide Time: 33:49)



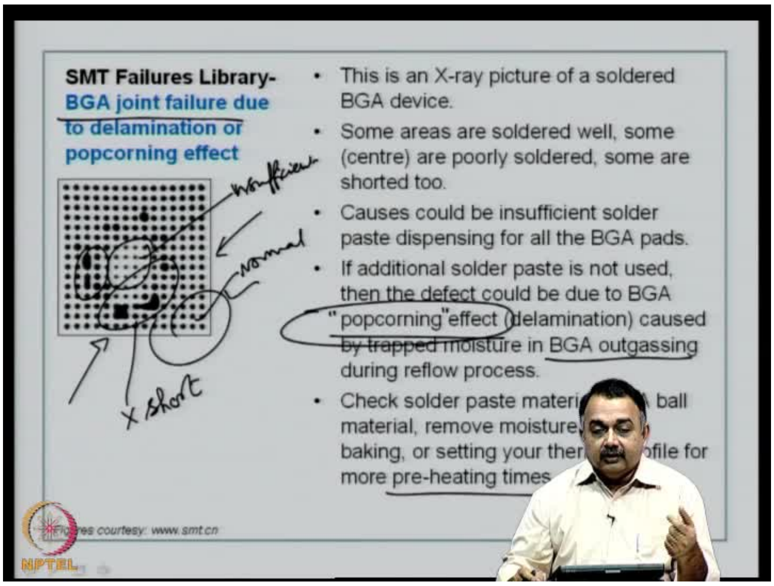
Now you can see flipped component here. This is supposed to be at the bottom. This side is supposed to be at the bottom. So, it is a flipped component. This is a defect; wrong placement, upside down component, mostly seen in resistor placement process. A flipped chip capacitor may not be recognizable and may not cause problems. I think I have mentioned that in most cases, chip capacitors do not have any labels. So, it is very important that you look at the packaging material or the reel; where the markings are done on the outside of the tape to recognize what the capacitor value and the case form is. If you look at it, it may not cause a major problem. But there will be less insulation between the component and the track. This could be an issue if you have a flipped component, especially in the case of resistors. Now, rework on the component, if you want to do. You have to manually desolder the material or the package and then solder it again with a new component. This is the only solution for replacing a flipped component.

(Refer Slide Time: 35:20)



Now, the other failure is a shifted component and tantalum out-gassing. Here, you can see there is a large tantalum capacitor and here you can see a small component. Now, the solder paste has been dispensed; reflow has been initiated. During the reflow process, the component here you see has been shifted from its footprint due to out-gassing of gases from this particular large tantalum capacitor. The reason is simple. The capacitor has absorbed moisture during its storage period. Now, what you have done is, you have directly used the component without doing a preheating. Then during the preheating of the soldering process, the water is being removed from the component. During the heating process, there is an outburst of the gases. There is out-gassing and this is so powerful that it displaces the nearby small component. How does it displace? The small component is actually swimming in the solder paste during the reflow process before it is cooled. So, it is very easy to displace the nearby component if it is light weight. So, low temperature baking of certain components is highly recommended for these kinds of out-gassing defects. Plastic casings of packages do absorb moisture. So if you look at the data sheet of packages, you have to necessarily look at this problem, because this is a global phenomenon. Many defects have been seen due to moisture absorption by packages and especially large packages like this capacitor. Therefore, you have to do a prebaking before you actually send them out to a reflow soldering process.

