Presented in the ECWC 10 Conference at IPC Printed
Circuits Expo®, SMEMA Council APEX® and
Designers Summit 05

Filling Pastes in PCB Production – Fields of Application, Possibilities and Limitations

Sven E. Kramer
Lackwerke Peters GmbH + Co KG
Kempen, Germany

Abstract

In the past the use of filling pastes in PCB production was largely limited to via hole fillers. These materials with a solids content of 100% are still successfully employed today to close via holes and thus ensure their proper sealing for vacuum adaptation during incircuit testing. Furthermore, they are used to avoid the deposition of flux residues that may create critical microclimates in the holes and/or under components. However, there is only limited use for these products in newer fields of pcb manufacturing.

The latest filling materials (also referred to as plugging pastes) are largely used in Sequential Build-Up technology. Due to their specific properties, these materials enable the manufacturing of buried and blind via holes.

Thick film fillers are of growing importance in case of extremely high copper build-up (also termed 400 µm technology) in order to allow a leveling of the traces before a solder mask can be applied.

All of the discussed materials belong to the electrically non-conductive type.

Filling Pastes for "Thick Copper Technology"

Especially in the automotive industry the demand for PCBs manufactured in thick copper technology is growing. These high copper thicknesses are needed in case of high current applications. Typical fields of application with 400 µm high copper build-up are PCBs for fuse boxes. Also multilayers with up to 400 µm high layers are possible. With regard to the 42 Vs discussion the 400 µm technology is also one of the main topics.

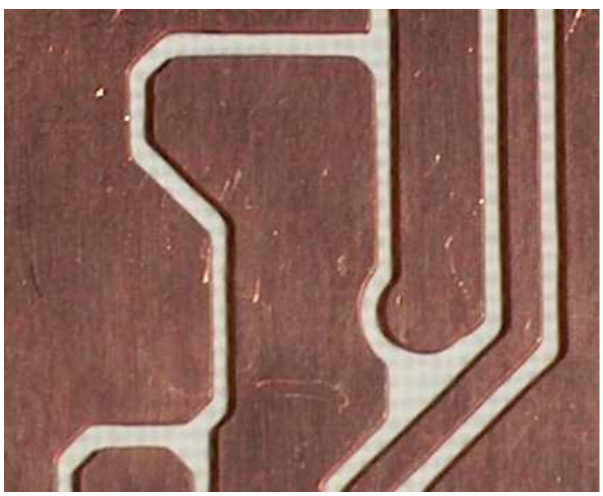

Figure 1 - Detail of an Unfilled PCB in 400 µm Technology

The coating of these 400 µm high copper traces, however, confronts PCB manufacturers with considerable process as well as reliability problems. Compared to "conventional" PCBs, which of course can be coated with solder resist without a prior filling of the gaps in between the traces, an additionally carried out filling step increases the edge and side wall coverage of these traces, which ensures the desired electrical insulation properties. (See Figure 1.)

Limitations of Photoimageable Solder Resist Processing

When curtain coating, a considerable amount of solder resist will flow from these high traces into the area in between the traces. As a consequence, the resist thicknesses in these areas can be enormous. A complete removal of the formulated solvents/thinners is not possible within the typical drying window that leads to high amounts of residual solvents (typically, the residual solvent content should be below 2%). This remaining solvent can be found underneath the dried solder resist surface and will inevitably lead to blisters due to evaporation effects during subsequent temperature stress applied to the PCB or assembly (e.g. solder resist curing process and soldering processes).

A similar problem occurs when a screen defined process is used for solder resist application. Additionally, the difficulty arises that the substrate between the traces has to be wetted with solder resist. When using electrostatic coating methods the side wall coverage is particularly difficult, and – due to lacking "dielectrically mass" – a potentially insufficient insulation results.

Usually, the maximum conductor height that can be coated is in the range of 150 – 160 µm when the above mentioned modes of application are used. In some cases and especially when air-assisted coating methods are used the maximum coatable conductor height is in the range of approx. 200 µm. However, it has to be considered that in most cases multiple coatings have to be applied (at least a double coating) in these instances. Besides the costs involved in such an approach also the resulting risks should be observed. Usually, multiple coatings are not applied "wet-on-wet" or after predrying of the first layer, but after carrying out all processing steps in order to minimize a potentially critical residual solids content.

Thick Film Filler Requirements

The main difficulty for all of the above mentioned solder resist application methods is the so-called solvent retention, i.e. the collection of residual solvents due to incomplete drying. A potential solution to this problem is the use of solvent-free, UV curing materials that basically have to fulfill the following requirements:

- solvent-free
- fast and complete cure also in deep layers
- good adhesion to copper and base material, moreover also enabling subsequent solder resist coating/adhesion
- good sandability
- low volume shrinkage
- flexibility
- solder bath resistance (including compatibility with lead-free soldering processes)
- UL approval according to UL 94

A few of these so-called thick film fillers are available today which can completely fill the space between 400 µm traces and enable the subsequent application of solder resist with any of the usual coating methods.(See Figures 2 and 3.)

