



Conformal Coating Defects: When Things Go Horribly Wrong

By Pierce Pillon, Senior Field Engineer
Webinar originally presented Sept. 2020.

Unless otherwise credited, the following images were originally presented for the Minneapolis Chapter of SMTA, May 2014, and are used with permission from SMTA.



One of the most common customer support calls we receive is to solve conformal coating application issues. Given the number of variables involved in a conformal coating process (e.g. coating formula, viscosity, substrate variations, temperature, air mix, contamination, evaporation, humidity, etc.), no wonder issues come up frequently. Let's take a look at some of the more common problems that can crop up when applying and curing conformal coatings, the potential causes, and solutions.

This paper is not meant to be an all-inclusive, in-depth discussion of the conditions that may be deemed a coating defect. What I'm covering is a broad overview, intended to familiarize you with establishing and refining your coating program by recognizing these conditions.

How to Define a Conformal Coating Defect & Inspection Criteria?

First, you have to define what a defect is. This can be determined in three broad manners:

1. **Internal documentation** - Your company will generate these through your engineering and operation staff.
2. **Customer driven** - This could include internal or external customers. For example, let's say your department has been tasked to provide an assembly for another division of your company. That's an internal customer. External customer driven requirements are often determined by contracts, along with referenced documents.
3. **Industry standards** - These may include one of the more commonly used standards in electronics manufacturing: IPC-A-610G, Acceptability of Electronic Assemblies and IPC J-Std 001, Requirements for Soldered Electrical and Electronic Assemblies.

I use IPC A-610G as the prime set of examples throughout this paper.

From those resources, you next have to establish your criteria for identifying conformal coating defects. You can start with the following:

1. **Target properties** - Essentially a wish list of specifications and capabilities for the perfect coating on your assembly.
2. **Acceptance criteria** - They may not meet your targets, but these are what are considered acceptable.
3. **Inspection criteria** - These are the methodologies you use in your quality process.

Defect vs. Process Indicator

A defect is a condition that negatively impacts the form, fit, and/or function of that assembly in its service environment. On the other hand, a process indicator is a deviation from the standard that does not affect the form, fit, and/or function of that product, but indicates something in the process isn't working properly. It's important to understand that while a process indicator doesn't fully meet the acceptance criteria, it doesn't have to be considered a defect. The search for perfection can lead to an over-engineered and over-priced process.

For example, if bubbles and voids are in a populated area of the PCB, then they may be considered a defect. In an open area of the board, the same bubbles could be considered a process indicator. Is there something wrong? Yes. Your process is out of control in some manner and should be addressed. But based on IPC A-610G, it should not be considered a defect.

Curing Issues

If a coating is not cured, it's not providing the protection that it should to the underlying PCB, so it is most likely a defect. For example, many silicone coatings are RTV coatings, room temperature vulcanization polymers, and as such, they require a minimum amount of moisture to initiate that cure mechanism. If you are coating a board in Phoenix in the middle of August, and the relative humidity in the facility is essentially nil, you may get incomplete or just partial curing. According to IPC A-610G, this is a condition that should be considered a defect.

Missing Coating in Required Areas

If a coating is not applied to the required areas, it would generally be considered a defect (which is why they are considered *required*). Those required areas should be specified in the engineering drawings for that assembly.

Keep-out Areas

In contrast, you may have coatings on areas of the PCB that are required to be free of coating – keep-out areas. For whatever reason, those are required by the engineering drawings to be masked off. An example would be a connector that wasn't masked or booted, and now you have insulated coating around conductive pins or in a conductive female housing.

Entrapped Materials

Other examples of common defects include exposed conductive surfaces caused by loss of coating adhesion, voids, bubbles, dewetting, cracks, fish eyes, flaking, and any entrapped material. Entrapped material has the potential to bridge the lands or the adjacent conductive surfaces. If the contamination causes there to be less than the minimum clearance between the components, lands, or conductive surfaces, it should be considered a defect.

