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. # 34

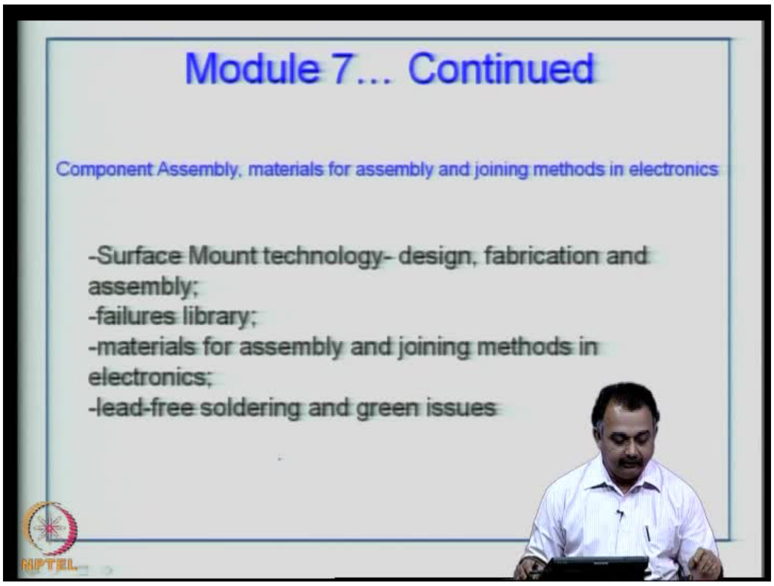
Solders

Wetting of solders

Flux and its properties

Defects in wave soldering

(Refer Slide Time: 00:15).

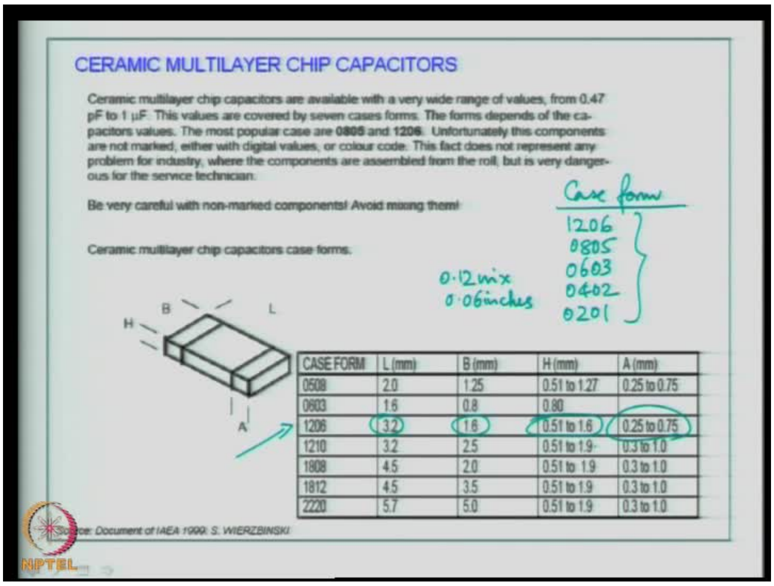


We will begin with another lecture that is concerned with module 7 of this video course on electronic systems packaging. As you have seen in the previous lectures concerning this module, we are discussing component assembly, materials for assembly and joining methods in electronics. When we say joining methods, typically, we are looking at soldering process. As you know in the previous modules, we have had a glimpse about the soldering methods and also methods that can replace soldering. For example, conductive adhesive is one method that is being used to join flip chip or a BGA or a

CSP. So, there are different joining methods, but we will focus on the mass or volume manufacturing scenario, which is soldering.

And just to recap, we are discussing surface mount technology-design, fabrication and assembly issues. We will also look at failures library, which means we will study what kind of failures will happen or that can be expected if your process design is faulty. We will also look at materials for assembly in joining methods and also briefly, lead free soldering process, the thermal profile issues associated with lead free soldering and then environment issues like replacement materials for lead.

(Refer Slide Time: 02:01).



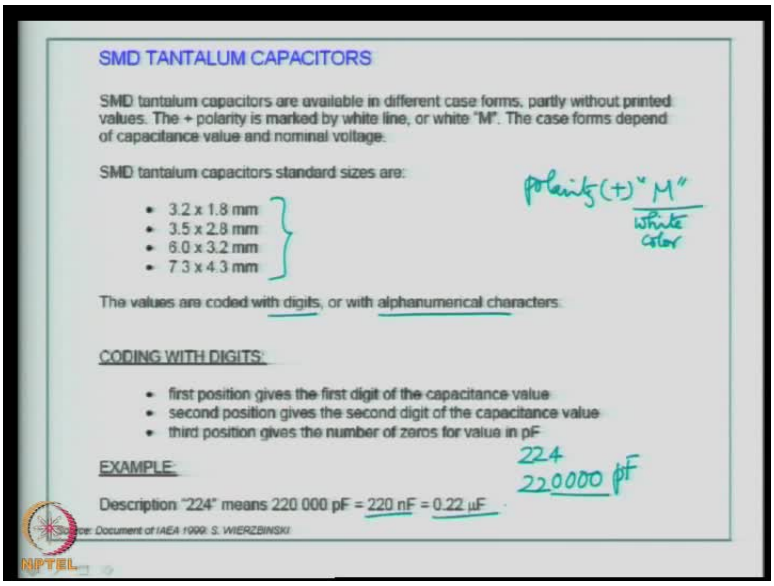
In the last class, we have been discussing the case forms of surface mount devices; especially, chip components like chip capacitors and chip resistors. We have seen how the chip resistor is typically reference designated; what kind of case forms it can carry; and also what kind of values are depicted by certain kind of notation on the resistor in terms of reference designations. Today, we will look at chip capacitors - ceramic multilayer chip capacitors. They are available in a very wide range of values: typically from 0.47 picofarads to 1 microfarad. These values are covered by 7 case forms. We need to know what a case from is. Typically, the case from can be 1206, 0805, 0603, 0402, 0201 and so on. When I say 1206, that means the dimensions of the chip component. In this case, the capacitor could be 0.12 inches by 0.06 inches and so on for the other case forms. I just hope that you are very clear about the case form notations.

This is the similar thing that we have seen in the resistors. In the resistors, the ohmic value representation - the denotation, we have seen; how they are depicted on the component. Now, in the case of capacitors, mostly these components are not marked, either with digital values or color code. This fact does not represent any problem for the industry, where the components are assembled from the roll. Therefore, the technician is really aware of the values of the capacitor at the loading stage itself in the pick and place equipment. But it can be an issue that it can be faulty. It can be very dangerous for the service technician to end up with lot of defects, but an experienced technician can manage in identifying the right side of the capacitor and the right value of the capacitor, when loading in the pick and place machine.

So, be very careful with non-marked items. Avoid mixing them. Look at the labels that are pasted in the rolls or the tape to identify the value of the capacitor. Storage, again, is very important in the bins of the equipment. It could be a small desktop equipment or it could be huge equipment - large volume. But, right storage in the right bin with proper marking is essential.

Here in the case of capacitors, again, you can see the dimensions of various case forms. For example, here, 1206 depicts 3.2 mm by 1.6 mm - that is the length and breadth. Height could be from 0.51 to 1.6 mm and the area available for the stub on the contact is also given here.

(Refer Slide Time: 06:23).

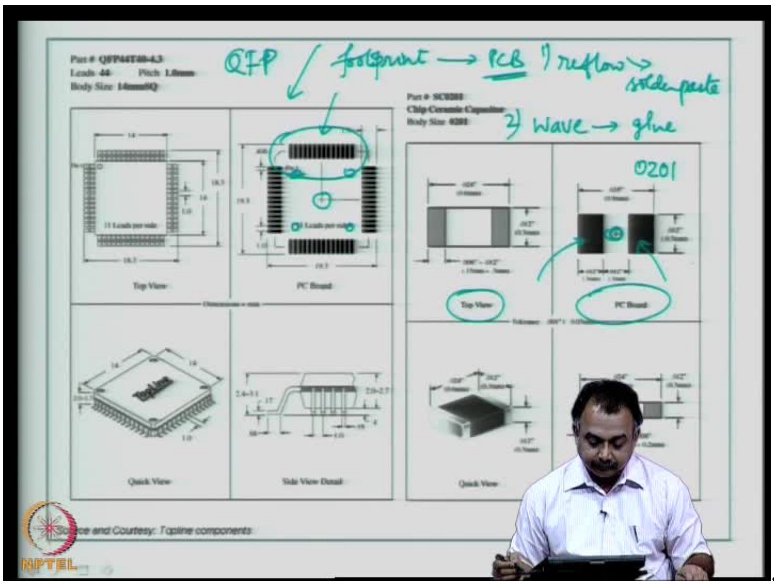


You can look at such data sheets available and then identify what kind of pads need to be used in the footprint design of a capacitor. Coming to the case of a surface mount device tantalum capacitor, they are available also in different case forms, like we have seen with multilayer ceramic capacitors, partly without printed values. Therefore, the polarity of such devices, where you mark positive or negative - in this case the positive - is marked by a white line on the device or white M marking. This is marked in white color printing.