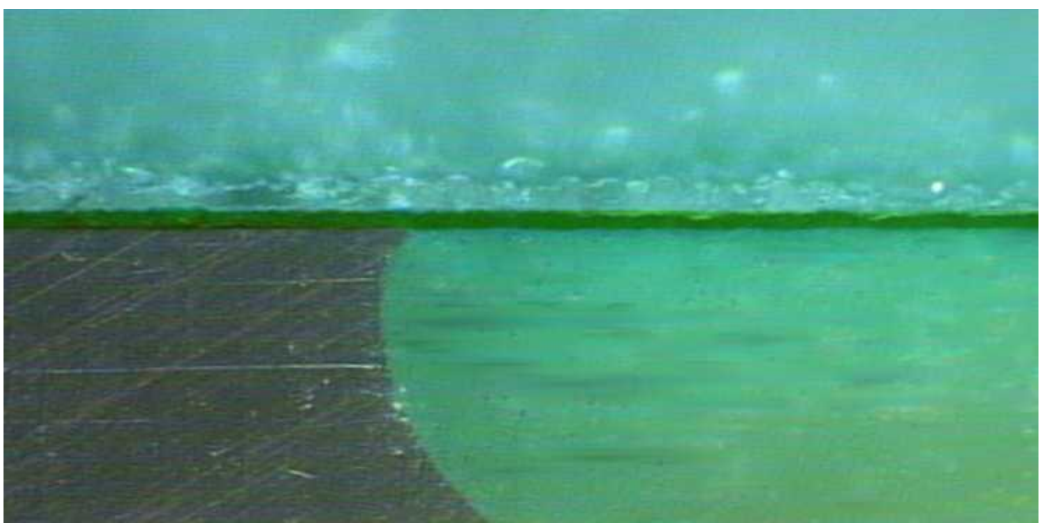

Figure 2 - Detail of a Cross-Section of a Filled PCB Coated with Solder Resist in 400 µm Technology

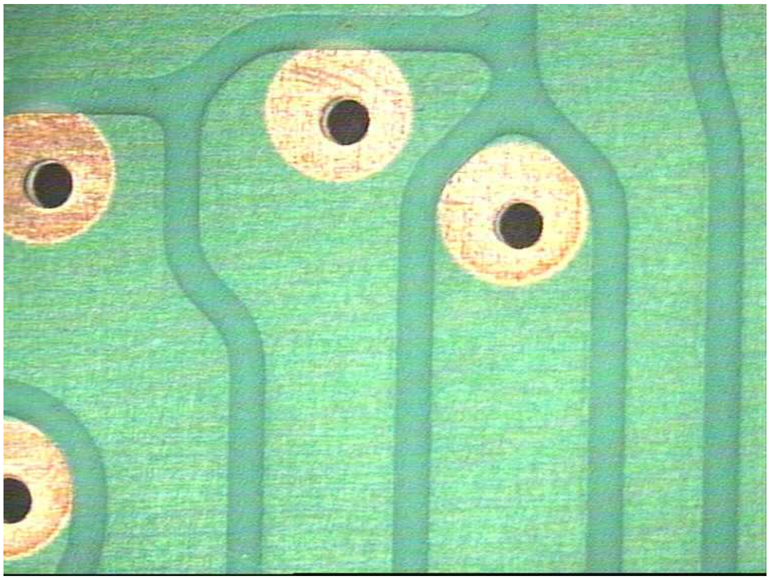

Figure 3 - Example of Static-Flex Application of Filler in 400 µm Technology

Basic Characteristics, Mechanical and Electrical Properties

Recently introduced 1-pack filling pastes were developed in order to minimize the risk of any air bubble formation during mixing of 2-pack systems resulting in a potentially reduced reliability as well as generally increased user friendliness. Of course, the basic properties and characteristics had to remain the same. (see Table 1.)

Table 1 - Characteristics, Mechanical and Electrical Properties

Characteristics	2-pack thick film filler	1-pack thick film filler
Viscosity at 20°C	26,000 ± 3,000 mPas (A+B)	26,000 ± 3,000 mPas
Density (mixture)	1.37 ± 0.05 g/cm³ (A+B)	1.37 ± 0.05 g/cm³
Solids content	100%	100%
Pot life of mixture	> 10 days	not applicable
Storage ability	6 months	6 months
Dielectric strength	70 kV/mm	75 kV/mm
Surface resistance	2×10^{14} Ohm	2×10^{14} Ohm
Volume resistivity	2×10^{15} Ohm x cm	2×10^{15} Ohm x cm
Solder resistance 20 s at 265°C	passed	passed
Glass transition temperature	(TMA) 40°C	(TMA) 32°C
Coefficient of thermal expansion	(TMA) 140 ppm/°C < Tg (TMA) 190 ppm/°C > Tg	(TMA) 52 ppm < Tg/°C (TMA) 260 ppm > Tg/°C

Processing of Thick Film Fillers

These materials are usually applied directly onto the substrate before drilling/punching is carried out, typically without the use of a screen. In case a set of double squeegees is used, they can temporarily be "parked" on the aluminum metal frame/stencil. (See Figure 4.)

