New Power Modules Improve Surface Mount Manufacturability

By Chris Thornton (Email: cthornton@ti.com)
Plug-in Power Products

Introduction

Texas Instruments' latest generation of board-mounted power modules utilize a pin interconnect technology that improves surface mount manufacturability. These modules are produced as a double-sided surface mount (DSSMT) subassembly. This yields a case-less construction, with subcomponents located on both sides of the printed circuit board (PCB). Products produced in the DSSMT outline use the latest high-efficiency topologies and magnetic component packaging. This provides customers with a high-efficiency, ready-to-use switching power module, in a compact space-saving package. Both non-isolated "point-of-load" (POL) switching regulators and the isolated DC/DC converter modules, for use in telecom applications, are being produced in the DSSMT outline.

Power module sub-assemblies are made available in both through-hole and surface mount package options. Surface mount modules produced in the DSSMT outline use Solderball Pin™ * interconnects for their attachment to a host PCB. This attachment method is designed to be more reliable than other surface mount interconnects, which translates to improved manufacturability for customers that employ high-volume, surface mount manufacturing methods.

Component Coplanarity

In the electronics industry the term coplanarity is used to define the maximum distance that the physical contact points of a surface mount device (SMD) can be with respect to its seating plane. When placed on a flat surface, an SMD will rest on its three lowest points. This defines the seating plane of the device. The number given for coplanarity defines the maximum gap that can exist from the underside of any pin to that seating plane. This measurement is unilateral.

To provide a reliable solder joint, each pin of an SMD must make contact with the solder paste covering its respective solder pad. Solder paste is deposited on the host PCB using a solder stencil and squeegee. The thickness of the solder stencil determines the thickness of the solder deposited. The thicker the solder paste, the more likely the SMD pin will make contact with solder. During reflow, the surface tension properties of liquid solder cause the solder to wet between the pin and pad. The solder bridges any physical gap between them to form a fillet. Figure 1 shows a cross section of an acceptable solder joint. If

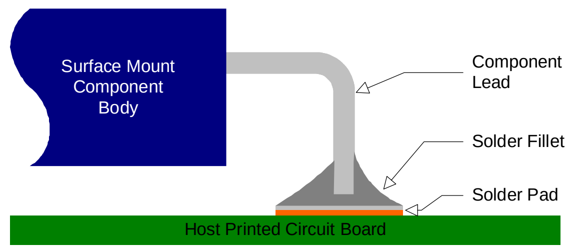

Figure 1 Acceptable SMT Solder Joint [1]

the measured coplanarity of the pins is too great, for the amount of solder deposited, some pins may not make contact with solder paste. In this situation the liquid solder simply forms a pool on the PCB pad. It does not wet to bridge the gap between the pin and pad, resulting in an electrical open circuit. Figure 2 show how an excessive gap

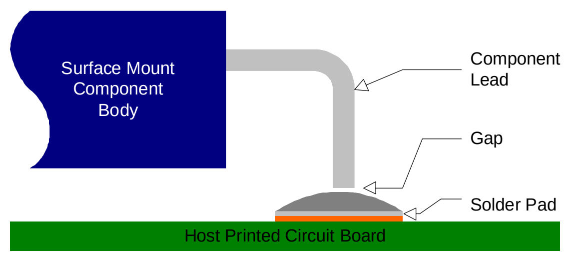

Figure 2 Excessive Gap Prevents Fillet Formation

between the component lead and solder pad prevents the formation of a solder fillet. In this case the finished assembly must either be reworked or rejected, with a corresponding impact to manufacturing yield and cost. A thicker solder stencil can be used to deposit more solder. This will accommodate parts with a higher coplanarity variance, but causes problems with smaller components that have fine lead pitches. Excessive solder to the pads of small parts can result in adjacent pins being bridged and shorted. The additional volume of solder increases the risk of solder balls being formed during reflow. Solder balls can break loose to become electrically conductive debris within the end-product's enclosure.

* Solderball Pin™ is registered trademark of Autosplice, Inc.

One commonly used thickness for a solder stencil is 0.006 in. (0,015 mm). This thickness generally provides a sufficient amount of solder to ensure the pins of the components make contact with the PCB solder paste. This dimension is consistent with the coplanarity of SMD packages that limit the maximum distance of any pin from the seating plane to no more than 0.004 in. (0,01 mm). For large complex components, such as power semiconductors, magnetic components, and power modules, the package coplanarity is often higher. These parts require a larger solder pad and a thicker layer of solder paste to ensure they are soldered.