(Refer Slide Time: 37:35)

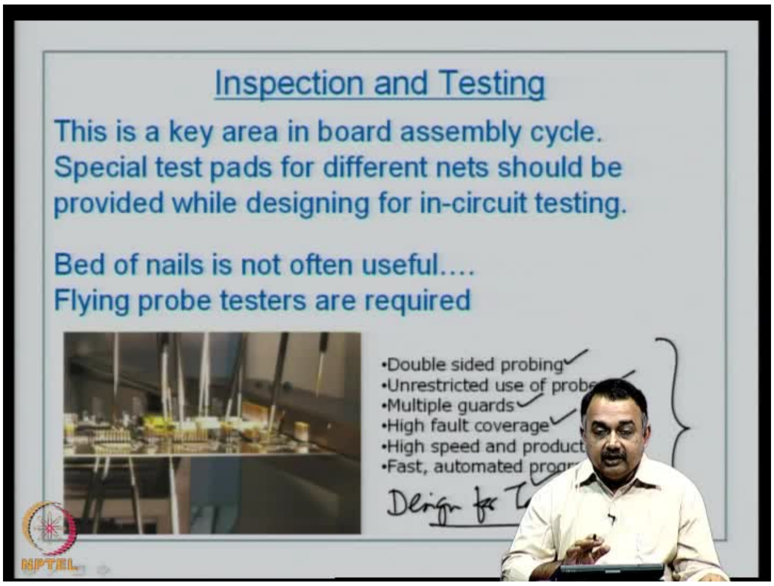


Now, the other failure that we would like to highlight here is a BGA joint failure. Just like other components, BGA will have a failure. The difficulty with observing BGA failure is that it is an area array package. The connections are at the bottom of the package. So, it is an array of solder ball. As you can see here, you have to use an X-Ray picture of a soldered BGA device to understand what kind of defects are there. Now, if there is a thermal stress built on a BGA during the working of the board, you have to look at and if you observed a defect and you are sure about that the BGA has got a defect, then you have to use an X-Ray, picture X-Ray micrograph and see the defect whether there is a cracking of the die under the BGA or if there is a shorting between two adjacent solder balls or some other defect. We will look at a couple of defects here in this particular slide.

What you are seeing here is an X-Ray of a BGA that has been soldered. Some areas are soldered well. So, these are highly perfect. Some of the areas you can see here, they are shorted. So this is a short. This is not accepted. In some cases, you can see there is insufficient solder compared to the normal areas. This is a normal area. You can see here at the center, there is insufficient solder. These are major defects in this particular viewgraph that you are seeing. Either the solder paste has not been dispensed properly if you are using solder paste to attach a BGA. If you have not used additional solder paste for attaching a BGA that means you are using the BGA solder ball itself to reflow and then form an interconnect to the board. Then this could be due to a delamination or popcorning effect as you call it. Very common in BGAs. Again, this is due to the plastic body of the BGA absorbing moisture. Epoxy, as you know is well known for its moisture absorption and before you are doing the process you have to do a prebake. If you have not done a prebake then you will see this kind of a popcorning effect. So, avoid BGA out-gassing, because it is very difficult to desolder a BGA and then repair at that particular site. It is very easy to do a repair for a through-hole component or an SMD device but if it is a large pin count BGA, then you have to spend a lot of time on reworking the device.

So check the solder paste material, BGA, ball material, remove moisture by prebaking or setting a thermal profile for more preheating times. If you have not done prebaking, to be on the safe side, in your thermal profile spend more time for the board to be in a prebake area and then go into a soaking zone where the solder paste gets activated and then go to the reflow zone where you attain peak soldering temperatures.

(Refer Slide Time: 41:38)