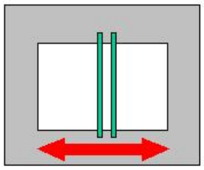

Figure 4 - Application set-Up for Thick Film Filler

The principal process flow is carried out as illustrated in Figure 5:

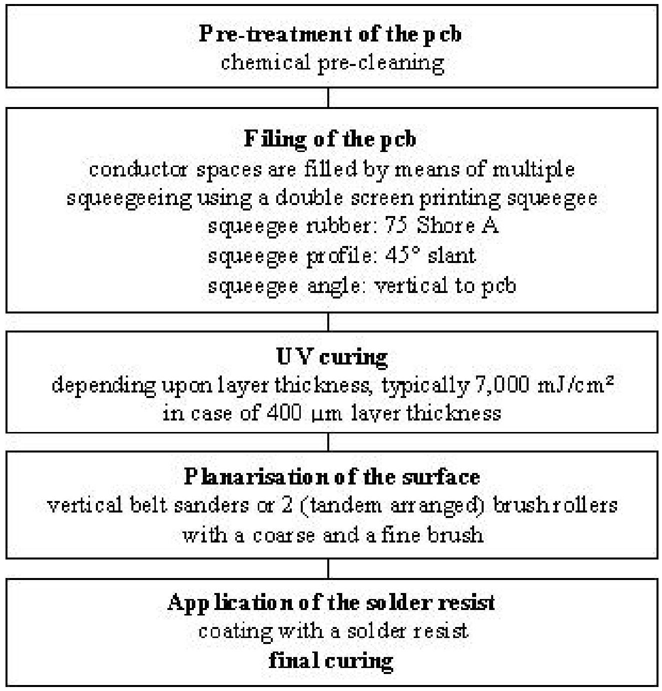

Figure 5 - Typical Process Flow for Thick Film Filler Application

PCBs already drilled and with plated-through holes before the filling process unfortunately cannot be processed as described above because these holes would be filled as well. Depending on the individual PCB layout, however, a stencil could be used or a temporary tenting of the areas (e.g. with dry film) not to be filled can be carried out. Promising is also the use of a temporary plugging paste, which was originally formulated to protect plated-through holes during etching processes. Initial trials confirm a general compatibility with typical thick film filling processes. In this case the holes are initially plugged and the temporary filler is cured followed by the application and curing of the filler. Finally, the plugging material is removed by means of an alkaline stripping process.

Independent of the filling process used for the application of the thick film filler it is obvious that not only the spaces in between the traces are filled but that also material is deposited (in a thin layer) on the copper surfaces which has to be removed after curing the filler. For this cleaning step especially vertical belt sanders have proven to be very useful due to the effective removal and low mechanical stress applied (Figure 18).

Application of Solder Resist

Multiple testing revealed that a joint thermal cure of filler and solder mask is a prerequisite in order to ensure stability during soldering processes and adhesion to both substrate and copper. In principle, both conventional thermal curing 2-pack solder resists as well as photoimageable solder resists can be used with very good results. Pure UV curing solder resists, however, have proven to lead to a fairly low bonding between filler and resist simply because a thermal cure is missing which "bakes" both systems together. Thermal curing solder resists with a high degree of flexibility as used in the field of flexible circuitry have proven especially useful because the thermal expansion characteristics of fillers are compensated, reducing the risk of potential cracking of the solder resist especially at the interface of filler-copper – solder resist. (See Figure 6 and 7.)

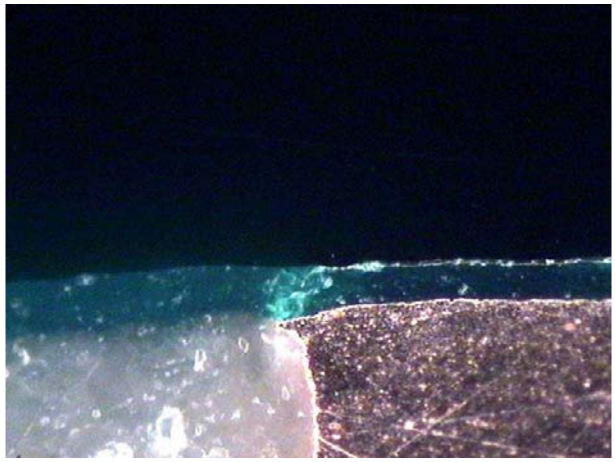

Figure 6 - Example of Crack at the Interface of Filler-Copper – Solder Resist

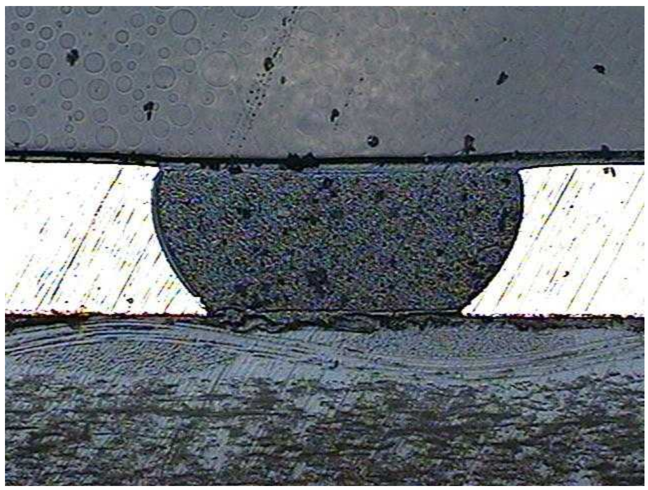

Figure 7 - Example of Separation at Filler / Copper Interface

Via Hole Plugging of SBU Printed Wiring Boards
Introduction
Owing to the increasing use of High Density Interconnect PCBs (HDI PCBs), the plugging of microvias has become an inevitable subject for many manufacturers of printed circuit boards. It becomes obvious that the plugging of vias has its "peculiarities". Starting with all the diverse types of vias, the various requirements on the filling materials, the different application processes right through to the required post treatments - numerous questions arise. Therefore a considerable number of PCB manufacturers outsource this process to specialized service enterprises that carry out both the plugging and planarization process.