Discoloration

Discoloration or loss of transparency is another potential defect. For example, urethane conformal coating tends to yellow over time if exposed to sun. That could be ok, but if it hazes in any way, it could well be considered a defect. This is an important distinction because you have to be able to see through the coating to properly inspect labels, markings, etc.

Main Causes for Conformal Coating Defects

To help you segment the common causes for conformal coating defects, I will be using icons throughout this paper.

Cleaning



Cleaning (or perhaps lack of cleaning) is an important category that can be ripe for creating conditions that lead to defects. To isolate this variable, you can take a new board or assembly, clean it, coat it, and see if it corrects the defect. You may want to validate that by additional testing, but at least you've narrowed down the potential issues.

When cleaning flux from the PCB, it might be tempting to use the same solvents that are used to remove coating from your selected spray system or a spray gun. While it may be a very strong solvent, it won't necessarily be the appropriate solvent to clean flux residues. The cleaner could merely smear the residue around instead of breaking it down.

Lack of cleaning is a common cause of defects like fisheyes, dewetting, and delamination.

Application



Application problems can include improper or drifting equipment settings, wet film thickness that is too low or too high, and viscosity changes. Variability can be caused by equipment, but is more commonly caused by human error. Inconsistency of operator application, whether it's from one application to another using the same operator, or the same application method using multiple operators over a shift, can encourage defects.

Application problems can lead to defects like shadowing, orange peel, wrinkles, voids and bubbles.

Curing



The curing process includes the specified time, temperature, and in some cases, the humidity conditions in which you are curing that coating. If you're not following the coating manufacturer's recommended guidelines, it can set you up for a coating failure or a defect.

Curing issues can cause defects like cracks and ripples, wrinkles, delamination, voids and bubbles.

Common Conformal Coating Defects

Non-Uniformity

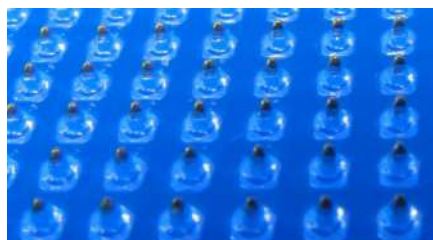


Fig 1

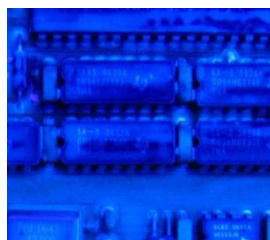


Fig 2



The coating must uniformly cover that board and all of the components. If it does not, it could be a condition that may be a defect. One common example is shadowing. Shadowing is caused by the geometry of the board, where you have a short component right next to a tall component. The method of application may prevent that shorter component from being uniformly coated because it's being blocked by the taller one.

Brushing can be a cause of uneven application. When you're using a brush to apply the coating, the tendency will be to apply more coating at the beginning of the pass and less coating at the end of the pass.

In a dipping process, you'll often see coating sag. Sagging is specifically allowed in IPC-A-610G in some cases. As the coating settles from a dipping process, it accumulates at the lowest point. Could it be a defect? Possibly, but it is allowed, as long as it does not affect the form, fit, or function of that component or that part in its service environment.

If you have sharp angles or edges like through-hole solder joints (fig 1), coating tends to be less consistent than on a flat surface of the component or on the board laminate. A through-hole solder joint is essentially an upside down cone. When coating is applied over the top, it tends to fall by gravity down to the base, so you generally have a little bit less coating coverage at the tip of the lead.

You can have differences in the surface tension of the coating liquid compared to the substrate energy, which can cause non-uniformity (fig 2). Surface tension is a method of quantifying the tendency of a liquid to wet over a surface, or spread out. Surface energy is the tendency of a surface to allow wetting. For example, water beads on a freshly waxed car because the surface has low surface energy, and water has relatively high surface tension. In general, the liquid coating surface tension must be less than the substrate energy to allow it to flow for adequate wetting, adhesion, and bonding energy to that substrate.