The case form depends on the capacitance value and the nominal voltage supply. The SND tantalum capacitors standard sizes are denoted here. I can read it for you: 3.2 by 1.8 mm, 3.5 by 2.8, 6 by 3.2 and 7.3 by 4.3 and these values are coded with digits or with alphanumerical characters.

Now, coding with digits: For example, if you have, as it is mentioned here, 224 - if this is the coding, as we have seen in the case of resistors, 22 followed by 4 zeros is the value of capacitors. The first position gives the first digit of the capacitance value, the second position gives the second digit of the capacitance value and the third position - here, as an example, if it is 4, it means you have to enter 4 zeros - and this is the total value in picofarads, which is equal to 220 nanofarads or 0.22 micro farads. This is example for you to look at the notation or the marking done on the Tantalum surface mount device capacitors.

(Refer Slide Time: 08:27)



I will explain an important aspect from these two sets of figures: the one of the left here shows the footprint of a QFP package. This is available from any manufacturer. If you buy a component from a manufacturer A or B, you will get from the website or in hard copy, the foot print of the surface mount device.

Here, what you are seeing is a QFP device - Quad Flat Pack. This is the footprint. Why do you require footprint information? This is the footprint that is established on the Printed Circuit Board, where the respective components will come and sit. So, you require to know the actual dimensions of each of the pads.

In this case, you have the quad arrangement and then, in this particular example, you have the quad flat pack pin numbering starting from pin number 1 and so on. The dimensions of each of the side are denoted here and the dimensions of each of the pads, where the QFP will land is given and the pitch of this particular example here is 1 mm; the leads are 44. Based on this, you can identify the total area or the component density of a package and secondly, you will use this information to drop your solder paste.

If you are going in for reflow soldering, then you will have to know the pad area available for dispensing your solder paste. In the earlier class, we have seen how solder paste is dispensed on the copper pad or the tin platted copper pad and therefore, here we have to know the dimensions to adjust the volume of the solder paste.

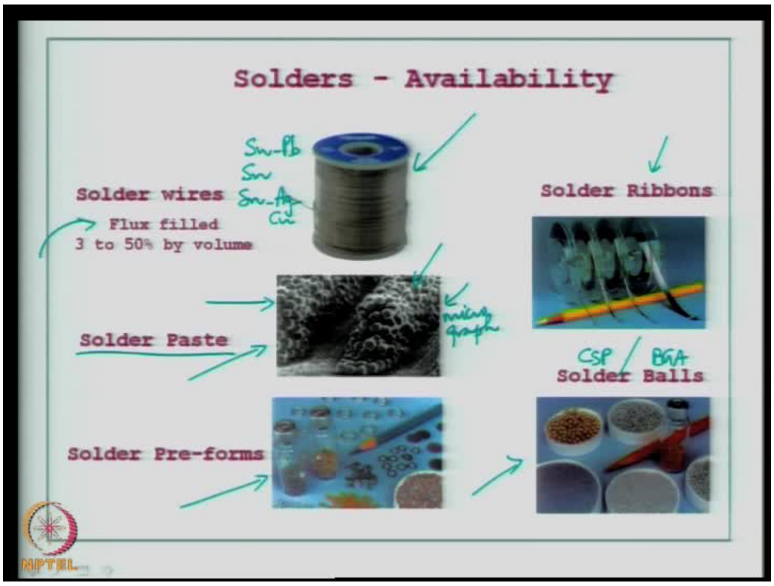
So, knowing the dimensions here, you can adjust your syringe dispensing or you can also go in for stencil printing. The dimensions or the foot print information is very important for the kind of soldering process that you are engaged in. If it is a reflow process, then you will dispense solder paste. All of you are aware of this. But if you are going in for at any point of time, some kind of an alternate like wave soldering, then you have to dispense glue.

I think you are clear about this. The first option is the reflow, the second is a wave. In the case of wave, you have to dispense the glue to hold the component. In that case, you can dispense the glue at the center of the footprint. In some cases, they also dispense additional areas like these to dispense the glue, so that the component is firmly held during the full cure process, before it goes for a wave soldering process. Whether, you actually use a QFP, with pitch of 1 mm for wave soldering or not, is another issue. But in some cases, where you have hybrid boards and few assembly components mounted and if you decide on an option of wave soldering then you have to have these kind of information on the Printed Circuit Board during the preassembly process.

Now, on the right side here, you see that there is information available for a surface mount device like the resistor or a capacitor. In this case, it is a capacitor. It is a 0201 case form, which means 0.02 to inches by 0.01 inches. You can see the dimensions given here. This is the top view and this is the Printed Circuit Board view.

Now, in this case also, you can go in for dispensing the solder paste right on the pad here and then do a reflow process or if you are going in wave soldering, you dispense the glue at the center of the area covering the component area on the Printed Circuit Board and then do a complete cure of the glue before it goes for wave soldering. So, this kind of information is very useful. You can upload this information if you are creating a library or a footprint from the CAD package to your assembly process. So, the coordinates that are picked from this data is available to you. These are very useful information for assembly processes.

(Refer Slide Time: 13:55)



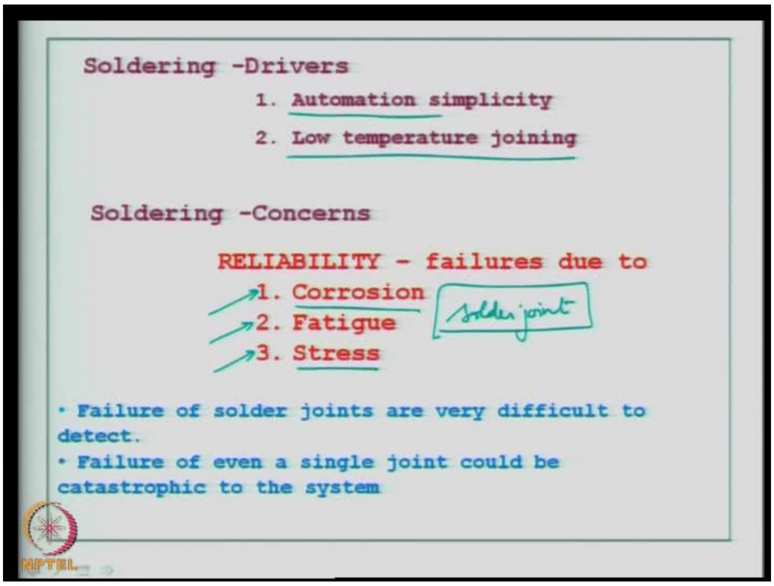
Now, we look at the basic soldering process. Before that, we need to understand what the solder is and then how you handle solders. Solder wire is the most common form. We can see here. This is a solder wire that is available in different lengths, different diameters depending upon your requirement. Today's solder wires come with flux coated. Earlier days, we were having a flux as a separate entity and when you do hand soldering, you will use flux separately to wipe the areas on the board; that is, the copper areas and as you know flux has got different applications when we use them during assembly on a Printed circuit Board.

Today, solder wires could be tin-lead or tin or it could be other components like tin-silver, copper and so on. Depending upon the material choice, you can get the wires and if you carefully look at the data sheet, you will see that it is filled with some kind of a flux component to ease this soldering process and also reduce the temperature of soldering and remove the oxides present on the Printed Circuit Board surface.

Solder wire is one thing. You also get solder ribbons. If you are looking at large areas soldering or large electro mechanical component to be soldered then you has solder pre forms. When you use a reflow soldering, you are interested in using a solder paste. This is a micro graph you see here. This is the micro graph of a solder paste. You can see small granules, very small fine particles of the conductor and this conductor particles are dispersed in a media, typically an organic media, like epoxy. This also contains flux, a binder, and so on. It will also contain some solvents to keep the media intact until it is used. Therefore, it has to retain some kind of a viscosity for application as a syringe dispensing process or during a stencil printing process.

