DVD-24C
Handling in Electronics Assembly

Below is a copy of the narration for DVD-24C. The contents for this script were developed by a review group of industry experts and were based on the best available knowledge at the time of development. The narration may be helpful for translation and technical reference.

Contamination

When you're in too much of a hurry -- or when you're careless in your work -- things happen. Every time you *handle* an electronic assembly there's a chance of reducing quality and reliability.

To build an electronic assembly, components are attached to a circuit board, and are then soldered in place to complete the electrical connection. There's a lot more to making a good solder joint than meets the eye. In fact, that's exactly the problem. It's often the things that you *can't* see that cause soldering problems -- like contamination. Only the *results* are visible.

Defective solder joints are one of the primary causes of electrical failures. No matter what kind of device we're manufacturing, the last thing we want is to build a product that fails after assembly. *Cleanliness* plays a major role in the formation and reliability of each solder joint. Contamination comes from a variety of places -- some of them obvious...some a little more subtle.

In this video we're going to explore some of the *causes* of contamination...and show how it can affect the formation of a good solder joint - *and* the electrical performance of the final assembly. We'll also be explaining how proper handling can *prevent* contamination.

Handling actually begins with the supplier, or manufacturer of the electronic devices or printed circuit boards. Components or circuit boards that are not properly packaged from the supplier, or have been sitting on a shelf in the open air for an extended period of time will most likely be affected by the process of *oxidation*. *Oxidation* occurs as a result of the interaction between oxygen and other materials. Oxygen is a *very* reactive element. Whenever it comes into contact with other metals such as lands on a circuit board or component leads, it creates *oxides*. Oxidized metals can be *very* difficult to solder.

The critical issue is to receive materials with a known cleanliness and solderability -- and to *maintain* that cleanliness and solderability throughout the various assembly processes. Once the components arrive at your facility, care must be taken not to tear, rip, puncture or open the packaging until it is time to process the devices. But this is easier said than done since most inventory systems require the material handler to verify the *component count* as delivered by the supplier. This verification procedure usually violates the integrity of the packaging.

In the case of *moisture sensitive devices*, the issue becomes even more serious. Either the devices need to be stored in a dry and *inert*, or perfectly still atmosphere – which helps eliminate oxidation – or they need to go through a *baking* operation prior to assembly. Baking will remove any absorbed moisture.

Circuit boards should be stored in their sealed dry pack until the time they are needed for assembly. Some companies require a *baking* operation prior to assembly. Baking will help reduce moisture absorption, oxidation buildup and possible physical damage.

At this point, let's take a closer look at *contamination*. The *solvent extract test*, also referred to as the ionic contamination test, is designed to wash any ionic or electrically conductive contamination off a board and into a test solution. The *level* of contamination can then be measured by the *change* in the conductivity of the solution. This contamination on the surface of a board can be measured by changes in the *electrical insulation* -- or the resistance to the flow of electricity -- between two adjacent conductors. *Insulation resistance* is a measure of the ability to keep two signals or electrical currents apart.

Every time a bare hand touches the surface of a board, invisible oils and salts are left behind. And if these contaminants happen to lie *between* two conductors, the insulation resistance between them will be reduced. As the resistance value *decreases*, the electrical flow, or noise between the conductors *increases*. If an electrical pulse bleeds over to an adjacent conductor, it can affect the electrical performance of the system.

Initially, you can *see* the contamination left by a fresh fingerprint. This will eventually dry up and leave an invisible residue of salts or chlorides -- just like the fingerprints the police might take off a doorknob. As these dried salts *absorb moisture* from the air, they can become electrically conductive – causing corrosion and electrical short circuits.

Here's a real time example of the growth of *dendrites or spikes* – caused by the interaction of salts and moisture – in between two conductors of different electrical charges. A drop of water on a contaminated area provides the ideal conditions for *dendritic growth*. Normally the moisture in the air won't act this fast with the salts, but-the *potential* for failure is always there.

Clean gloves or finger cots may help eliminate a lot of contamination, but wearing cotton or nylon gloves still doesn't mean you can safely touch the surface of the boards – since oils and salts will seep through the fabric. Your facility may or may not require you to wear gloves. In either case, handling the boards *only by the edges* is the best protection against potential contamination - especially *prior* to soldering.

So far we've discussed contamination of the board *surface*. Contaminating *component leads* can be even more critical since that's where the solder connection will be made. Whenever a lead is contaminated by oils or salts, it may become *unsolderable* – or result in an unreliable solder connection.

We've seen that solderability is reduced by oxidation and the introduction of contamination. Oils and salts on our skin, food left on the hands after a meal, and skin creams can all contribute to the problem of contamination. Certain kinds of packaging materials can also cause contamination. Check with your supervisor on component handling requirements.

