ELPEPCB®
peters

AI APPLICATION INFORMATION

# AI 2/1: Processing instructions for photoimageable Elpemer® solder resists

This Application Information Sheet gives detailed and in-depth information that is imperative for a secure and reliable processing of the photoimageable **Elpemer®** solder resists as well as for obtaining ideal coating results.

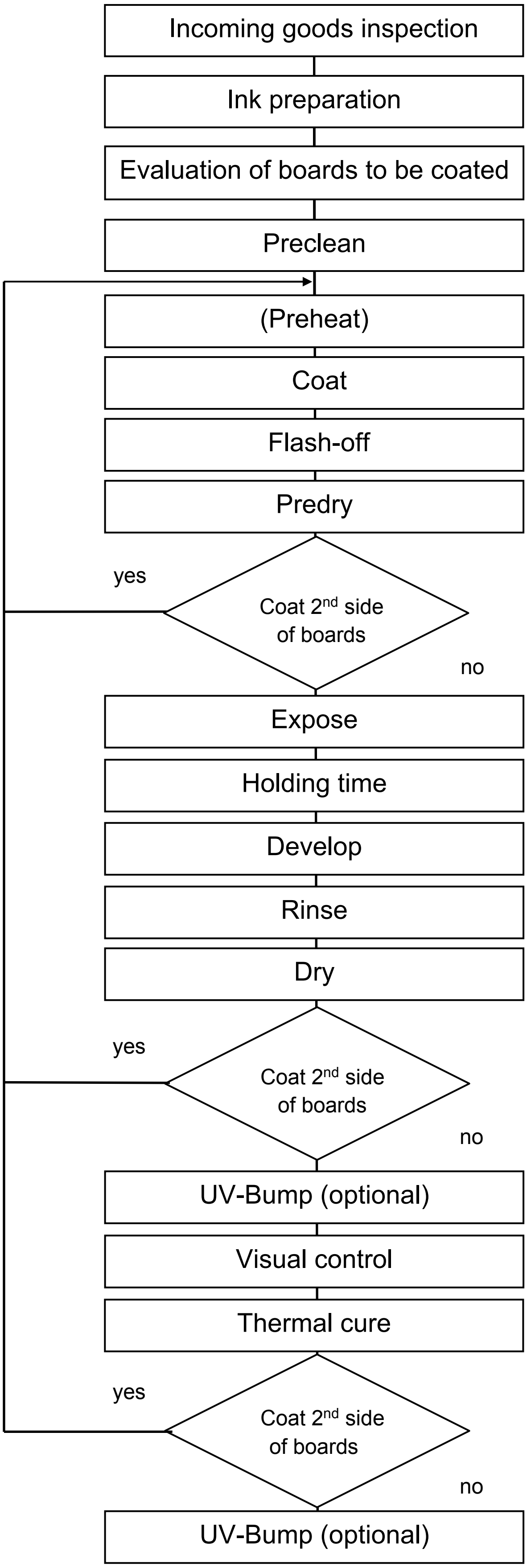
This AI 2/1 covers all photoimageable **Elpemer®** ink systems (curtain coating, screen printing and spraying inks). Basically, the advice on processing photoimageable systems is also applicable to photoimageable **Elpemer®** photoresists and marking inks.

**Contents**

LP 141212 E-3 / ai2-1e.003

# Schematic of the process flow

Exact parameters for each process step are given in the relevant process data sheets.



# Applicable documents

| | |
|---|---|
| ℹ | Please read this technical report and the publications listed below carefully before using the product. These sheets are enclosed with the first shipment of product or sample. |
| **MSDS** | The corresponding material safety data sheet contains detailed information and characteristics on safety precautions, environmental protection, transport, storage, handling and waste disposal. |
| **PD** | The process data sheet contains product-specific data such as characteristics and recommendations for processing parameters. |
| **TR** | The Technical Report gives information and details on auxiliary products and standard packaging. |
| **TI** | Technical information TI 15/3 “Protective measures when using chemicals including lacquers, casting compounds, thinners, cleaning agents“ |
| **TI** | Technical information TI 15/10 „Processing of 2-pack systems“ |
| **TI** | Technical information TI 15/13 “Precleaning in the pcb fabrication process“ |

# Safety recommendations

→ When using chemicals, the common precautions should be carefully noted.

→ Ensure that extractor units of workplace ventilation arrangements are positioned at solvent source level.

→ Please also pay attention to national guidelines or directives concerning operating safety such as the German TRBS (technical rules for operating safety) and those concerning the handling of flammable liquids as for example the German TRbF (technical rules for flammable liquids) or European directives.

→ Ensure that the utilised equipment meets the requirements of the material safety data sheet.