Dewetting

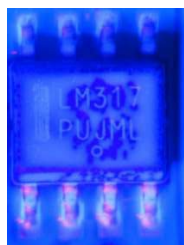


Fig 3



fig 4



fig 5



Dewetting is essentially where the coating pulls away from that surface because of differences between the surface energy. It could also be a contaminant like little spots of silicone from mold releases, adhesive residues, or in the case of fig 3, component marking ink.

Dewetting of conformal coating could also be caused by oils from your hands and fingers from handling the board. There can be some interaction with flux residues because of the binders, modifiers, and other ingredients left over after the soldering process. This can be fairly easy to identify if dewetting is more prevalent around solder joints. In the case of a wave soldered board, the soldered side might have more dewetting. To verify, clean the solder areas before coating. If dewetting persists, keep looking for other clues of the cause.

There can also be interaction with the coating component casing or PCB laminate, also called overcoat and resist (fig 5). We have had documented cases when bare boards came from two different suppliers, and one caused dewetting while the other did not.

Fish Eyes

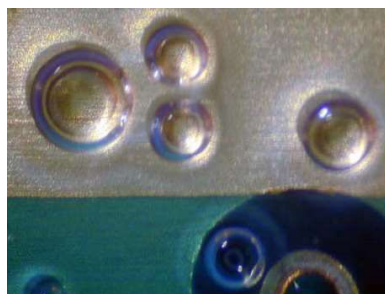


Fig 6

Fish eyes occur when coating goes over a small point contaminant, it raises up and repels from that contaminant to form something like a bubble or bulge in the coating. The contamination could be from a wide variety of sources, including:

- Spot of silicone or wax
- Residual coating from a previous rework
- Scoring dust
- Contamination coming in from your ventilation system
- ... or even a piece of lint off of your smock

Anything that can settle on that surface prior to coating, can be considered a point contaminant.

If you're using temporary solder mask in your process to protect areas that you don't want soldered, it can add potential contamination. If it's a peelable mask, once the soldering process is completed and you peel that mask off, there could be a little piece of mask that you can't even see with the eye. If you are using a water washable temporary solder mask, those are generally composed of either clay-based or cellulose-based material in a water-soluble adhesive binder. When it goes through the cleaning process, the binders dissolve, releasing either the clay or the cellulose, and that material goes down the drain (or processed in some way). If you don't adequately clean it, there can be little spots of clay or cellulose that can cause fish eyes.

Voids or Bubbles

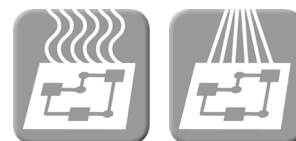


Fig 7



Fig 8

Bubbles are most often caused by solvent that becomes entrapped when it flashes off (evaporates). For whatever reason, the solvent vapor can't escape the coating, so the vapor creates a void within the coating that bulge out.

Normally, the solvent diffuses out slowly as the coating cures. When the coating skins over faster than the solvent can flash, the solvent vapor becomes trapped. Is it a defect? That depends on your criteria, according to IPC-A-610G or whatever standard you choose to use. If it's in a non-populated, blank area of the PCB, you may decide to let it go. You may have a bubble, but it's not bridging lands and adjacent components, so most likely will not affect the form, fit and function of the electronics package. It is, however, an indication that something has gone wrong in your application process, so should be addressed and resolved.

You'll commonly see bubbles at the base of leg components, where you have applied the coating over the top and around the adjacent areas. Coating will tend to creep into the stand-off areas underneath the component. As you apply heat, that coating will then cure over the lead surface at a quicker rate than in that standoff area. So the coating may skin over at the base and around the edges of that component. Then the solvents underneath flash off and have nowhere to go. The vapor will hit that partially cured coating, and because of the vapor pressure, will cause the coating to bulge and create a bubble.

Manually brushing conformal coating is another common opportunity to introduce bubbles. When you dip your brush into the coating, it can trap bubbles between the bristles. Then, when you make your pass with the brush, it releases that coating with the entrained air bubbles. Sometimes those bubbles will dissipate as the coating levels out, and sometimes they become trapped.