To fully understand solderability problems we need to discuss the role of *flux* in the soldering operation. The purpose of the flux in any of the soldering materials -- including pastes, wave soldering fluxes and cored solders -- is to *clean* the metal surfaces that are to be soldered. The more *active* the flux, the more aggressively the flux will clean the surfaces. As the flux is heated, it releases a weak acid. This acid reduces, or chemically *removes* the natural oxides that form on metals from exposure to moisture and gases in the air.

The flux forms an *encapsulant* that protects the hot metal surfaces from the atmosphere until it is displaced by the solder. In this example of wave soldering, as the solder wets the metal -- it *wicks* up the lead and inside the plated through hole. Sometimes component leads can be *so* contaminated from exposure to the environment or from human contact -- that the flux will not completely clean the surface. In this case, the solder won't be able to metallurgically bond to the contaminated metal -- and the joint will have internal voids as we see in this microscopic cross section. This solder joint will eventually fail from the mechanical stress created by temperature changes -- as the product is turned on and off.

If cleaning is *required*, it is important to clean the assembly as soon after soldering as possible to remove flux contamination. A good rule of thumb is within four hours of the soldering operation. That's because the flux residue will eventually cause corrosion on the assembly.

Other Handling Issues

Contamination isn't the *only* cause of unacceptable solder joints. *Stacking* boards without non-contaminating *slip sheets* is another way to degrade the solderability of a board. Even with a solder mask coating on the boards -- the land areas are left uncovered. These exposed lands can scrape against each other under the weight of the stack.

The problem with scratches -- even tiny scratches you can barely see -- is that *underneath* the shiny tin lead coating lies another metallic component that is not easily soldered once it's exposed to air. We call this material the *intermetallic* compound. This *intermetallic* is created when copper and tin are diffused together during the hot air leveling operation – which is one of the final steps in the circuit board manufacturing process. A human hair is about six times thicker than the solder that covers this intermetallic layer. Once this thin protective cover is removed or scratched off, the exposed intermetallic layer will oxidize and a dewetted solder joint will usually result. A dewetted solder joint may function for a while, but any amount of power or temperature cycling can cause a crack to form and grow until a failure occurs.

The best way to avoid scratches is to keep the boards from touching one another. Conductive bins or tote boxes are the preferred method for storing and transporting both bare boards and assemblies. If there are no *lids* for the totes, the boards or assemblies should be placed in *a static shielding bag* – which provides protection against contamination and electrostatic discharge when kept properly closed.

Assemblies are even more likely to be damaged by improper handling during various manufacturing operations. Several operations that contribute to missing, broken off and damaged components include stacking of boards; inserting items such as press fit connectors and other hardware; and the use of ladder style racks. Ladder style racks can cause damage to fragile surface mount components near the edges of boards. Carelessness during these operations can damage or cause broken off components. The same care needs to be taken into account during testing of boards. Some Automated testers use fixtures. Care must be taken during placement into these fixtures. When the fixture is closed, and the board is not properly positioned into the fixture, components or the board itself can be damaged. Let's take a closer look at how stacking boards can damage assemblies. *Through hole component leads* can drag across an adjacent board and scratch or damage a conductor.

But the major problem with stacking assemblies is the potential for damage to components. Since surface mount components are soldered directly to the land patterns, they are generally more susceptible to damage than through hole components. Components can also be damaged if you accidentally scrape or bump them when working on the assembly. Another type of damage can occur if you pick up the assembly by a connector or a component.

We've seen how *mishandling boards* can physically damage them, but they can also be *electrically* damaged. *Another* critical handling issue in electronics assembly is controlling *electrostatic discharge*, or ESD. ESD occurs when static electricity from your body comes in contact with and is discharged into an electronic component. This electricity can overload and degrade or destroy the microscopic circuitry of certain types of components.

There are a wide variety of materials and techniques to eliminate the buildup and discharge of static electricity. IPC has a detailed DVD on ESD prevention. The general principles, however, are easy to remember. Always be sure that your body, your workplace and the workpiece are protected from static discharge by properly grounding the workplace with grounding wires, and making sure that you are properly grounded by wearing ESD wrist straps or ESD footwear. Having yourself and the workplace electrically *tied to ground* ensures that any built up electrical charges are conducted harmlessly to the ground.

Another potential cause of ESD damage is transporting *unprotected* assemblies on an *ungrounded* cart. The metal cart may develop an electrical charge as it rolls across the floor, and then transfer that charge to the ESD sensitive assemblies. This will likely lead to a discharge when a grounded person touches the assembly. Your company will explain its policy for *transporting* assemblies. Again, conductive totes with lids and *closed* static shielding bags are proven methods for controlling ESD problems during transport.