"Microvias" is a key term in HDI technology. Microvias are defined by:

Hole diameter < 300 µm (< 150 µm acc. to IPC 6016)
And/or hole density > 1,000 vias/dm².

The term HDI (High Density Interconnection) means a structure with lines/spaces < 120/120 µm and/or the use of blind/buried vias (European Technology and Trend Report 2001/2002). The build-up structure comprises an inner layer (core) – with or without plated-through holes – and the subsequent build-up layers. Various combination possibilities exist.

The term "plugging" is connected with these buried vias, the plated-through holes in the inner layers.

The term "plugging", that is meanwhile internationally interpreted as the planar filling of buried vias, has become a key process in microvia technology. In order to properly define this term, the subsequent sections list the different types of plated-through holes and the various demands on the filling material related thereto.

Different Types of Plated-Through Holes

Plated-Through Holes

Plated-through holes are metallized holes that pass through the entire PCB. These vias do not require a planar, complete filling as no sequential layers are built up.

In this case "traditional" via hole fillers are used. They serve the purpose of filling the via hole in order to prevent solder from passing to the component side and also of ensuring the sealing for vacuum adaptation in the in-circuit test as well as of preventing flux agent residues from settling in the vias and forming microclimates there or underneath components.

Demands on the via hole filler:

- solvent-free or solids content of 100% to avoid volume shrinkage and solvent inclusions
- resistance to process chemicals
- resistance to Hot-Air Leveling processes

These products have been available for many years and their processing is well understood. For this application purpose also photoimageable products are used. The main difference to pure photoimageable solder masks is the fact that the solids content is as high as possible. These fillers are printed prior to solder resist application and are then jointly further processed. The advantage of using these products is, in the case of high integration, the avoidance of the resolution limitations encountered with conventional products and the avoidance of the practically ever-present "hump" in case of the traditional fillers. A disadvantage, though, is the potential risk of solvent inclusions that can lead to cracking or blistering during curing or soldering operations. A slow ramp-up of the curing temperature is therefore mandatory.

Blind Vias

Another form of vias are the so-called blind vias. They provide the interconnection of the sequentially built-up layers as well as to the core.

During lamination of the next layer air inclusions may occur that would lead to cracks in the PCB in case of later thermal stress. In case of resin-coated foils, extensive filling is achieved by using the laminate resin. However, this is not filled and may lead to cracking in case of thermal shock loads owing to the substantial expansion in z-axis direction.

The plugging of blind vias turns out to be very difficult because in this process air inclusions can hardly be avoided. The use of liquid epoxy dielectrics that enable the flow-filling of the vias without air inclusions is a possible alternative since suitable filling materials can affect lower coefficients of thermal expansion of the dielectrics. The use of conventional methods for filling blind vias has largely proven to be extremely difficult because air inclusions are practically unavoidable. In these cases newer technologies are used that operate under vacuum and which are also able to fill blind vias. Principally, this technology is very similar to vertical screen printing machines. (See Figure 8.)

The plugging paste is pressed through the vias and into the buried vias under vacuum and the especially created squeegee is moved over the PCB. The special squeegee is shaped like a trough that lies airtight on the PCB and is fed with the pressurised plugging paste by means of hoses.

Figure 8 - Vacuum Plugging Machine with Opened Front at the Front the Special Squeegee with Filler Reservoir

Buried Vias
The third type of via which is the subject of the subsequent plugging process is the so-called "buried via". These plated-through holes pass through the entire inner layer core of a multilayer and are covered by the sequential build-up layers. They differ from blind vias in that in the beginning they are "open" and can thus be more easily filled as regards the avoidance of air inclusions because filling is effected from one side and the air can escape on the other.

Reasons for Plugging
Buried vias must be plugged completely and as level as possible in order to avoid air inclusions in the course of the HDI build-up. For production processes on the outer layers, such as photo processes, use of liquid lacquer, liquid dielectric for photo vias or laser/additive build-up, planarity is required.

By means of coppering the via only, plugging enables a "virgin" inner layer that can be structured without limitation.

When using photoimageable liquid dielectrics as well as laser-ablatable liquid epoxies plugging is indispensable because the liquid dielectric cannot flow sufficiently into the microvias and a very strong dip formation can be observed. However, even over the use of resin-coated copper foils, with which the filling of the microvias can be affected via the resin of the copper foils, plugging offers decisive advantages. As mentioned above, only the plugging process enables a subsequent structuring without the limitations on account of the sunken via holes. Thus a significantly higher integration density of the circuit can be achieved. Also because of the partial filling of the vias a slight dip in the build-up layer results that limits the application field of the HDI circuit.

The coefficient of thermal expansion (CTE) is another value that has become more important as a result of HDI technology. The trend towards smaller components, smaller spaces between the I/Os and the demand for lead-free soldering requires the adjustment of the coefficient of thermal expansion of the base material to the values of the components. With regard to the dimensional stability, the CTE is becoming a central value.