You have to understand what kinds of solvents are used, so that you can dilute the material, if you want, in certain applications, and use it. Then, there is another format in which solder is available and that is in the form of solder balls. Typically, as you know if you are using a BGA or a CSP, the IO pads are typically in the form of a solder ball grid. Therefore, today in the market you have the availability of solder balls of various dimensions, diameters; and you could easily use them to repair your BGA or to make a new connection in your lab itself to an existing BGA component, in which a few solder balls have been damaged during repair and rework.

(Refer Slide Time: 17:45).

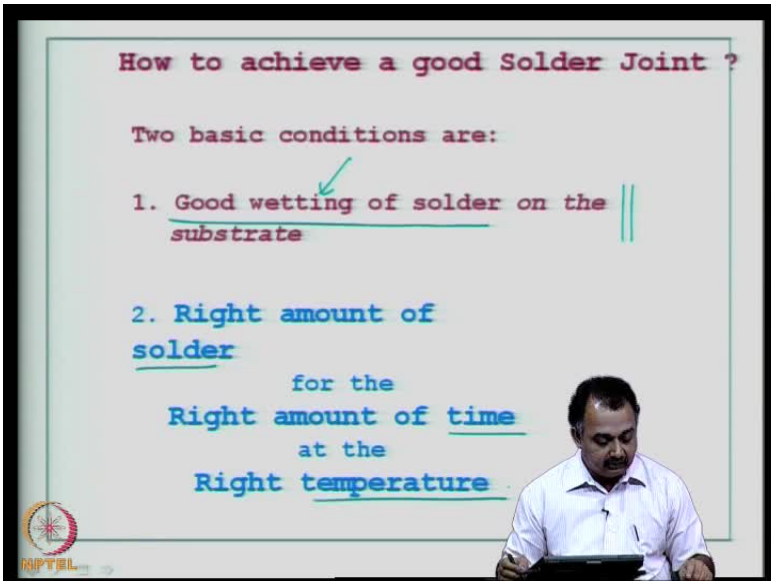


So, the drivers for soldering processes, typically in large volume manufacturing or in industry, is the simplicity that it should provide and enhance the automation process to increase the through-put and then finally in the product, the yield should be very high with few defects and high reliability. Secondly, an important thing is the right choice of material, so that you can establish a low temperature soldering. Obviously, in electronic manufacturing, you are concerned with the energy usage. Therefore, new material should aid in being energy efficient. Therefore, solder materials that are being used today have to be considered from the point of view of low temperature, so that you are saving in energy. The concern with any new material on the soldering process is that reliability is an issue. You always have to understand reliability when you use a new material. If you are using an existing material, then you have to go through the previous experiences and data sheets, or the experiences of various companies that have been using these materials for long time. One of the important things is corrosion; second is fatigue and the third is stress. When you look at solder joint, that is where all the failure takes place in an electronic product.

As I mentioned in this example previously, we take a mobile phone and if you dropping it a few times there is the failure. The mechanical shock results in a fatigue or a crack at the solder joint resulting in an electrical open. Therefore, this is one example of a solder joint failure and the other thing is due to thermal stress build-up at the solder joint. There could be load, which can result in material stress and over period of time, this will result in a failure.

Here again, the failure will be an electrical open or a short. Electro chemical corrosion or simply chemical corrosion is another example which can be noticed in electronic products. This can lead to weak solder joints or it can lead new interface mechanisms that can be experienced or seen at the solder joint resulting in a weak joint or a dry joint or void created at the solder joint. So, fatigue of solder joints are very difficult to detect. You need to have a very good troubleshooting procedure to understand where exactly the failure has occurred and to understand how this failure has occurred. Failure of even a single joint could be catastrophic to the system. Therefore, component choice is one thing; the board is another issue, that is, what kind of material you are using in the board; and finally, the type of solder material that you are using along with the soldering process will lead to understand the entire mechanism for a highly reliable product.

(Refer Slide Time: 21:23)

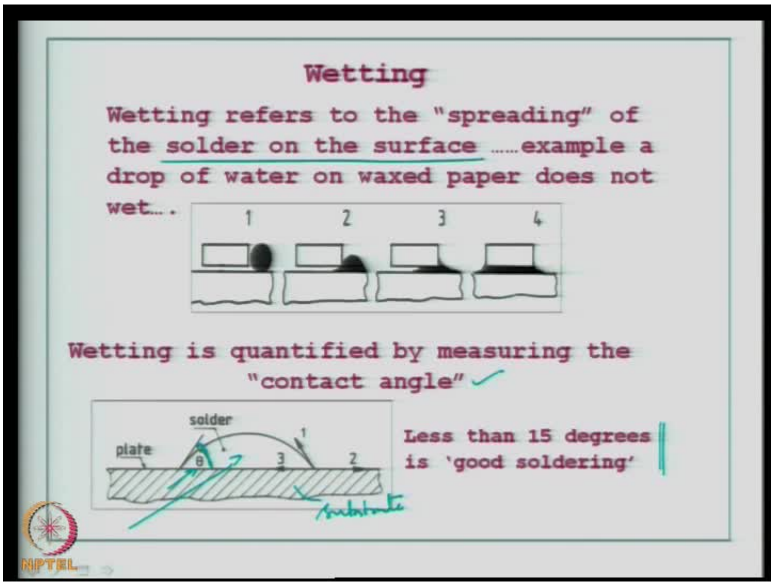


Now how to achieve a good solder joint? The basic conditions are:

There should be good wetting of the solder. If you look at the board or mother board of a PC or a typical product hand-held product and if you look at the solder joints, the striking example is a shining nature of this solder joint. You have to look carefully through microscope at the solder joints and see how well the wetting has taken place. Wetting ensures that your solder joint quality is good; the materials have reflowed; for example, in the case of a wave soldering, the molten wave of the material has adhered well to the through-hole component and so on. So, good wetting of the component, along with the solder on the substrate, is essential.

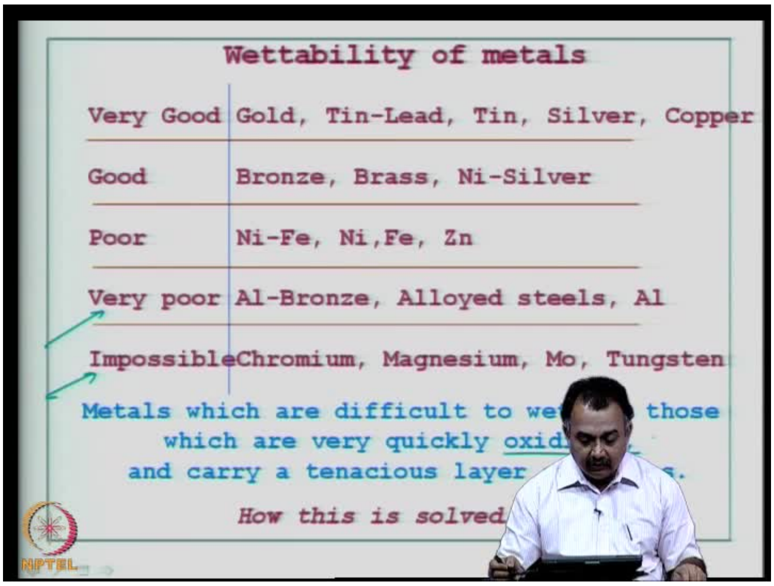
The second thing is using the right amount of solder for the right amount of time at the right temperature. These three go in unison when you establish a joining process.

(Refer Slide Time: 22:38)



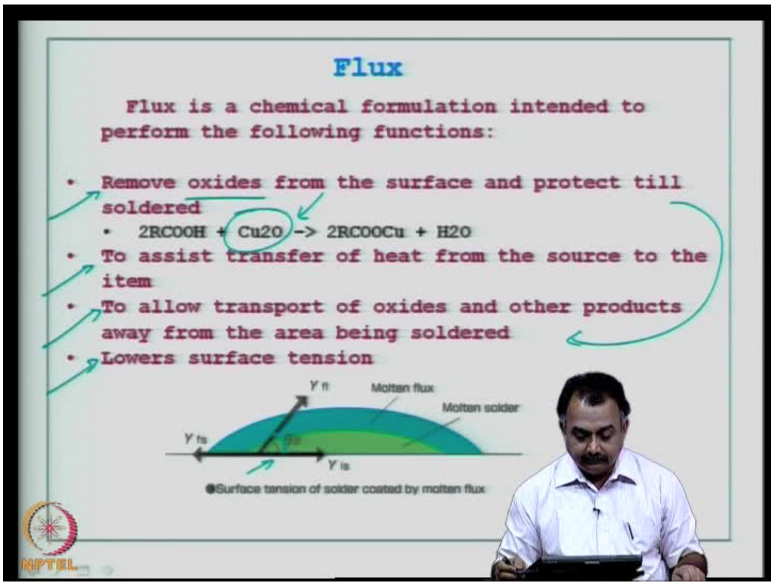
Let us look at what is wetting. Wetting refers to the spreading of the solder on the surface. There is surface tension that aids the wetting process. Surface tension, as we have seen even in the BGA self-alignment, even if you do a mis-registration, you are able to attain or achieve 100 percent yield because of the surface tension of the solder material with the component pad and the substrate pad. Here, wetting refers to the spreading of the solder on the surface. For example, a drop of water on waxed paper does not wet. So, we do not want that kind of a situation. It should not wet on the dielectric material here. So, the dielectric should be free of the soldering material, after the soldering process is complete. It could be a wave soldering process or it could be a reflow soldering process. Wetting is quantified by measuring the contact angle.