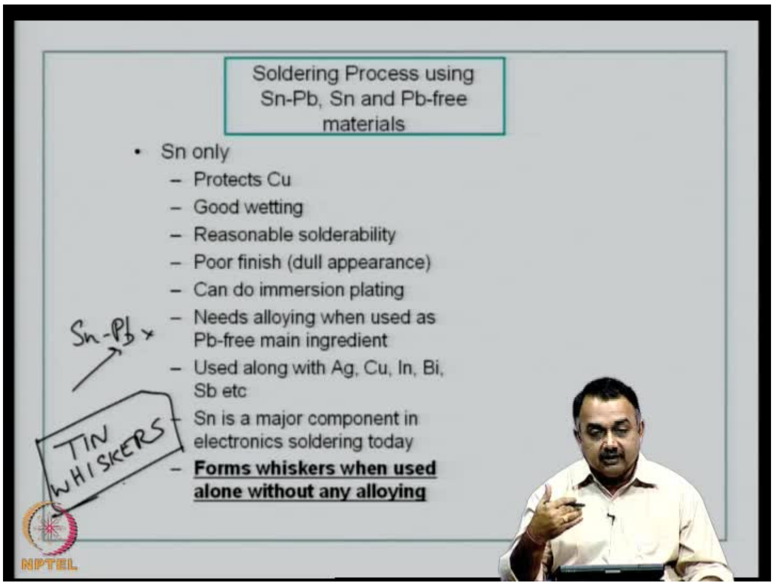
So, these are some of the defects that we have seen. There are many such defects but due to lack of time, we will conclude with this. At any process after the assembly is done, you have to do inspection and testing. This is a key area in board assembly cycle. Special test pads for different nets on the board have to be provided while designing for in-circuit testing because just like you did a bare board testing for the finished bare board, you have to do a testing for assembled boards to see that your assembled process or your assembly using a particular solder material or a reflow or a wave soldering has been done to perfection. Because there are different profiles of components on the board you have to use flying probe testers. Typically there will be various probes that are available in the equipment and that will go net by net based on your CAD information and then look at interconnection or connectivity.

So you will have one probe stick stationed in the net in a particular pad and then you will see the various other probes going through and completing the net and giving you information about the shorts or open in that particular board after the assembly process is over. So, you can troubleshoot if there are any problems.

Now, typically for flying probe testers, you should have double-sided probing. Unrestricted use of probes, multiple guards, high fault coverage high speed and productivity should be there because in large volume you cannot waste time on this testing and fast automated programming methods are normally available in current equipments.

The question is here again, it emphasizes the need for looking at the designers has to look at design for testability. So have your design in such a way that your boards can be tested by automated test equipment using your CAD data or the layout data so that you spend minimum time for testing. Although testing is very critical, this could also go in for an advanced test procedures where you look at electrical issues, electrical parameters. Although you would have done simulation before, testing of a finished assembled board becomes a very integral part of packaging. That completes the assembly process.

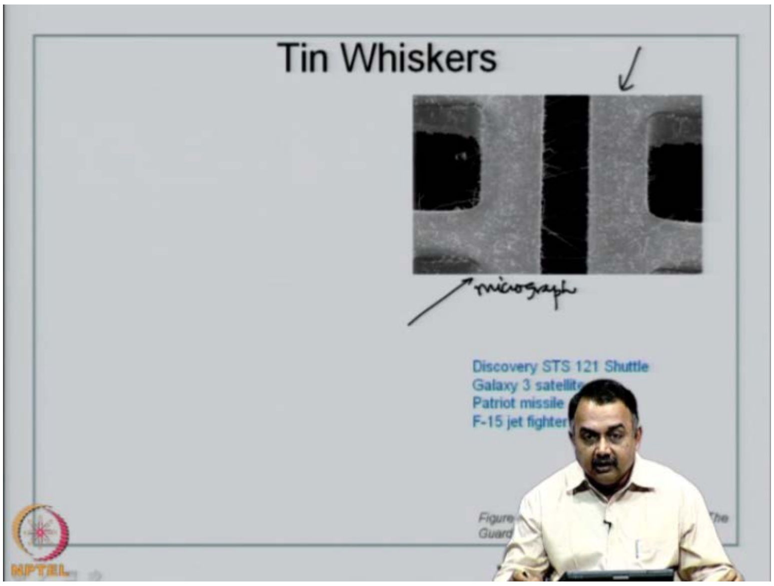
(Refer Slide Time: 44:32)