In case of thermal shock loads, especially during soldering processes, the thermal stresses that occur on account of the different coefficients of thermal expansion of the combined materials are increased by the air entrapped in the vias which may result in partial delamination or cracks that may lead to failures. (See Figure 9.) Here, thermal shock loads on account of repeated soldering processes have to be regarded very critically.

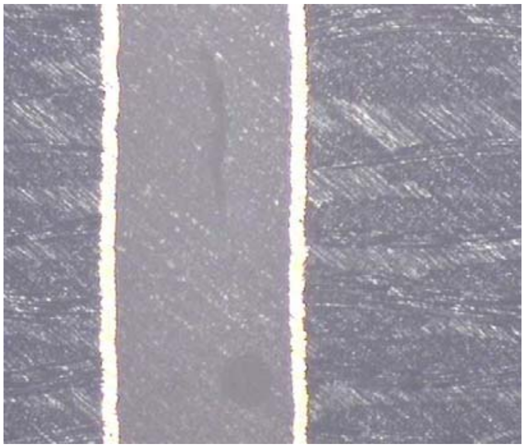

Figure 9 - Crack and Bubble in Plugged Hole

The basic requirements of a plugging paste are summarised in the following list:
- 1-pack system, if possible
- solvent-free, 100% solids content
- good adhesion to the copper barrel, even under thermal stress and soldering conditions
- good adhesion of dielectric and photo resist
- free of air inclusions
- highest possible glass transition temperature, Tg > 140°C
- CTE < 50 ppm (below Tg) and as low as possible above Tg
- no shrinkage during curing
- easy brushing without any risk of cavernous or dipped surface
- good adhesion to the copper plating under thermal stress as well as under soldering conditions

- depending on field of application (e.g. avionics and space) ASTM E595 requirements regarding total mass loss (TML) and collected volatile condensable materials (CVCM) have to be met
- good storage stability in normal room and ambient conditions should be evident

The Plugging Process

As a rule, plugging is effected after the PCB - usually the inner layer - has been drilled and the vias have been subsequently metallized, but prior to structuring. After plugging of the vias and thermal curing of the plugging paste the plugging paste - that exhibits a slight nail head - is mechanically levelled.

Depending on the further use, a coppering of the plugging paste is effected so that an intact, "virgin" copper layer is achieved. The process that in detail is much more comprehensive is schematically represented in Figure 10.

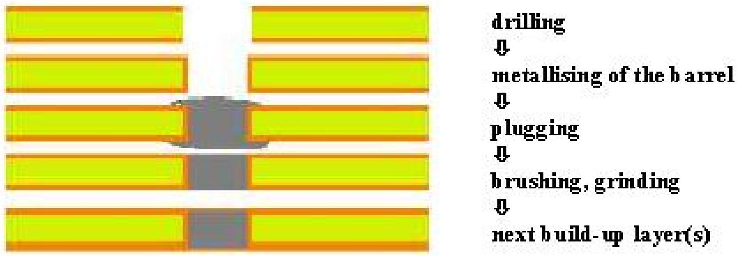

Figure 10 - Process Flow for Applying the Plugging Paste

An important identification value for the plugging process - also for the choice of plugging process - is the ratio of PCB thickness to hole diameter, the so-called "aspect ratio". As the aspect ratio increases, the difficulty of a reliable plugging increases also. Thanks to the roller coating process, the aspect ratio has meanwhile increased to 15. Prototypes with inner layers of up to 3 mm have been plugged by means of a roller coater (Figures 15 and 16). The vacuum plugging machine as well as the power squeegee for sample series' enable even higher aspect ratios to be plugged.

The plugging pastes have to be filled into the holes without any air inclusions and close them evenly (Figure 11). Existing air inclusions in plugged vias may lead to caverns in the required even surfaces after planarization – sanded air bubbles – that do not warrant uniform coppering. (See Figure 12.)

If required by the process the planarized plugging paste can also be metallized by means of common processes. Such coppered buried vias enable a higher integration density of the next layer (Figure 13).