If you look at this particular example here in this illustration, this is the substrate and (Refer Slide Time: 25:01) then you have certain amount of solder applied on the metallic plate, here; you can see that a meniscus is formed; and then the extent to which the material spreads on the surface is important. According to literature, it says less than 15 degree contact- as you can see here, is known as the contact angle – less than 15 degree is good soldering. Because if it spreads too much again, it does not exhibit good wetting. So, a host of environmental issues are at the site or the status of the substrate. The metallic contact point is very important to establish a wet joint, especially, for example, preparing the substrate before applying the solvent.



Let us look at materials that are considered wettable in this soldering process. The material that have very good wettability are gold, tin-lead, tin, silver, and copper; fairly good are bronze, brass, nickel and silver; poor wettability - nickel iron, nickel, iron and zinc; very poor wettability aluminum-bronze; alloyed steels, aluminum etcetera and impossible to wet is chromium, magnesium, molybdenum, tungsten and so on. So, materials which are difficult to wet are those which are very quickly oxidized.

(Refer Slide Time: 26:16)



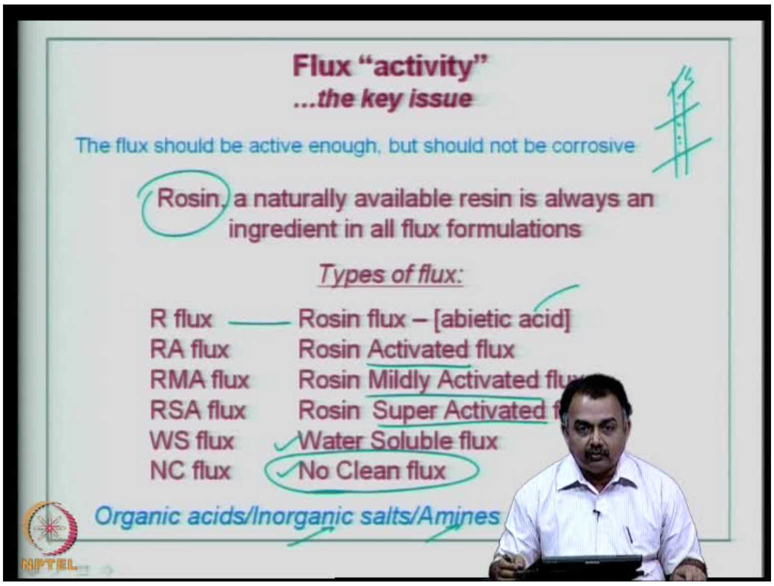
Materials surface which gets oxidized wear quickly and which have a very tenacious layer of oxide present on a surface cannot wet. The same is the case with a Printed Circuit Board. If you are not preparing the surface properly, even though you are using tin-lead or tin or gold, then you can expect poor wettability. So, how is this solved? We use material called flux. I mentioned earlier that today flux is present as an ingredient in most of the solder materials of the shelf including wires or solder paste or solder balls. Flux is chemical formulation intended to perform the following functions:

What does it do? Firstly, it removes the oxides which pose a great problem for us and which react with the solder material, it will form new interfaces, and new intermetallic layers which can become very weak; so, that is why we say remove the oxides. Now, that itself is the very big topic - to look at the intermetallic layers formed at the solder joints. After cleaning the surface, whether it is gold or copper or a tin-lead or tin or tin-silver-copper, the surface has to be protected till the time it is soldered. So, we make sure that the oxides are removed from the surface. That is essentially the process that is depicted in this chemical equation here.

Then, the second important point or the functionality of the flux is to assist, transfer of heat from the source to the item when you do the soldering process. Because you want to dissipate the heat quickly, you want to make sure the there is no heat built-up during the component. Therefore, flux can aid in the transfer of heat and make a good wet solder joint.

Thirdly, to allow transport of oxides and other products away from the area being soldered, which is again a functionality that you have seen in the first point. We have to make sure that the oxides are transported, but then these oxides have to be removed later by a cleaning process, after the fluxing and soldering process is complete. Finally, the important thing is that it lowers the surface tension; so, it reduces the contact angle that is actually required for a good wet solder joint.

(Refer Slide Time: 28:46)

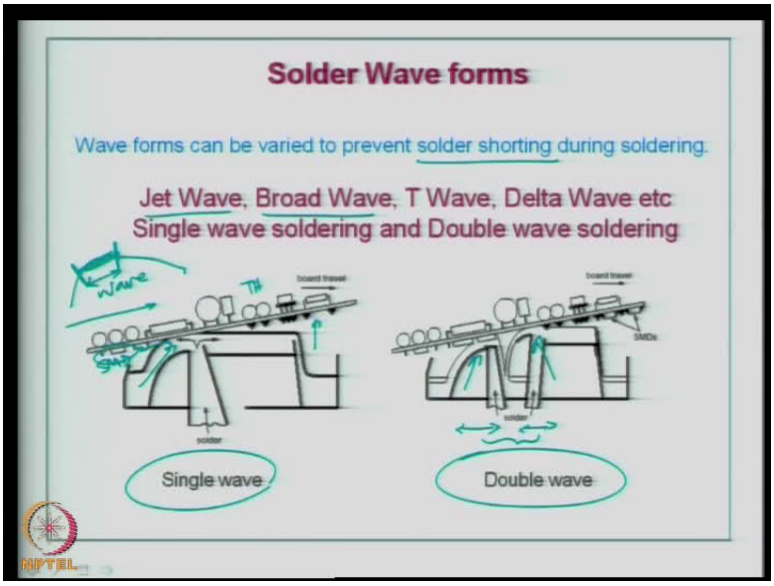


So we can understand that flux helps in improving the reliability of the board. The flux should be active enough but it should not be corrosive. Certain flux materials, if wrongly chosen can create problems; in the sense that flux material itself can introduce oxide materials or by-products of fluxing process, which if it remains on the surface of the board after soldering and which if it is not cleaned or taken care of, will create problems with corrosion. That means they can act as a very good nodal points and if they are present between two small tracks, let us say, if there are products oxides during its operation or during the life time of the product, this can act as a very good bridge between the two tracks, creating an electro chemical cell and therefore, this can end up in a short.

Therefore, the material has to be carefully chosen. Because as you see here, there are different materials available. What is the actual material used in a flux? It is called Rosin, it is to be differentiated from word resin. Rosin in itself, is naturally available resin; like your epoxy resin, this is an organic resin. It is available as an ingredient in all flux formulations.

What are the types of flux? Firstly, you have R flux which called Rosin flux, basically containing abietic acid. RA flux contains rosin activated flux that means there is an activator used. RMA flux stands for mildly activated flux. RSA flux stands for super activated flux. That means, the ingredients, apart from the abietic acid are different in each of these varieties. Then, you have the water soluble flux; and then you have a no clean flux. This development, today, in the materials concerning solder and flux activities that you have what is known has no clean flux, which means when you use a no clean flux for the soldering process you can be sure that even if you do not clean the board, you will not find any residues. So, the formulation is such that the by-products after using this no clean flux will not leave any residues on the board; so, basically the activation is done by adding some inorganic salts, amines so on.