While it is always possible to dispense more solder paste to a select few pads on the host PCB, it complicates the soldering process. A thicker solder stencil can be used, but efforts must then be made to reduce the amount of solder deposited onto the pads that do not require it. The stencil thickness can be “stepped down” or the apertures (openings) reduced. There are issues with both approaches. The main disadvantage with step-down stencils is that they are more expensive, and are impractical to implement for a few pads on a densely populated PCB. The reduced aperture option has to be applied to a large number of solder pads, and often requires trial and error to determine a workable aperture pattern. Because of the issues associated with these techniques, original equipment and contract manufacturers are reluctant to employ them. The expectation is that all components should comply to coplanarity limits that are compatible with a 0.006 in. (0,015 mm) solder stencil thickness. From the industry’s standpoint this simplifies the design of the solder stencil and minimizes the cost.

Power Module Construction

Due to their size and construction, the surface mount packages of power modules are challenged to meet the same coplanarity requirements as smaller surface mount components. Power components tend to have larger PCB footprints with thicker, longer pins, located on a wider pitch. These characteristics make it more difficult to manufacture power modules to the same coplanarity tolerances as small semiconductor ICs. It is not unusual for power module packages to specify a maximum pin distance from the seating plane of 0.006 in. (0,015 mm), or greater. Irrespective, both OEM and contract manufacturers expect power module vendors to provide SMD packages that are compatible with their standard manufacturing process.

Power modules are usually constructed from a sub-assembly PCB. The leads or pins can either be part of a lead-frame or independently attached to the PCB. Depending on the construction, the PCB may be attached to a plastic or metal case, or even covered by an exterior molding. The pins can either be solid (rolled or stamped), or flat. Their function is to elevate the module to give clearance underneath the body, and provide a foot that can be soldered onto a pad of the host PCB. Flat pins may be used on modules that employ a case or heat sink. These must subsequently be cropped and formed to a specific shape, which adds cost.

As the industry has pushed toward higher levels of integration, power modules produced in the DSSMT (double-sided surface mount) outline have been well received. The case-less construction allows components to be placed on both sides of the module’s PCB (see Figure 3). This results in a more compact module with a correspondingly higher power density. DSSMT modules traditionally use solid pins. The pins are mounted directly beneath the module to provide mechanical support as well as an electrical connection to the host PCB.

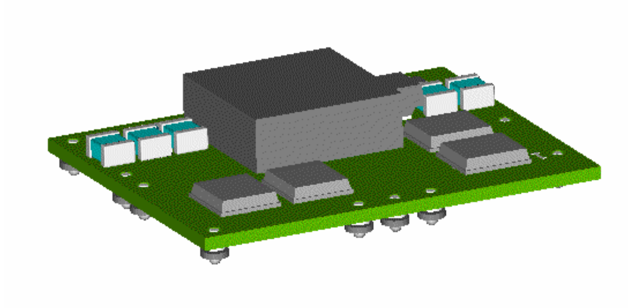

Figure 3 DSSMT Power Module Assembly

Factors Affecting DSSMT Module Coplanarity

There are three principal factors that affect the coplanarity of DSSMT power modules. They include:

- Dimensional variations in pin length
- Warping of the module PCB
- Soldering imperfections on the module PCB

DSSMT power modules use a solid pin as an interconnect to the host PCB. Compared to flat pins, which must be cropped and formed, solid pins are relatively thick and short. This makes them more robust, and less susceptible to misalignment through handling. They are manufactured to a pre-defined length using modern machine tools; a process that results in a consistent product with tight tolerance limits.

The DSSMT package outline places the module PCB in control of the mechanical integrity of the package. This includes the physical alignment of its pins; both lateral and axial. The PCB material is a laminate, and subject to manufacturing variations, including warping. Over the dimensions of a semiconductor IC or even a large discrete component, the effects of warping are negligible. Power modules, on the other hand, can measure up to three inches

(75 mm) along one side. Warping over this distance can introduce a significant amount of coplanarity variance.