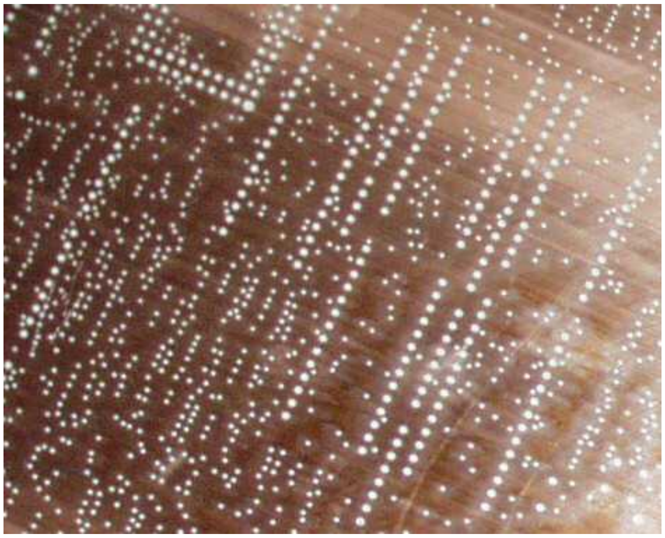

Figure 11 - Plugged PCB prior to the Brushing/Grinding Process

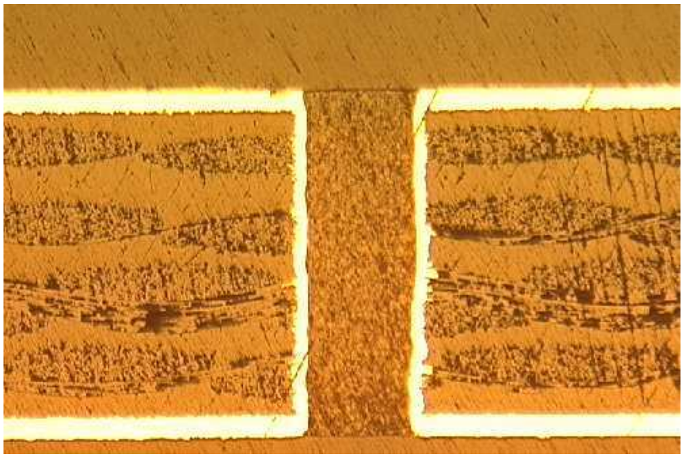

Figure 12 - Highly Planar Surface and Bubble-Free Filling of a Microvia Circuit after Brushing/Grinding

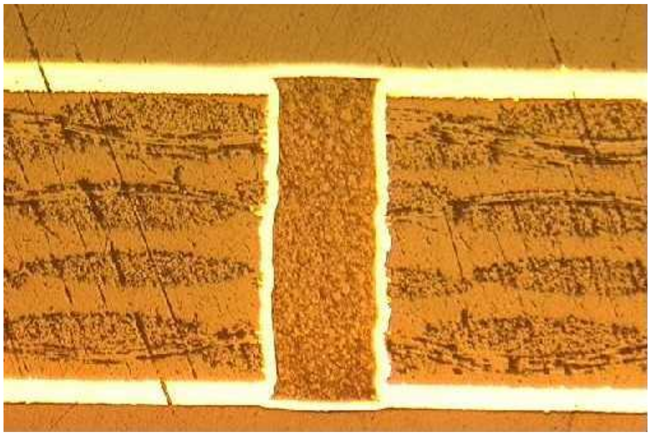

Figure 13 - Highly Planar Surface of a Microvia Circuit with Subsequent Metallization after the Plugging Process and Prior to Application of the Dielectric of the Next Layer

Usually plugging pastes have been especially developed to fill multilayer buried vias. The solids content of 100% as well as a specially adjusted filling should render a low coefficient of thermal expansion of <40 ppm possible so that in case of thermal shock loads no cracking or delamination occurs and, as already explained, a sufficient dimensional stability is ensured. A second key thermal identification value is the glass transition temperature (Tg). Generally, the highest possible Tg is required - the value should not be below 140°C. A specification of the Tg is designed to ensure that in the required temperature range no changes in the properties that are typical for the glass transition temperature occur.

Generally, there are two processes available to plug buried vias. Both processes have already been used in series production:

1. screen/stencil printing process
2. modified roller coating process

Both processes are based on non-structured, plane copper-plated PCB cores.

Plugging by Means of Screen/Stencil Printing

When plugging by means of the stencil printing process with a plugging paste the vias are filled by means of a corresponding stencil. Placing a stencil that was drilled with the same drilling pattern but with larger hole diameters underneath the PCB (backup board) has a favourable effect (see Figure 14). The paste should be printed in such a manner that a so-called nail head remains on the underside. Of course, the printing parameters largely depend upon the aspect ratio of the vias.

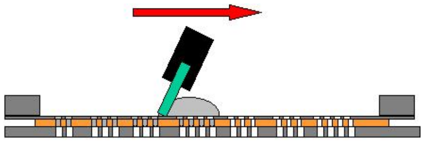

Figure 14 - Plugging Principle using a Screen/Stencil Printing Process

The following parameters are typical for via holes with a diameter of 0.4 mm (Table 2).

Table 2 - Typical Printing Parameters

PCB	Diameter of holes to be plugged	0.4 mm
	PCB thickness	1.45 mm
Screen	V2A screen mesh	80 mesh
	Stencil thickness	100 µm capillary film
	Hole diameter in stencil	0.65 mm
Flooding	Type	metal (standard)
	Squeegee angle	90°
Printing	Type	75 Shore A
	Squeegee angle	90°
	Squeegee profile	45°
	Printing speed	slow

Advantages:

- selective plugging, i.e. only those vias are filled that need to be filled (positioning holes can be kept free)
- stencil printing is a well-known and established process
- the equipment already exists

Disadvantages:

- a printing stencil is required for each layout
- difficult printing in case of substantially varying hole diameters on the PCB
- more limited due the aspect ratio

Plugging By means of Modified Roller Coating

Plugging by means of the roller coating process is effected by forcing the paste on a roller through the vias (see Figure 16) and requires materials with specific flow characteristics. Usually the viscosity is lower compared to screen/stencil printing materials. The plugging paste is kept in a trough under the coating roller that picks up the paste on its underside. The board is pressed against the coating roller from above and the paste that is located between roller and PCB is pressed though the vias. Then the board passes between two stripper squeegees so that the plugging paste residues are removed from the board. If the process requires a highly even and planar surface of the vias it is necessary to plug with a slight hump that is then levelled mechanically. In order to realise a slight nail head on both sides it is proven practice to position the stripping squeegees as flat as possible and slightly offset. The angle between squeegee and PCB should be < 45°. In the plan view of a modified roller coater in Figure 15 the two transport rollers for the PCB can be seen on the right while the coating roller can be seen in the centre. On the left is the guide for the squeegee system.