(Refer Slide Time: 31:41)



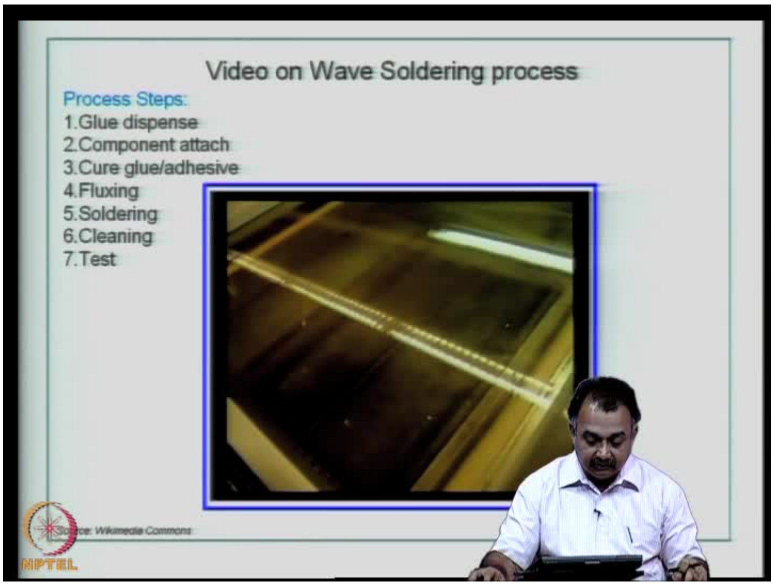
Now, let us a come to the equipment aspects of wave soldering. We have seen the process step for a wave soldering. We have seen process steps briefly for a reflow soldering process. We also looked at wave soldering equipment. If you want do plated through-hole technology, even today, in some cases, it is being done using wave soldering. So, wave forms can be varied to prevent solder shorting during the soldering process. Solder shorting or solder bridging is a common defect that we will see in spite of the fact that the surface tension of the solder and the pitch of the component is a key concern in removing these kinds of defects.

What are the kinds of solder wave forms that you can see in equipment? It can be a jet wave, a broad wave, which means the wave in itself the contact point here. The area available for the wave to touch the through-hole component on the surface mount device is of great concern. It affects the reliability and the assembly therefore, you can in some equipment adjust the width of the wave so you can have broad wave, T wave, Delta wave and so on. These are different presentations available in different equipment. Generally, if you look at the classification you can have a single wave which means you can see here, there is the single wave and this is the board that travels like this and you can see a mixture of through-hole components and you can also see surface mount devices. At one shot, you can see the pins wicking the solder through the solder wave and also the surface mount devices getting attached.

You can see at this point, the soldering process is complete. This is the wave. The same thing can be changed to ensure that in one pass of your board passing through the equipment, you can establish better reliability. Because in this case, there could be a chance that some of the components are not soldered in the first pass itself and therefore, you can have two different waves so to make sure that the components are completely attached.

Now, there are other issues that are concerned with this like what should be the width of the first wave, what should be the width of the second wave, and what is the time difference between the first wave and the second wave and so on. So, these are equipment considerations that one need, to really adjust considering the board density, number of components, the type of components, and so on including the thermal short that can be taken by individual components when it is passed through two waves and so on.

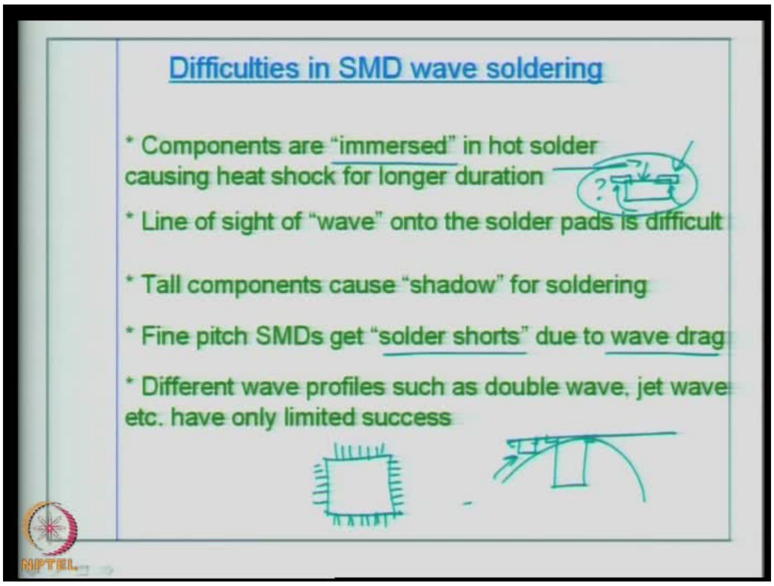
(Refer Slide Time: 35:06)



Now, we look at brief video clip on a wave soldering process. Once again I want to make sure that you understand the process steps for wave soldering. First thing is, if you are doing a complete 100 percent wave soldering process, let us say for surface mount devices including glue dispense, attach the component, cure the glue or the adhesive completely then you introduce flux that can be present in the wave soldering equipment itself. You can have a foaming flux or a simple dip flux and so on and then you introduce it in to the soldering wave after that it is cleaned and tested.

Have a look at this video, where you can see the board enters through the wave. You can see the shiny wave; and then the board travels through the wave. Sometimes, it is a double wave to ensure complete attachment in one pass. You can see the video once again; the time taken for the board travel through the wave is determined by the components density, the Tg of the substrate, the quality of the wave, height of the wave, the width of the wave, and so on. So, it is a demanding process. It requires careful consideration of all the process parameters.

(Refer Slide Time: 36:50)

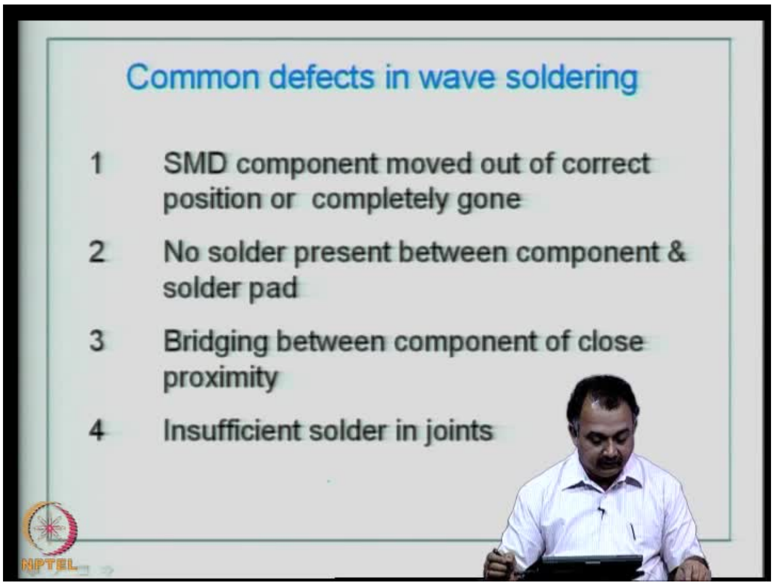


Now, what are the difficulties in SMD wave soldering? We are assuming that in some cases, wave soldering will be used for surface mount devices. If you are using wave soldering process, for such a scenario, let us look at the possible difficulties. Components are immersed in hot solder. So, that is a fact. Therefore, we have to make sure that the components do not experience heat short for a longer duration. Line of sight of wave onto the solder pads is difficult because the pads are at different ends and there is the body of the component. For example, this is the pad and this is the pad on which the component is attached through the glue here. Now, the wave has to the picked at the copper surface only then you can attach through surface tension on the mountain wave.

So, the line of sight is difficult. Therefore, you can experience for example, if the board is moving this way, a very good attachment here but probably poor attachment on one side. So, how do you take care of that? Tall components cause a shadow for soldering. For example, in a board if you have a very tall component and then if you have a small component, how do you take care of attaching this component because you have to adjust the wave and make sure that the wave is dragged on to the copper pads. Fine pitch surface mount devices get solder shorting due to wave drag. So, here again, as I mentioned, the height of the wave and the nature of the wave has to be carefully understood if you are using fine pitch surface mount devices. For example, if you want to use a QFP with fine pitch and if you have decided to use surface mount soldering then how do you take care of solder bridging not being present after wave?