We will now look at some specific points regarding the soldering process using tin lead, tin and lead-free materials. Firstly, we will look at tin only. Suppose you are doing tin only soldering process, what does tin do? It protects copper. It provides a very good wetting surface for the copper and it provides a very good wetting surface for your component to get attached and provide a reliable joint. The next thing is, tin provides a reasonable amount of solderability. It has been well established for over the years and people have understood tin very much in electronics as a soldering material. If you use tin alone, the finish is rather poor. It is kind of dull compared to a tin lead finish. Tin lead finish is usually very bright, very attractive and aesthetic in terms of the color. But if we use tin only, which normally you use in prototyping small volume, you will get a very poor finish. You can do simple immersion plating for tin on copper. Very less expensive.

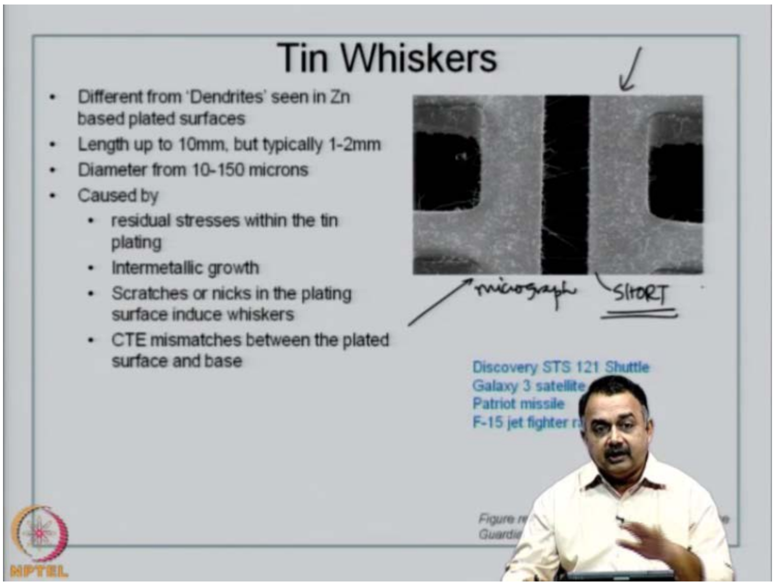
When you want to use lead-free soldering, in that case tin will be the main ingredient. Normally, we are now used to tin lead as a major alloy in electronics. If you want to remove the lead, then tin becomes a major alloying element along with other materials like silver, copper, bismuth, indium, and so on, which you are going to see shortly. So, it will be used along with copper, silver, indium, bismuth, antimony and then tin is a major component in electronics soldering today undisputed. The only problem with tin which has been noticed for years now has been that if you use tin alone, you see whiskers. It is a defect when it is used alone without alloying. So, tin whiskers are another defect. As we have seen defects during the reflow soldering process, this defect will be seen let us say, a year after the board has been assembled or 2 years, 3 after the board has been assembled, utilized and in operation. So, it is a very slow phenomenon and I will explain to you what tin whisker is and how it looks like?

(Refer Slide Time: 48:17)



This picture here, this has been taken from a Kurt Jacobsen who wrote an article in The Guardian third April 2008. This is from this newspaper that this picture has been taken. It gives a very good illustration or an example of how tin whiskers. This is a micrograph. It is a micrograph of tin whisker. You can see there are two tracks and you can see thin lines of tin from one end of the board or the interconnect spreading slowly and contacting the other end resulting in a short. So, tin whiskers can create major problems. It can create large spark. It can create an arc. It can create a quick short and destroy the complete PCB because of this shorting. They are different from Dendrites. If you are aware of this term Dendrites, which you normally see in zinc plated surfaces, tin whiskers are normally seen in tin plated surfaces. So, here in PCB, this particular phenomenon that you are seeing here in this illustration or micrograph is an example of copper plated with tin. There is no other intermediate interface and the components have been soldered on tin surface which means you are providing more tin over time and these whiskers are formed due to various factors which I will explain shortly.