Figure 15 - Plan View of a Roller Coater for the Plugging Process

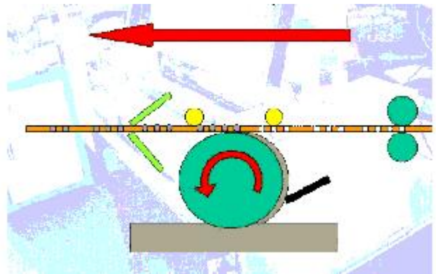

Figure 16 - Scheme of a Roller Coater for the Plugging Process

Advantages**:**

- fast, secure filling of all vias
- simple process, only a few parameters have to be adjusted
- no screen stencils are required, i.e. independent of layout
- simpler planarization as the PCB is almost produced clean
- depending upon the stripping squeegee setting, a slight dip (sinking-in of the plugging paste) or a slight nail head (hump) can be realised
- the modified roller coating process can be used for in-line technology

Disadvantages:

- vias that are not to be plugged have to be masked
- cannot be used for structured PCBs because the squeegees cannot remove excess material from between the conductors
- only suitable for plugging materials that have a good storage stability at room temperature because the roller coater trough capacity is 8 - 10 kg

Thermal Curing of the Plugging Pastes

After filling, thermal curing of the plugging pastes is effected. Typical curing conditions are 45 min at 150°C. This time has generally to be seen as an object holding time, i.e. the time is measured from the point when the panels reach the curing temperature after a heating-up phase. Depending on the paste used, longer curing times and higher curing temperatures are possible and give slight advantages regarding the achievement of a higher Tg.

With a view to the plugging paste's required properties of a glass transition temperature of > 140°C [284°F] and a lowest possible CTE value, it is absolutely necessary to observe these curing conditions. Incomplete cross-linkings can have a negative effect in subsequent thermal stress situations.

Planarization of the Plugging Pastes

During levelling – planarizing –, smears and residues of the plugging paste are removed from the PCB by means of brushing or grinding processes and the PCB as well as the plugs are levelled.

This process involves some critical aspects. For instance, attention must be paid to the fact that the edges of the copper barrels are not removed. Moreover, it has to be ensured that the plugging paste is not partially removed from the vias.

The planarization plays a key role in the plugging process. Various levelling technologies are available that are described below.

Practical experience shows that hard brushes with a ceramic surface give very good results in order to ensure a reproducible and perfectly even surface. But there have been some innovations in this field and meanwhile further brushing and grinding processes are available.

A grinding process that works with rotating grinding discs like with an orbital sander is offered. The advantage of this process is surely the favourable cost-benefit ratio, however it is currently not yet optimised for mass production but an interesting process, especially for small series' (Figure 17).

Figure 17 - Orbital Sanding Unit for Planarization of the Inner Layers after Plugging

A further grinding process is based on a grinding belt combination with an abrasion in the micrometer range. Especially the removal of the relatively hard "nail heads" of plugging paste is effected with highest precision.

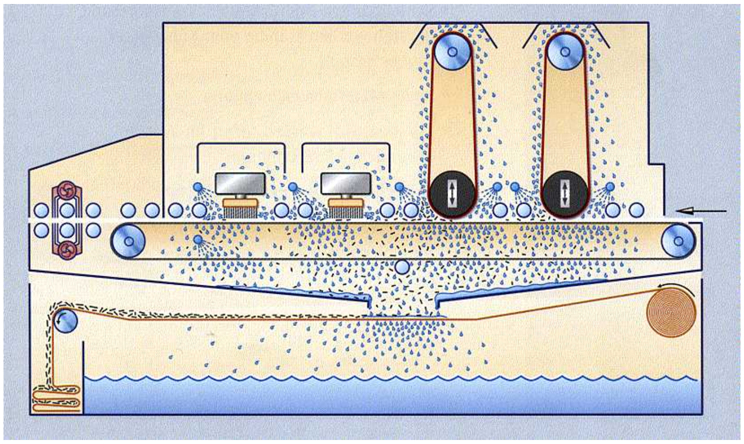

Figure 18 - Belt Grinding Unit for the Planarization of Inner Layers after Plugging (Schematic)

The belt grinding unit consists of two or three vertical belt grinders of different grades and pressure force (Figure 18). Belt grinding units of this kind are, for instance, also used successfully to remove resin residues from pressing plates. A further advantage is that when using two grinding units with grinding belts of various grades a very fine copper surface can be achieved.

This process is distinguished by the possibility of enabling high throughput but also entails higher investment costs than the formerly described orbital sanding principle. This grinding process is also offered by contract grinding companies.

Figures 19 and 20 show two examples before and after the grinding process.