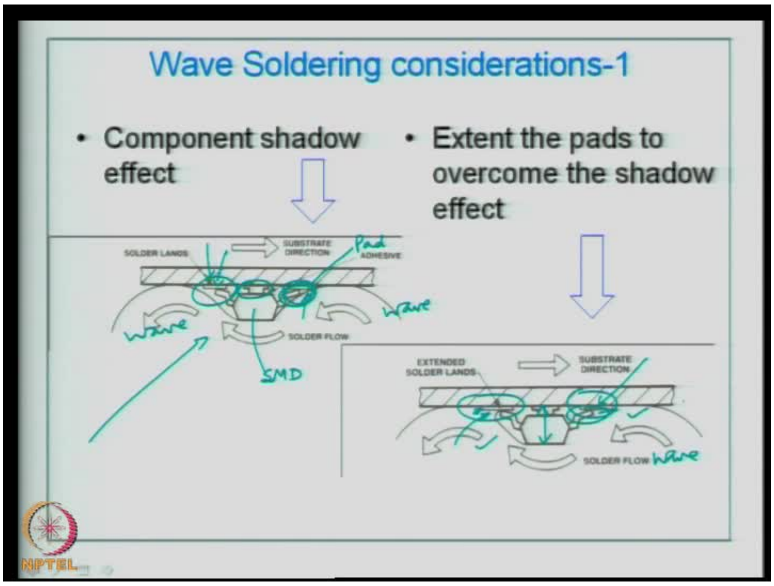
Different wave profiles such has double wave, jet wave etcetera have only limited success. These are some of points that probably ensure in considering reflow soldering as a process for surface mount device rather than wave soldering.

(Refer Slide Time: 39:33)



Common defects in wave soldering, as a summary, SMD component is moved out of correct position or completely gone because if your glue is not holding the component firmly, because of the light nature of the surface mount devices and because of the molten wave. Some times, it could be just plucked out or dismantled from its position during the wave soldering process. The second point is, there is no solder present between the component and the solder pad. Bridging between the components of close proximity, if it is the fine pitch component and insufficient solder in joints. These are some of the defects.

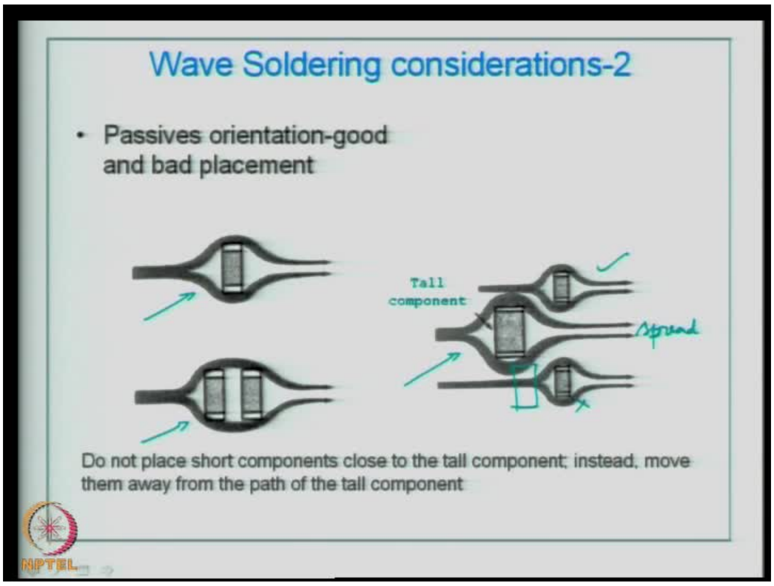
(Refer Slide Time: 40:19)



If you look at the defects that we have discussed pictorially, what is known as component shadow effect? What is it? If you look at the picture here, there is a surface mount device here - this is the SMD device; you can see here the pad on the substrate; then, there is glue that holds the device; and here you can see the wave. This is the wave of this wave soldering machine. Now, when the wave touches the substrate, here it touches the copper pad and due to surface tension, it pulls the solder and establishes a very good solder joint. But here, you look at the height of the component. The wave is not in a position to be reached by this copper pad here. Therefore, there could be cases, where this will end up with a no joint - no soldering the joint, because it is away from the line of sight of the wave. How do you overcome this? Instead of having a very small pad here, you extend the pad length during your design; so, that is why I keep saying that designers, if they can understand the manufacturing process, it can end up in very good design that could yield during assembly too. But in many cases, the designer probably are not aware of these kinds of design issues for manufacturing and therefore, it comes for some kind of a repair or some kind of a manual assembly even though the board has gone through automated process.

So, if you increase or extend the solder land like this, now, this is the wave, you can see that there is a good opportunity even though this is a tall component you can see the copper land drags the wave on to itself and establishes a very good solder joint and here again because this pad length is larger, due to surface tension, the wave is pulled on to entire area of the copper pad and establishes very good solder joint. So, these are very small design issues that can be considered by designer keeping assembly and manufacturing in mind.

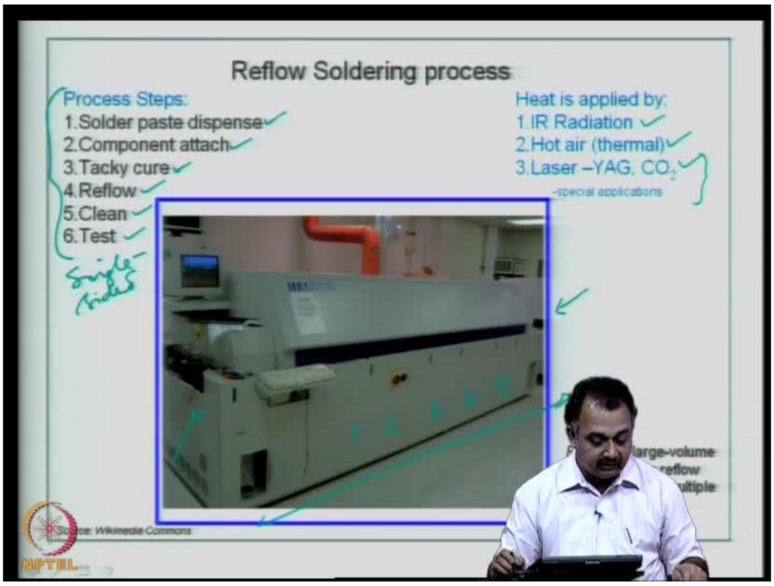
(Refer Slide Time: 43:01)



The other important thing, when you are using small components and wave soldering is that the passives orientation has to be carefully considered. What is known as a good and bad placement? For example, if this is the orientation of the resistor or the capacitor; this is the single component, now, if there are two components, then you are probably going to experience difficulty in good addition of the solder wave in the second component. So, there could be chances if the component is having a large profile or if the wave width is not ideal enough for an attachment. So, when you place resistor in series, you have to be careful.

The prospect of attachment of the resistor or the capacitor when a tall component is used also has to be carefully considered. Do not keep the tall components in line with your surface mount devices because as I mentioned briefly, this could block the line of sight of the wave; and you can expect poor soldering here. So, spread the tall component away from the smaller components like this (Refer Slide Time: 44:27) to achieve very good solder attachment. Do not place short components close to the tall component; instead, move them away from the path of the tall component.

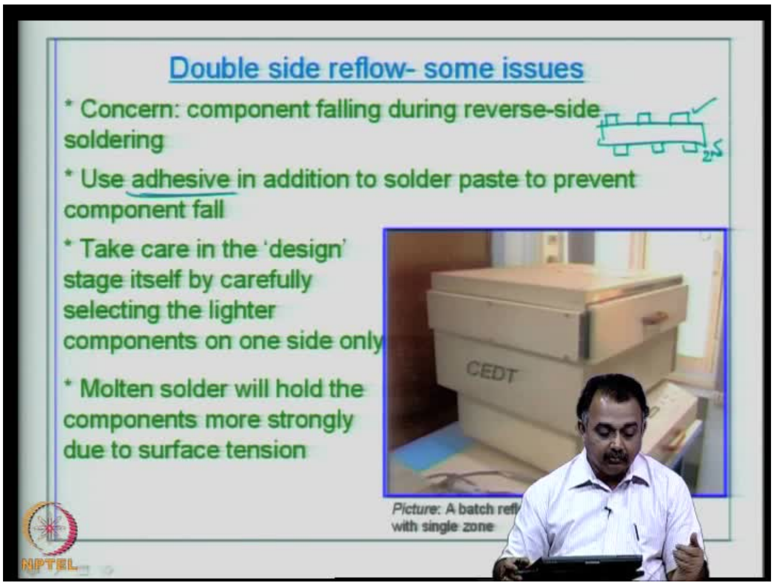
(Refer Slide Time: 44:39)



What are the methods of doing reflow soldering? We have seen that heat in a reflow soldering process is applied by infrared radiation. Equipments are available for reflow soldering with IR with thermal air - convection air, and third one is laser using YAG Yttrium aluminum garnet or carbon dioxide. So, these are, typically, for special applications. The picture that you are seeing here is a large volume manufacturing reflow equipment with multiple zones. Typically, it is fairly large equipment used for volume manufacturing. You can see many zones. You can have five different zones of a process before the board comes out. The board is fed on one side and comes out through the other side, after it is totally cooled to room temperature. In between, you will see the equipment working in different zones of heating. We will see in a short while, the thermal profile that has to be set for a reflow process and how you take care of thermal shorts.