Solderball Pins™

The DSSMT power modules produced by Texas Instruments use Solderball Pin interconnects. These are solid copper pins that incorporate a solder ball on the end that interfaces with the host PCB. The solder ball can comprise of regular (63 Sn/37 Pb) tin-lead solder, or high-temperature (96.5 Sn / 3.5 Ag) tin-silver solder. The interconnects are designed for improved solder-reflow capability. Most significant is their ability to automatically compensate for coplanarity differences between the module and the host PCB.

Figure 4 shows a solid image of a single Solderball Pin. It comprises of a high conductivity solid copper pin, which incorporates an integral contact or shoulder. The top of the pin is formed into a barrel, which is designed to centralize the pin within a plated-through hole on the module PCB. The shoulder provides a contact area for the copper landing pad on the underside of the module. The lower part of the pin extends through a small fiber washer into a ball of solder. When the module is placed on the host PCB, the solder ball end of the pins are placed into the same thickness of solder paste as other components. The module is then processed in a normal reflow operation.

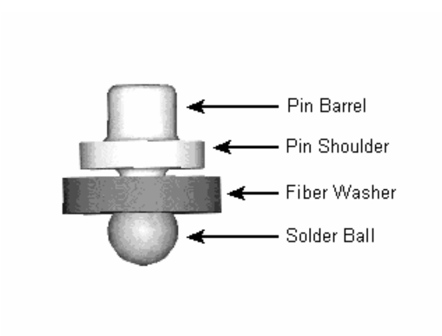

Figure 4 Solid Image of Solderball Pin

The purpose of the fiber washer is to prevent liquid solder from wicking up the pin. The washer retains the solder around the butt joint formed between the module pin, and the pad on the host PCB. This ensures that there is sufficient solder to form a solder fillet when standard paste levels are used.

Coplanarity Compensation

The solder ball adds two important attributes that allow a higher coplanarity variance to exist between the module and host PCB. It provides an additional source of solder, and also allows the sub-assembly to drop slightly when the solder becomes liquid during reflow. This drop occurs when the weight of the module overcomes the buoyancy of the molten solder. It is made possible by the fact that the bottom end of the pin extends only part way into the solder ball. (See Figure 5). The amount of drop is equal to the distance that the solder ball extends beyond the end of the pin (within the ball). This is the dimension 'D_2' in Figure 5, and is known as the coplanarity compensation zone [2].

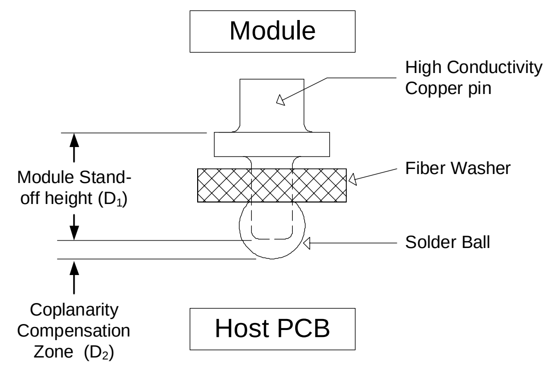

Figure 5 Detailed Outline of Solderball Pin

The dimension, D_2, corresponds to the additional amount of coplanarity adjustment. The extent of this adjustment is such that the solder ball end of the pin does not have to make physical contact with the solder paste prior to reflow. The seating plane of the module need only bring each solder ball to within a distance of less than D_2 of the solder paste. Figure 6 demonstrates how the coplanarity compensation zone works when a DSSMT module is passed through a standard solder reflow process. For the purposes of this illustration the coplanarity variance between both the module and host PCB has been exaggerated.

Prior to reflow only the solder balls of the two outer pins are shown making contact with the solder paste on the host PCB. The solder ball of the center pin is raised above the paste due to the warping of the boards. The gap underneath this pin is not a problem as long as it is less than the distance D_2. This is the distance that the lowest pins are raised above the host PCB pads by the solder balls. During reflow, the solder balls become liquid, allowing the module to drop by this distance. The solder ball of the center pin is then able to make contact with solder paste. Once contact is made the solder from the ball and paste coalesce to form a solder fillet. A typical module may have a dozen or more discrete connections, several of which could be raised off the host PCB pads. The joints of these connections would all be brought into compliance as a result of the module sinking towards the host PCB during the solder reflow process.