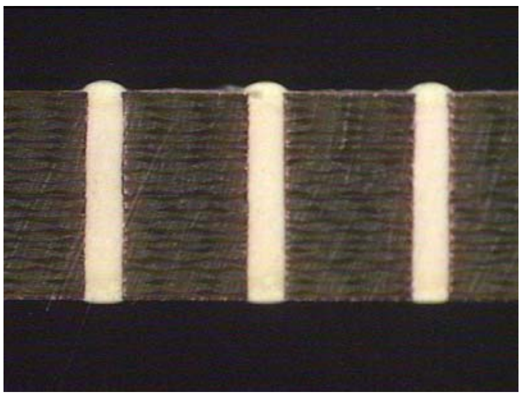

Figure 19 - Plugged Plated-Through Holes prior to the Grinding Process

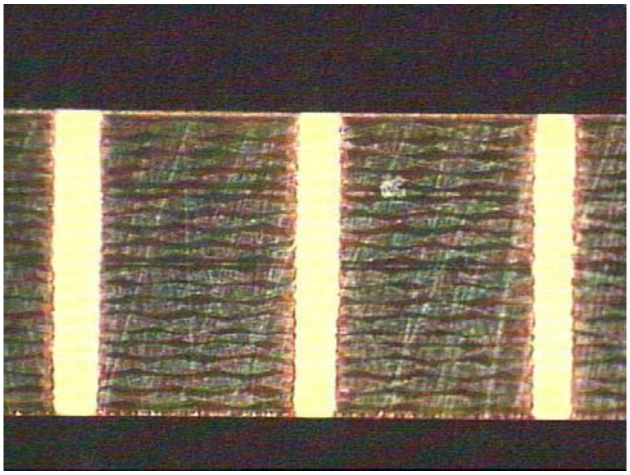

Figure 20 - Plugged Plated-Through Holes after the Grinding Process

Moreover, special ceramic rollers are used with which excellent results can be achieved. These rollers are used in deburring machines with a special brush regulation chiefly in the Far East but have also proven their worth in European series production. Their disadvantage is the relatively high process costs.

Metallization of the Plugging Pastes

Dependent upon the requirements on the production process a metallization of the filled vias is necessary. On the one hand a metallization of the inner layers in case of buried vias is required to achieve a higher wiring density in the next layer, and on the other hand in case of outer layers when the plugged via leads to a pad (**v**ia **i**n **p**ad (VIP)).

Generally, the process to metallized the plugged vias is similar to drill hole metallization. After levelling the plugged PCB the surface must be activated.

The preparation of the resin matrix is ensured by means of the so-called "desmear" process.

A schematic flow is as follows:
1. swelling of the resin with special solvents, e.g. N-methylpyrolidone
2. etching with a permanganate solution
3. reducing

The process as well as the process equipment of the desmear process have to be adjusted to the plugging technology. Horizontal process flows are favoured here also.

The following SEM photographs show the surface topographies of a plugging paste before and after the desmear process (Figures 21 and 22):

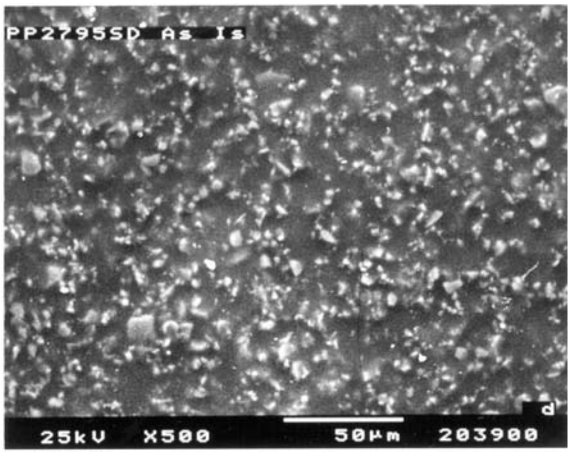

Figure 21 - Plugging Paste prior to the Desmear Process

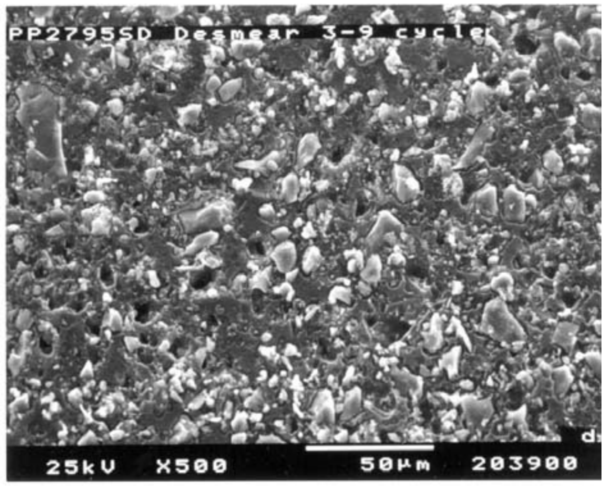

Figure 22 - Plugging Paste after the Desmear Process

Reference

1. Plugging processes – Filling of buried vias by means of the plugging process, Technical Information 15/15, Lackwerke Peters GmbH Co + KG, Kempen, 2002
2. Suppa, M.: Via-hole Plugging von SBU-Leiterplatten, report 154 D, Lackwerke Peters GmbH + Co KG, Kempen, 2002
3. Kramer, S.: Füllpasten in der Elektronik, report 157 D, Lackwerke Peters GmbH + Co KG, Kempen, 2004
4. Technical Datasheets PP 2795 and PP 2795-SD, Lackwerke Peters GmbH + Co KG, Kempen, 2003
5. Preliminary technical product information DSF 2706 UV and DSF 2707 UV, Lackwerke Peters GmbH + Co KG, Kempen, 2002