Once again, I would like to emphasize the process steps for a reflow soldering. For surface mount devices, displace the solder; dispense the solder paste; attach the component through the pick and place equipment; then do a tacky cure, here it is not fully cured; then you do the reflow process; clean; and test. That will complete the reflow soldering process.

(Refer Slide Time: 46:41)

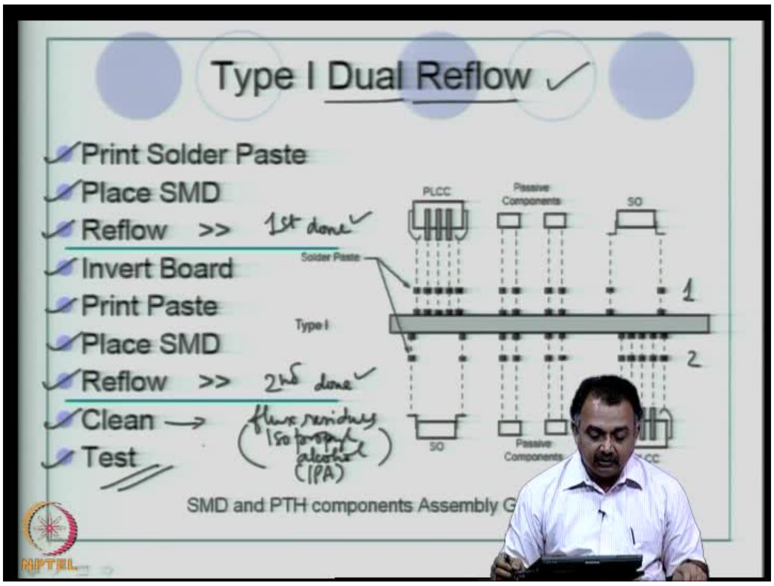


What I have mentioned are typically process steps for a single sided of assembly - very simple; there are no major issues; only thing that you have to make sure is that the temperatures are set; but when you do a SMD assembly on the Printed Circuit Board with SMD components on both sides, then you have some issues, which if you take care, will give you higher yields.

The first concern is the component falling during the reverse sides soldering. If you take a Printed Circuit Board, you have the SMD assembly; here you have done the reflow on one side and then you going to proceed for the reflow on the second side, so what are the issues? You have to make sure that the first assembly done should not be disturbed and since you are using only a tacky cure, you have to make sure that the components are not disturbed from its coordinates during the second side reflow soldering process. If there is a concern, because it is a reflow, even the components that are attached on the first side, there could be a possibility that the reflow will take place and your solder paste will reflow and then the components can move from its coordinates and then if it is a very light component, it can even fall. So, to take care of that, on one side, you can use adhesives in addition to the solder paste. You can use adhesive glue to make sure that the component does not fall during the second side assembly process.

You have to take care of this issue in the design stage itself, by carefully selecting the lighter component on one side only. Otherwise, if you have a mix of components then ensure that the assembly agency or the company, or if you are doing it in-house that you have used glue to make sure the lighter components do not fall. The molten solder will hold the components more strongly due to surface tension. Contrary to the belief that the component will fall because the solder is reflown again, the actual fact is that the molten solder will not allow the component to fall because of surface tension. These are again reliability issues and if you want make sure that you do not have to do a rework, then you can use glue.

(Refer Slide Time: 49:13)

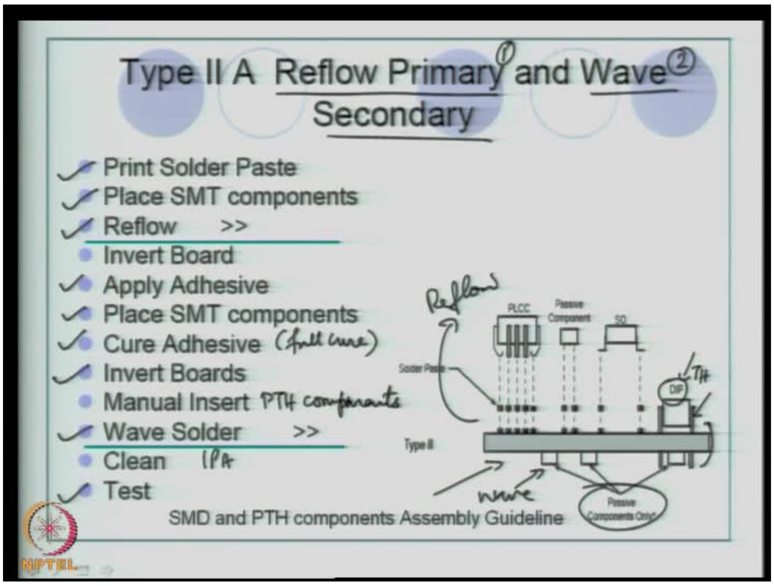


We will see different suggestive methodologies for doing reflow process. In certain cases, you will have to handle different types of components. Let us look at, first, Type I Dual Reflow. What do you mean by that? There is a reflow process that is taking place and you have to do twice. So, it is a double reflow process and as I keep emphasizing, make sure at the design stage itself whether your board material is suitable for a dual reflow. Otherwise, do not attempt dual reflow. If you have provision for having a larger board area, then you can assemble or design or do the layout for one side and end up with a single reflow. But today's boards are very complex. They are double-sided high density multilayered. So, dual reflow should really not be a problem, provided the parameters are maintained. How do you go about this?

First thing is print the solder paste on one side. Place the assembly devices using a pick and place equipment and do the reflow. The first side is complete. You can see in this picture. You have PLCC passive components and small outline devices on one side. On the other side, you have again similar components, let us say, small outline QFP passive devices, resistors, capacitors, PLCC devices and so on. Now you invert the board so that now we are turning our attention to the second side. Invert the board, print the paste again on the second side, place the SMD devices on the second side, and then you do the reflow.

The second side is also done. Second side is also reflowed. Note here we are not using any glue on the second side. We have to believe in the fact that the molten solder generally, if you maintain temperatures will not allow the first side component to again reflow and fall. Now, again there is temperature gradient between the surface of top side and the bottom side. Therefore, you are generally very safe in ensuring that the first side is not disturbed. Generally this process is well established and ideally you do not require glue for the second side.

(Refer Slide Time: 52:58)

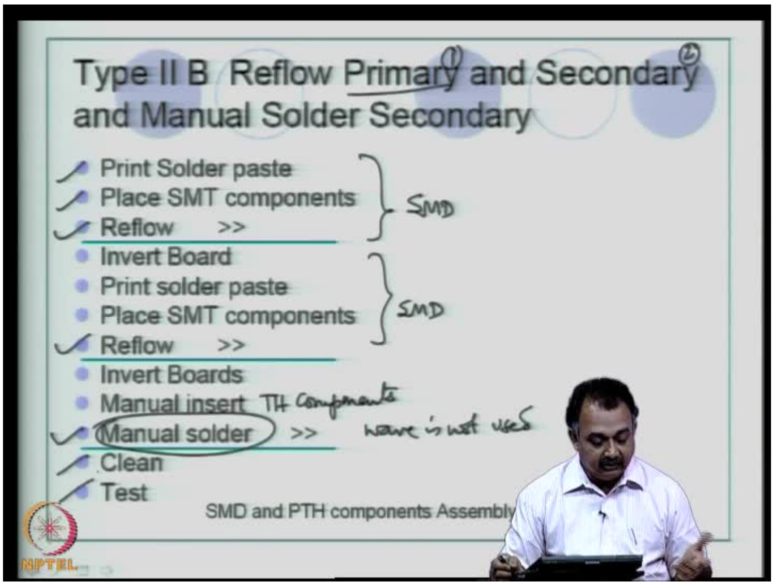


So, after that, you do a cleaning and test. Cleaning would be required to be done because you are using flux and the flux residues have to be removed. So, you can do cleaning using isopropyl alcohol or IPA as it is known. It could be diluted with water. This is because it is a mixable with water and therefore, very safe material and then, after it is dried in the oven after cleaning with isopropyl alcohol, you can do your continuity test to establish that your solder joints are prefect. This is Type I dual reflow. Type II A says that the primary process is reflow and the secondary process is wave that means here you have first process is reflow and second process is wave.