The last area we'll be examining is *component packaging and handling*. If your company utilizes a *no-clean process*, where the fluxes are less active and there is *no* cleaning operation, it is even more important to procure components with *known good solderability*. Likewise, it is more important to handle these components and assemblies *extra* carefully as well.

The majority of components are packaged in one of three ways -- tape and reel; stick or rail feeders; and waffle packs or matrix trays. Components can also be delivered in *bulk* packaging. Generally, the *tape and reel* devices are rugged with respect to their packaging. If the reel is dropped, the shock will be absorbed by the reel and tape. Damage to the components would be minimal. One drawback to tape and reel is the *punched paper* that holds the components. This material is a source of *dust particles*. And dust is one of the worst enemies of component placement machines. That’s because placement machines require speed and precision. Excessive dust can cause them to malfunction.

Components are also available in *stick or rail feeders*. These feeders may be frustrating to constantly replenish, but they don't add significant problems to handling. They are available in antistatic and conductive materials -- and can be reused. The antistatic rails should be retreated with an approved static prevention spray prior to reuse. Your facility will have preventive maintenance schedules and guidelines for using this spray.

The last type of component packaging we'll examine are *waffle packs, or matrix trays*. These types of trays must be handled carefully when replenishing the tray holders inside the component placement equipment. The trays have no protective cover once they are opened – meaning the components will be exposed to the atmosphere. This can be an issue when *tray feeders* are used during the *staging* of components; or when the components sit for longer periods of time and may absorb moisture.

Speaking of moisture -- care should be taken regarding plastic and other moisture sensitive devices. When moisture is absorbed from the air, a catastrophic mechanical failure called the "popcorning effect" can occur. *Popcorning* describes the abrupt vaporization, or eruption of moisture trapped in a package or component during a reflow soldering process. Therefore, exposure to moisture from the atmosphere should be minimized. Keeping components in their desiccated packaging until it’s time for placement will ensure the functionality of these kinds of components.

Another major issue for surface mount components is *lead skewing* and *coplanarity*. This is primarily true for *gull wing* components, especially as the lead *pitch* gets smaller and smaller. The key is to handle these devices as *little* as possible. Fine pitch gull wing components should be handled with vacuum pick-ups if possible –versus by hand or using tweezers.

When the leads are *slightly* askew, the placement machine may not detect this. In more severe cases, some of the leads may miss the land and cause an open or short circuit. Through hole components such as DIPs and pin grid array sockets are also subject to lead skew -- which affects *in-line planarity*. If the leads don't line up precisely with the holes on the board, the part can be cracked by pressure from the automatic inserter -- as it attempts to insert the component into the board.

It's also critical to carefully handle components that require *hand insertion*. Mishandling of the leads can create *mechanical stresses* in the component when the leads are forced into the holes. It is the misshapen lead in the hole that puts stress on the body of the component. When the component is properly handled by the body, the leads will drop into the holes with gravity and won't need to be forced.

Here is a collection of useful tips for successful handling, storing and transporting of electronic assemblies. Always handle and support the assembly using *both hands*. Assemblies should be handled by the *edges only*. It's important to *never* use the *components* or *connectors* to pick up or carry the assembly. Be careful not to scrape or bump the assembly when working on it, or when storing it. And make sure you don't stack boards or trays directly on top of each other. Always use *adequate spacing* when placing product on racks, or on shelves.

Many of these considerations *also* apply when handling assemblies and other devices during *final system assembly*, or box build. Always take special care when twisting, wedging, pressing and sliding assemblies into the chassis.

Improper handling of electronic assemblies and components is one of the leading causes of defective products. But the important thing is to know the *exact cause* of each defect -- because if you know what *caused* the problem, then you can prevent it from recurring. Whenever a defect is simply chalked up to "handling errors" – that's the *worst* type of problem – since all of the boards and components could have been handled the same way. And what that really means is that the reliability of *all* of our products would then be questionable.

Finally, it's important to keep your work area clean and neat. The only items on your work surface should be those that you need to do your job. Your workstation should always be kept organized – with the surface free of debris – especially clipped leads and solder drippings. Particles and debris are *board contaminants*. Good housekeeping habits also assist in ESD control.

Defects that are caused by improper handling won't be eliminated by simply *understanding* the causes and effects of contamination and component damage. It takes more than knowledge alone. It takes the *willingness* and *discipline* to continually make the effort to handle components and circuit boards properly. It's up to you to put that knowledge to work.