In the explanation of coplanarity compensation, the dimension D_2 is a key parameter. This is the thickness of

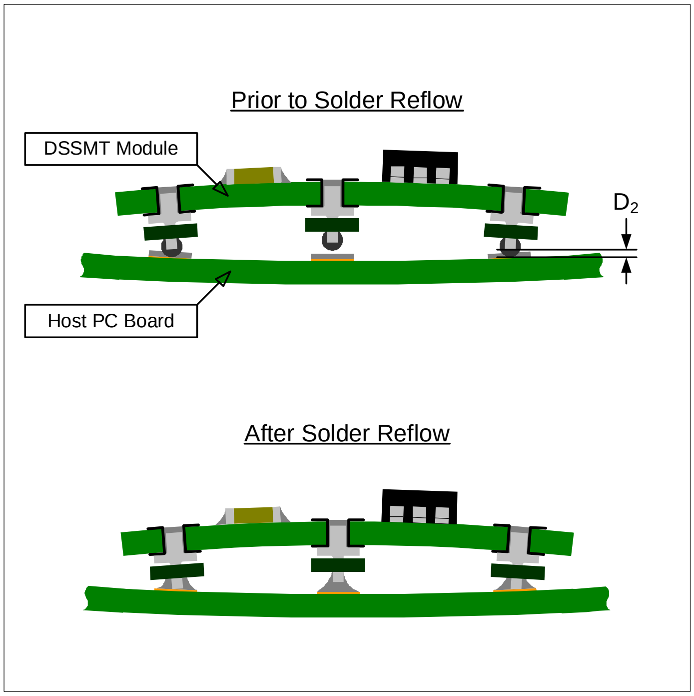

Figure 6 Behavior of Solderball Pins During Module Reflow

solder from the end of the pin to the tip of the solder ball. The pin manufacturer characterizes this dimension as 0.0127 in. (0,32 mm) nominal, with a standard deviation of σ =0.0013 in. (0,033 mm). This suggests that the manufacturer's process can easily meet a minimum of 0.008 in. (0,2 mm).

The amount of coplanarity compensation offered by the dimension D_2, adds to that provided by the thickness of the solder paste on the host PCB. The sum total of these dimensions represents the maximum gap that can exist between the end of a pin and its PCB pad in order that a fillet can be formed. If the minimum dimension for D_2 is 0.008 in. (0,2 mm), and the recommended solder paste thickness is 0.006 in. (0,15 mm), then the combination can accommodate a minimum of 0.014 in. (0,36 mm).

DSSMT Module Coplanarity Variance

The coplanarity variance of any SMD can be evaluated by measuring the distance that each contact point stands off its seating plane. In the case of DSSMT modules with Solderball Pin interconnects, the contact point (pin end) is contained within the solder ball and cannot be inspected. It is only during reflow, that the module is able to settle onto its seating plane. Inspection verification by applying heat to remove the solder would inevitably disturb the pin. This in turn would affect the coplanarity parameter. For this reason the coplanarity is best assessed by a review of the manufacturing process, along with empirical measurements on manufactured parts.

The three principal factors that affect the DSSMT module coplanarity include: The dimensional tolerance of the copper pin length, warp in the module PCB during solder reflow, and soldering imperfections of the pin/module joint. Each of these factors can be assessed for its impact on the module's coplanarity variance.

A variation in pin length has a direct affect on coplanarity. Statistical process control (SPC) data was obtained from the pin manufacturer. This gives the dimension of the copper pin, from the shoulder to the pin end (within the solder ball), as: 0.065 in. (1,65 mm). With a standard deviation, σ =0.00056 in. (0,014 mm), the variance of the pin will be: ±0.0017 in. (±0,043 mm).

To evaluate the variances due to PCB warp, samples of the module's PCB were studied under reflow temperature conditions using a "Shadow Moiré" test system. This is an optical technique that gives a precise measurement of out-of-plane displacements. This testing was conducted on PCB samples of the larger DSSMT modules produced by Texas Instruments, the PTH12030WAS. This product measures 1.37 in × 1.12 in. (34,8 mm × 28,45 mm). The deflection of the PCB was mapped using color 3-D plots, at various temperatures from 25°C up to 260°C ambient. Figure 7 shows the plot from one of the samples at 260°C. The vertical displacement is given in mils-in. The results of this testing [3] revealed that none of the PCB samples saw a