If you look at the picture here, the kind of devices that are needed for this type of soldering is that there are through-hole components here, DIP packages and so on and on the same side, that is the solder side of this, you have the SMD passive devices. On this side you can have the active devices like your small outline IC, PLCC and so on.

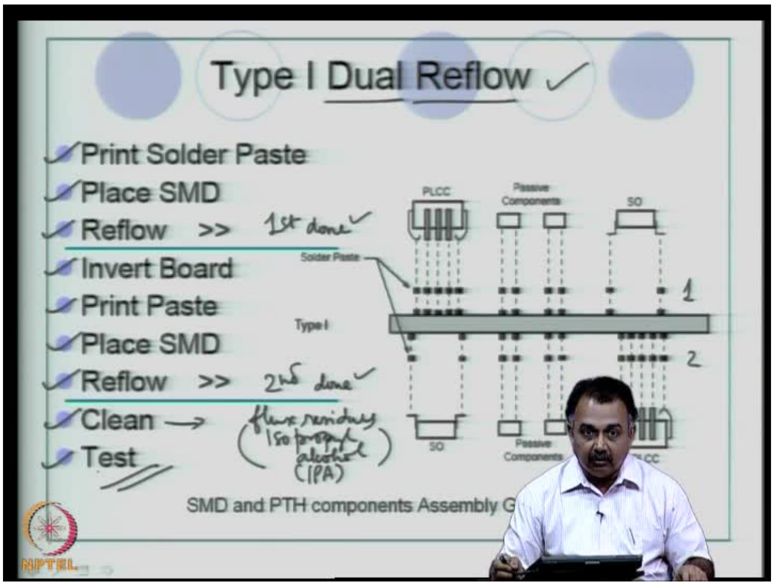
So, on the first side you print the solder paste, place the SMD devices, do a reflow then invert the board, apply the adhesive. Here, for this, you have to apply the adhesive, place the SMD device, cure the adhesive fully, full cure, now you can invert the board. The assembly components are not going to fall. Invert the board, manually insert plated through-hole components like you see here. Invert the board, manually place the packages like DIP and so on. Now, introduce this side for the wave. So, this side goes for wave and the top side goes for reflow. After that you clean and then test the board. This is how you do placement and assembly for a board which contains through-hole components, very few in number but largely surface mount device.

(Refer Slide Time: 55:01)

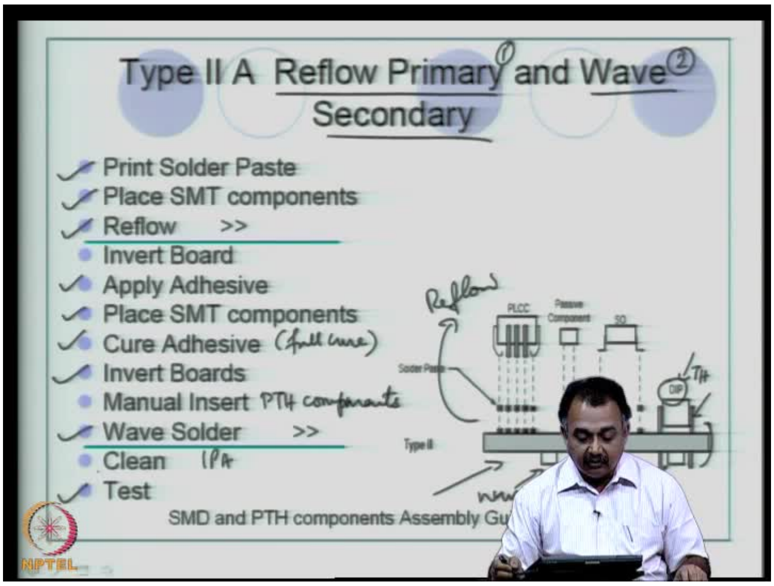


Then you have Type II B which says reflow primary is the first step and the second step is also reflow. Reflow primary and secondary. You do some part of it manual solder which is a minor component in this process. How do you go about print solder paste? Place SMT components and reflow. This takes care of all your SMD devices. Then invert the board, print solder paste, place SMT components and reflow. This again talks about SMD only. Then invert the boards, manually insert through-hole components and manual solder. Here, we are not using wave. Wave is not used that means the number of components here for through-hole type is very small. You are not adopting wave process. You do say you are doing manual soldering process and finally clean and test.

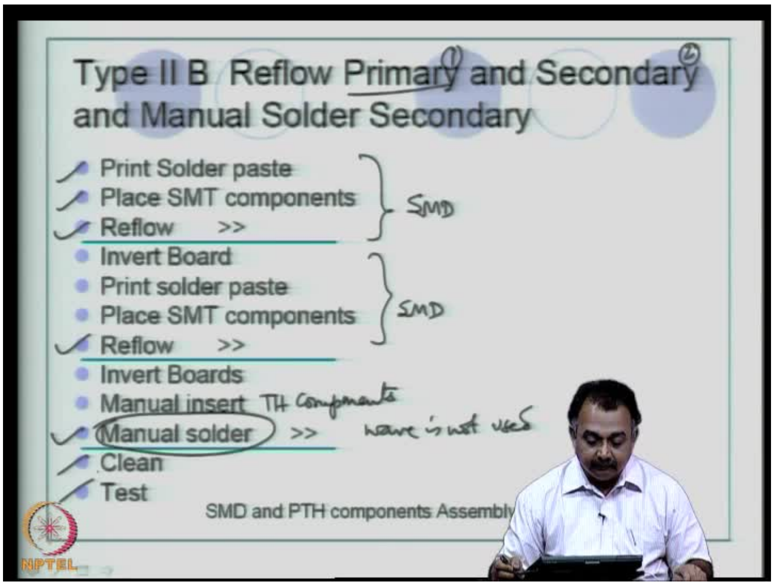
(Refer Slide Time: 56:23)



(Refer Slide Time: 56:27)

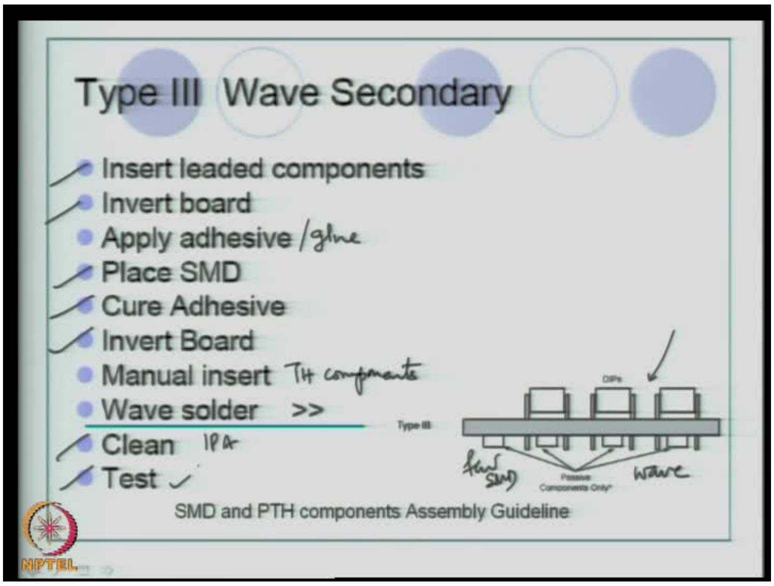


(Refer Slide Time: 56:34)



So this is known as Type II B. This basically says that kind of components in each of these, the first one you had a complete SMD, no through-hole components, the second one you had a few through-hole components that require wave soldering. In third type, you have reflow primarily and then a couple of through-hole components that can be managed with manual soldering. Even in this case, you can do manual soldering if you feel the percentage of through-hole components is much less.

(Refer Slide Time: 56:53)



Then, finally, we have Type III, which is known as wave secondary. That is, mostly it is a secondary and it is a wave soldering process. You can see in this figure here, a lot of through-hole components and a few passive devices in SMD format. Insert leaded component, then you can bend it. Then you invert the board; these leaded components will not fall. Apply the adhesive or the glue so that your SMD components can sit firmly and you cure it completely so that they are firmly held on the board. Now, you invert the board, manually insert through-hole components and then do a single wave soldering process, where all the through-hole packages and all the lighter surface mount devices are attached by wave soldering process. Finally, you clean with IPA and then test it.

So, the focus here in this is wave soldering process, where you are utilizing leaded components probably machine-based assembly and also manual insertion of heavy components and so on. But in a single wave soldering process, the entire assembly is done. These are different types. These are serving as guidelines for you in your proto typing or you have large volume manufacturing. In the next class, we will talk about another important process called vapor phase reflow soldering process.