In the case of a spray gun application, if it has been improperly setup, or your settings have drifted, it can lead to entrapped air bubbles. If your spray passes across that board too slowly, it can entrain air as it hits the surface of the board, possibly even air that's underneath the components (as mentioned previously). If the distance between your nozzle and your substrate is too close, you might introduced turbulence into the coating that can generate and trap bubbles.

If your air pressure (push pressure), either in your selective system or, more commonly, in your spray gun is too low, it can entrain bubbles.

Cracks and Ripples



Fig 9

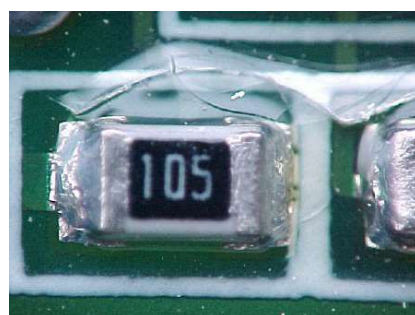


Fig 10



Excessively thick coating, especially in a high solid content coating, doesn't tend to cure evenly. It doesn't allow the solvent to evaporate evenly, especially at the surfaces. If you then get strains and stresses in that coating, it could crack. Force drying (curing at too high of a heat) can also lead to cracking. Fig. 9 and 10 illustrate what we call "mud cracks". In fig. 10, you can see the crack across just outside the top of the component. That's caused by uneven curing, which stresses and strains the coating as it tries to cure.

Delamination



Fig 11

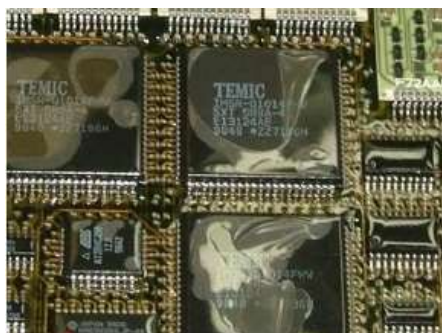


Fig 12



Delamination is not common, but it does occur. Unfortunately, you generally won't see this in your manufacturing facility, but as products come back from the field. Fig. 11 shows the coating popping up and presenting as a gap under the casing surface, while the coating is actually flaking off in sheets in fig. 12.

One of the causes can be excessive coating thickness. The surface of the coating will cure first, and with the slight sizing, adds stress between the coating and the substrate before adhesion is optimal.

If a thick coating is required, we recommend applying multiple thin coats, instead of one heavy wet coat. Multiple thin coats encourages better inter-coating cohesion than heavier coats. To ensure the coats meld together, wait until the previous coat becomes tacky before applying subsequent layers.

Various surface contaminants can prevent good bonding of the coating to the substrate. You may have a little silicone, flux residue around the leads, or it might be the solder resist itself.

If bare board resist is very slick, it is a sign of very low surface energy. As mentioned previously, the liquid coating surface tension must be less than the substrate energy to allow it to flow for adequate wetting, adhesion, and bonding energy to that substrate. When the surface energy of the PCB is lower than the coating's surface tension, it can either cause dewetting, or if it has an opportunity to cure, delamination.

Orange Peel

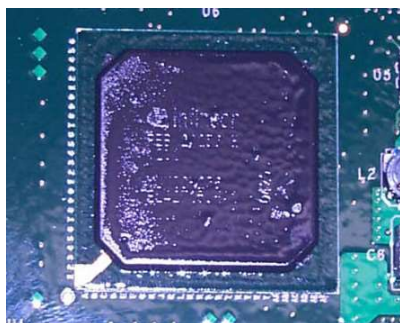


Fig 13

Orange peel is a very rough, uneven texture, similar to the texture of an orange (fig 13). Sometimes excessive film thickness and using heavy wet overcoats will cause this texture. If you have a wet coat and then apply another coat on top of it, the spray pressure can push it. In fig. 13, the coating on the left side looks thinner, like it's been pushed to the far right hand side.