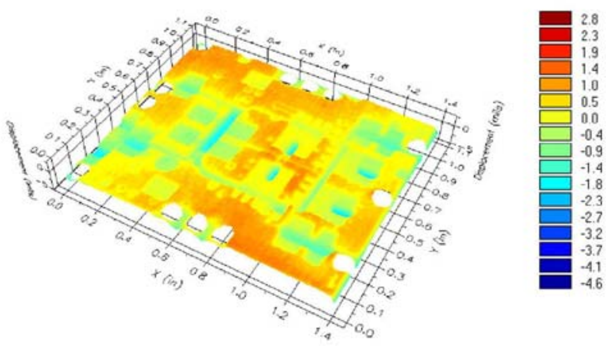

Figure 7 Shadow Moiré 3-D Plot at 260°C

deflection greater than 0.003 in. (0,1 mm), at reflow temperature. This maximum deflection was recorded in the areas around the pin landing pads, at the opposite corners of the PCB. PCB deflection was also reduced as the ambient temperature was raised from 25°C, and was lowest at reflow temperatures. This is notable as it is during reflow that the module establishes its seating plane.

Variances due to soldering imperfections are primarily caused by the pin's tendency to float on its landing pad.

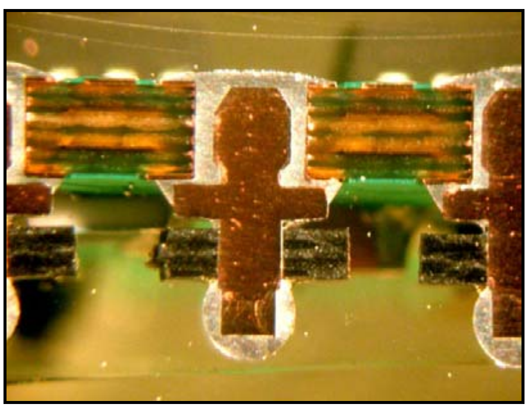

Figure 8 Cross-Section of Pin/Module Solder Joint

This is known as axial float. It can cause the pin to drop slightly due to its shoulder not being completely flush with the underside of the PCB. Figure 8 is a cross-section of a DSSMT module, taken through the pin/module joint. It shows the shoulder of the pins to be almost flush with the module's PCB surface. This indicates that pin float is well controlled during the manufacturing process. This variance is considered negligible compared to the other parameters examined, and promises to be even less significant if the module is exposed to a high-temperature, lead-free, reflow process. The pin is attached to the module with high-temperature (96.5 Sn/3.5 Ag) tin-silver solder. At high temperature this joint will also reflow, and any pins standing proud (due to axial float) will be reseated by the weight of the module. However, for the purposes of this assessment, we'll assume a token variance of 0.001 in. (0,025 mm), for this parameter.

Of the three principal factors that affect module coplanarity, pin length is the largest contributor. This is because it has a two-fold (2×) effect on the gap that can exist beneath a pin and the module's seating plane. Consider that the seating plane of the module might be established by three pins close to their maximum variance in length. A gap of 2× this variance will then exist beneath each pin that is close to its minimum variance.

Table 1 Module Coplanarity Variance to Seating Plane

Description	Variance	Multiplier	Contribution
Pin length	0.0017	×2	0.0034
PCB warp	0.003	×1	0.003
Pin float	0.001	×1	0.001
All dimensions in inches		**Total:**	**0.0074**

Table 1 summarizes the three major factors that affect module coplanarity. The results suggest that the module could contribute as much as 0.0074 in. (0,188 mm) in coplanarity variance, with respect to its seating plane.

To add confidence to this assessment, physical measurements were also made on sample lots of PTH12030WAS production parts. The measurements were made prior to their assembly to a host PCB. In each case the amount of gap, beneath the solder ball, most elevated from the component's seating plane, was measured. The results revealed that the average maximum lift of a module pin is 0.004 in. (0,1 mm), with a standard deviation of σ =0.0018 in. (0,0457 mm). This suggests the maximum process limit is 0.0094 in. (0,24 mm), assuming this parameter has a normal distribution. This compares to the calculated variance of 0.0074 in. (0,188 mm) for the pin ends. While this measurement gives some insight to the assessed variances, it includes the solder ball. The solder ball covers the pin ends and its thickness also varies[†]. Therefore the expected spread of the physical measurement is expected to be higher.