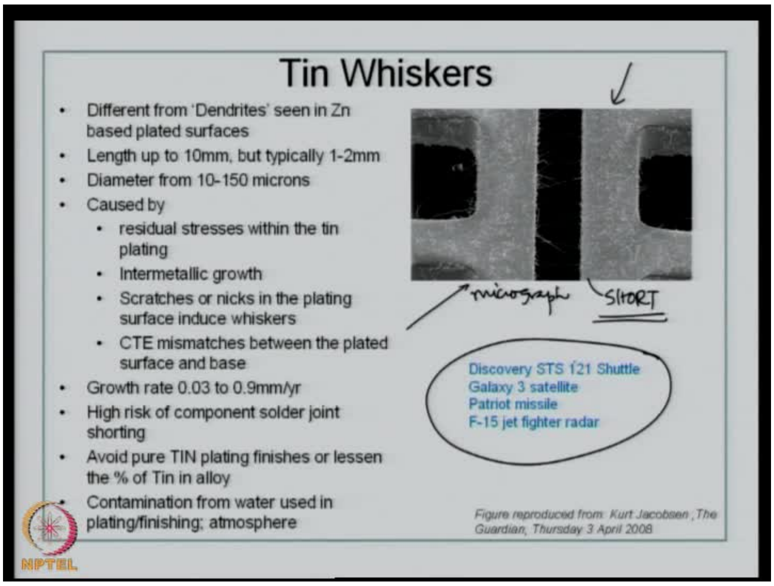
(Reference Slide Time: 49:57)



The physical parameters of these tin whiskers: typically they can grow up to 10 mm, typically about 1 to 2 mm very fast, diameters ranging from 10 to 150 micron. They are caused by residual stresses within the tin plating.

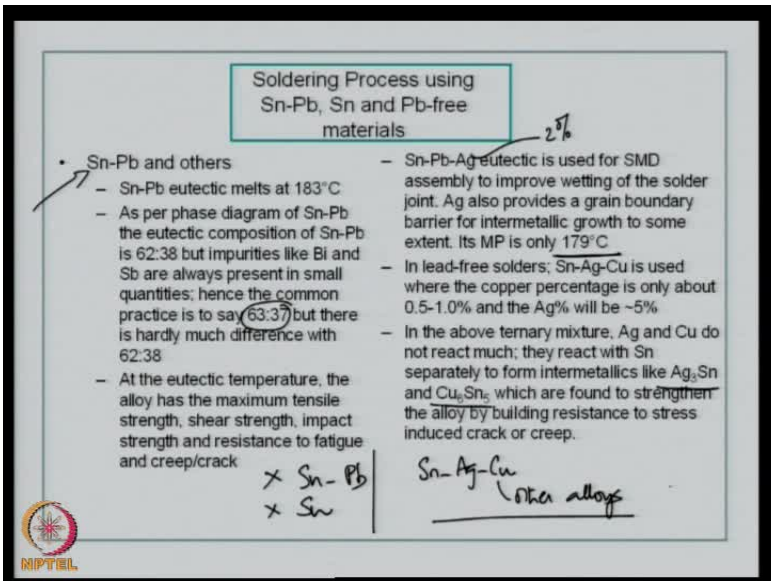
In a tin plating finish, if you create a nick or a scratch post process and that will be a very good source for tin whiskers to grow because that is something kind of an area where the stresses could be released from the tin thickness that you have plated. Then these whiskers are basically due to the intermetallic growth that your tin exhibits from the copper surface. So, scratches or nicks in the plating surface will induce the whiskers could be the starting point for formation of whiskers and these whiskers grow typically 1 to 2 mm per year kind of a thing. This is also caused by CTE mismatch that is coefficient of thermal expansion, mismatch between the plated surface and the base. So, growth rate is 0.03 to 0.9 mm per year that is almost 1 mm per year and typically not observed immediately but during the working of the product. One fine day, you find a defect in the product and if you troubleshoot and see on if it is a tin plated board you can definitely expect tin whiskers. So, it has a high risk of component solder joint shorting and the best thing is to avoid pure tin plating at all cost or lesser the percentage of tin in that alloy and it could also come from contamination of the water used that is in the plating process or cleaning process and also the atmosphere in which it is stored or used. The PCB could be used in very corrosive atmosphere because of the product that is being used in that application and there is a very interesting story about tin whiskers because I want to just tell you how very important it is to handle materials. Tin, lead is very safe, but because we are removing lead now because of ROHS legislation you are going to use tin in conjunction with other materials which is safe but if we use large percentage of tin you are going to end up with a lot of defects like these.