If your substrate is too warm relative to the temperature of the coating, it can also cause an orange peel texture. For example, if you've just cleaned your board and it has gone through the dry cycle in your wash system, it will be warm when you first take it out. If your operator is in too much of a hurry, they may spray cooler coating over the



surface. When the coating hits that warm substrate, the temperature difference will start the solvents quickly and unevenly flashing off, creating a textured surface.

The brand and type of coating can also affect how it levels. When a manufacturer develops a conformal coating, they choose solvents that evaporate at different, yet “staged” rates. Most formulators will include what they call a “tailing solvent”. It is generally the smallest amount of solvent, yet it is the slowest evaporating one. That allows that coating to remain wet in the final stages of curing, so it levels out to that nice, smooth, even finish.

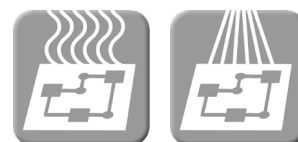
Orange peel texturing can also be caused by spraying with pressure that is too low, especially when using spray guns. The droplets of coating are too course or too far apart to join and smooth out before curing begins. This can sometimes cause that text book orange peel appearance.

If you are spraying in multiple passes, each must be parallel to each other, with consistent, slight overlaps. This provides the best opportunity for the coating to level out consistently and smoothly. You should also have the spray nozzle angled so the output is perpendicular to the substrate, not at an extreme angle. Otherwise, you are putting pressure behind that coating and pushing it away from the point of contact.

Wrinkles and Waves

Wrinkles and waves are cousins of the orange peel texture, so the root causes from application method are often the same. In addition, if you're curing in an IR conveyor system that is set too high, it can cause the tailing solvent to flash too quickly. The coating will not have an opportunity level properly and can result in wrinkles.

If you are spraying multiple coats, and you spray your second coat when the previous coat is still wet, it can cause a wave texture. As mentioned earlier, it is important to allow the first coat to become tacky before adding subsequent layers.



Summary: The Perfect Coat the 1st Time, Every Time

As mentioned previously, coating defects will generally have one of the following root causes:



- **Cleaning** – lack of a clean surface commonly causes fisheyes, dewetting, and delamination because it interferes with adhesion or interacts with the coating.

The cleaner the PCB is, the better chance you're going to have of good bonding energy, good adhesion, good coverage, and less potential defects. In addition, flux residues can interact with moisture and be a cause of either for dendrites (ionic branch structures that can form between contact points and cause current leakage or a short) or corrosion. This can even be an issue with no-clean fluxes, even though it is very common to coat over them. Different flux and coating combinations may react differently, so it is important to qualify every combination for final reliability.

Cleaning issues are the easiest defect cause to isolate. All you have to do is thoroughly clean the assembly before coating. If it solves the problem, you now have some idea of your next steps.



- **Application** - Application problems can lead to defects like shadowing, orange peel, wrinkles, voids and bubbles. Application problems can include improper or drifting equipment settings, wet film thickness that is too low or too high, and viscosity changes. Variability can be caused by equipment, but more common is human error.



- **Curing** - Curing issues can cause defects like cracks and ripples, wrinkles, delamination, voids and bubbles. If you're not following the coating manufacturer's recommended guidelines, it can set you up for a coating failure or a defect.

Please reach out to us with questions, problems, and to work through your qualification process. Chemtronics has expertise with both cleaning and coating, and an in-house lab with specialized equipment to help duplicate and solve any problem you run across. We also have application specialists and field engineers who have the mission to make your job easier. Contact us at 770-424-4888 or askchemtronics@chemtronics.com or go to www.chemtronics.com.

Chemtronics® is the acknowledged industry leader in solutions for the electronics, telecommunications and critical environments markets. Products are engineered to meet a full array of cleaning, protecting, repairing, and PCB prototyping needs. Products include degreasers, flux removers, conformal coatings, solder mask, desoldering braid, board repair pens, cleanroom swabs and wipes, and fiber optic cleaners. For more information, go to www.chemtronics.com.