Irrespective of whether the module's calculated or measured value represents the module's true coplanarity variance, the Solderball Pin interconnects provide up to 0.014 in. (0,36 mm) of coplanarity compensation. This is

[†] The solder ball thickness over the pin end has been characterized by the pin manufacturer as having a standard deviation of σ =0.0013 in (0,033 mm).

sufficient to ensure that all the module's interconnects form a solder joint with the host PCB.

Qualification to IPC-9701

The reliability of the host board solder joints were evaluated using the procedure set forth in IPC-9701 [4], "Performance Test Methods and Qualification Requirements for Surface Mount Solder Attachments." Thermal cycling qualification was carried out on forty two test modules, designed to simulate the PTH12030WAS product. This is the full production sample size per IPC-9701. An additional ten samples were used to verify the integrity of the joints after rework.

The PTH12030WAS is one of the larger DSSMT modules incorporating Solderball Pin interconnects. It uses the regular (63 Sn / 37 Pb) tin-lead solder ball. The test modules were fabricated using the same manufacturing methods used to build the functional PTH12030WAS module. They were then attached to seven larger host PCBs (six per host PCB), using 0.006 in (0,15 mm) thick tin-lead solder paste and 235°C maximum reflow temperature. This is the same solder-reflow limits recommended to customers. Both the test module and host PCBs were designed to allow the solder interconnects to be continuously monitored for electrical continuity.

Using the prescribed test and monitoring methods, the forty two module PCBs were subjected to a total of 3,500 thermal cycles, over the temperature range, 0°C to 100°C. The results revealed zero failures [4]. Additional analysis was made by way of cross-sections. These were conducted on parts that had been freshly soldered to a host PCB. They allowed the macro inspection of the solder joints around the interconnect pins. There were no apparent defects. Figure 9 is an example of a cross-section. It shows that the pin has established a generous solder fillet with the host PCB landing pad. This meets the requirements of IPC-A-610 [1] , for an acceptable SMT solder joint (Figure 1).

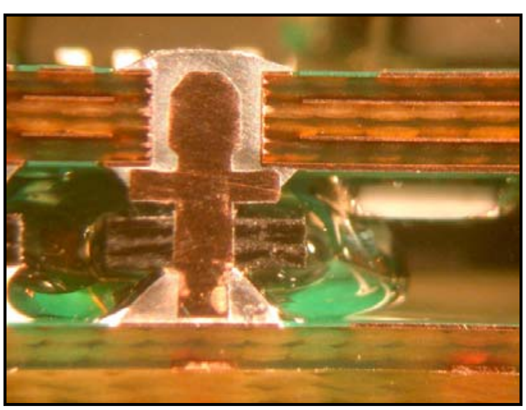

Figure 9 Cross-Section after Reflow to Host PCB

Summary

The latest power modules produced by Texas Instruments are manufactured in a compact, double-sided surface mount (DSSMT) package outline. The surface mount compatible versions of these packages use a Solderball Pin interconnect for their electrical connection with the customer's host PCB. These interconnects incorporate a solder ball at the end of a solid copper pin. The solder ball compensates for the coplanarity of a large module, allowing it to be assembled to the host PCB using a standard solder paste stencil and surface mount solder-reflow process.

The amount of coplanarity compensation provided by the Solderball Pin interconnects was compared against the variance that may be introduced by a large DSSMT module. The evaluation revealed a minimum compensation capability of 0.014 in. (0,356 mm), versus a potential total variance of 0.074 in. (0,188 mm). This concludes that the Solderball Pin interconnects provide more than sufficient solder to compensate for the module's own coplanarity variance. The integrity of the solder joints, between the module and host PCB, were also qualified to IPC-9701. The tests performed showed good component-to-PCB solder joint integrity. This translates to improvements in both manufacturing yield and component reliability.

References

[1] IPC-A-610, Rev. C. Acceptability of Electronic Assemblies, January 2000.

[2] Solderball Pin™ Interconnects Application Notes by Autosplice, Inc.

[3] 081-45028, PCB Flatness Testing, Texas Instruments internal document.

[4] IPC-9701. Performance Test Methods and Qualification Requirements for Surface Mount Solder Attachments, January 2002.

[5] 081-45026, Rev. 1A. IPC 9701 Test Report PTH/PTB Series; Texas Instruments internal document.