(Reference Slide Time: 53:37)



Typically the problem of tin whiskers was in 2005, Millstone nuclear generating plant in Connecticut observed that is in USA, observed a shutdown of its plant because there is a steam pressure line which short circuited and this was defected. This was because of a defect in one of the PCBs in the electronics that have been used to control these pressure steam pressure units, so that is a major setback and they had to replace these with better PCBs and also companies like Swatch that is a Swiss watch making company in 2006 also had to recall a majority of its products because of tin whiskers observed in the PCB. So these are information that is available in the web about the defects that have been major catastrophes. I have listed here some of the major catastrophes that have been observed in the larger scale; that is the Discovery Shuttle, Galaxy3 Shuttle, satellite Patriot missile, F15 jet fighter radar all had reported, we have in the literature about defects caused by tin whiskers in Printed Circuit Boards. So, this information has to be keenly observed when we are working with lead-free materials especially tin based.

(Refer Slide Time: 54:09)



Now, we will look at soldering process using tin lead, tin and lead-free materials. We have seen with tin. Now, we will look at tin lead and others. Tin lead eutectic melts at 183 that is the composition of tin lead at 63-37, tin 63 percent lead 37 seven percent. The eutectic temperature is 183 C which is the lowest of tin lead alloys.

Now, as per the phase diagram of tin lead, the eutectic composition of tin lead is 6238 but impurities like bismuth and antimony which are always present in small quantities in these materials. Hence the common terminology that is used to describe this eutectic composition is 63 37 rather than 62 28 38 but there is hardly much difference in properties between 63 37 and 62 38. So, for all purposes we will talk about 63-37 percentages of tin and lead.

At the eutectic temperature, the alloy has the maximum tensile strength, shear strength is good, impact strength is good, and resistance to fatigue and creep or crack has been in favor of using 6337 alloy. Now, tin lead silver eutectic has been used for a long time and especially for surface mount device assembly, so which means you are using a small percentage of typically about 2 percent of silver is used to improve the wetting of the solder joint. Silver also provides a grain boundary barrier for intermetallic growth which sometimes has been considered harmful for the solder joint and the melting point has been lowered by about 4 degrees so typically from 183 you can go up to 179 if you go to tin lead silver composition of alloy for surface mount device application.

In lead-free solders, tin silver copper is used. Very popular, where the copper percentage is only about 0.5 to 1 percent and silver will be around 5 percent in the above ternary mixture that is the tin silver copper ternary mixture. Silver and copper do not react much. They react with tin separately to form intermetallic phases like Ag_3Sn or Cu_6Sn_5 which are found in literature to strengthen the alloy by building resistance to stress or fatigue or stress induced crack or creep which means the reliability of this system is definitely much better compared to other systems. We are now at alternatives to tin lead, we talked about tin lead.Tin alone is not recommended. Tin lead is not recommended because of legislations. So, we are now looking at tin silver copper alloy. There are other alloys also which we will be discussing shortly as alternatives to tin lead material.

So, in the next class we will look at other materials for tin lead and we will also discuss in detail about how to use thermal profiling for your assembly. I will also work out online, how you can look at a particular situation of a substrate, various components of solder paste material and arrive at a good thermal profile for a system.