→ **When handling solvent-based systems, within the scope of the legally prescribed/ required hazard assessment measures, it is imperative to observe the relevant explosion protection regulations (including appropriate national health and safety regulations, technical guidelines, harmonised EN norms and EU directives plus any other recognised technical rules). The key physical characteristics of the individual products are stated in Section 9 "Physical and Chemical Properties" of the corresponding material safety data sheets.**

# Protection against sunlight and UV light

→ **Elpemer**® systems are light-sensitive, therefore it is imperative to avoid exposure to sunlight and UV light.

Protect against UV light

The use of yellow light or yellow filters/UV foils is mandatory. We gladly name manufacturers of suitable films and lamps upon request

# Ink preparation

Prior to processing, the **Elpemer**® solder resists and thinner (if used) have to be brought to room temperature. The containers to be processed the following day should be transferred to a room that has the same temperature as the processing room.

The two components (ink component A and hardener component B) are already packed in the correct mixing ratio. The volume of the container of component A is sufficiently large to accommodate the entire quantity of component B plus any thinner required for adjusting the viscosity:

➔ Observe the mixing ratio in percent by weight, when mixing smaller quantities.

➔ Mix the resist and hardener components as well as the thinner (if used to adjust the viscosity) very carefully until a homogeneous mixture is achieved.

We recommend to use mechanical stirring equipment, e.g. a basket stirrer, observing a stirring time of about 10–15 minutes. Stirring units that stir in air, such as propeller stirrers, are unsuitable.

➔ Pour **screen printing inks** into an empty container and stir again.

This prevents unmixed material from being processed from so-called “dead zones” thus avoiding potential quality issues.

For more detailed information on correct mixing please read our **Technical Information Sheet TI 15/10 "Processing of 2-pack systems".**

➔ Observe a holding time of 30 min after mixing.

During this time the ink can degas to avoid breaks in the curtain and micro bubbles in the coating.

➔ Observe the pot life at room temperature (18–23 °C [64–74 F]) specified in the appropriate process data sheets.

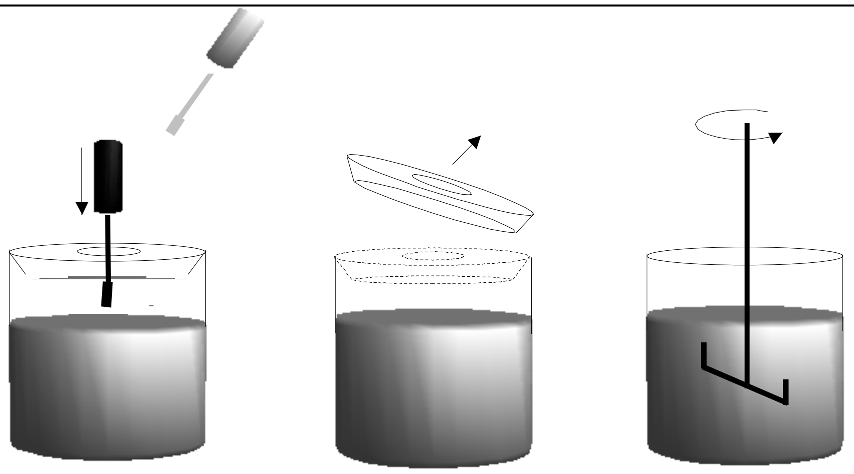
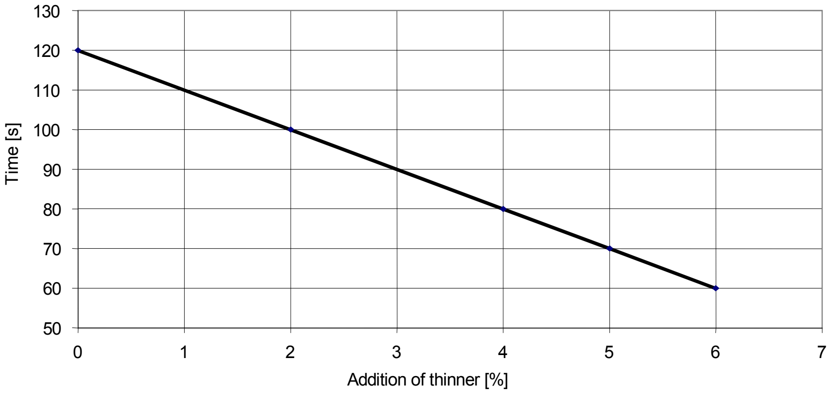


**Fig. 1:**
**Opening the combi-pack and mixing the ink components of the curtain coating inks**

The **Elpemer® curtain coating inks** are supplied in practical combi-packs where the hardener component is located in the container lid. This ensures a clear assignment of resin and hardener components. The integrated hardener container is pierced with a sharp object (e.g. a three-square or screw driver) so that the hardener can flow into the resin component. The conical shape of the top container guarantees complete discharge, after which mixing of the components can start immediately.



The flow time is measured – here according to DIN 53211* in a 4 mm dip cup – at 20 °C [68 F].

The percentual addition of thinner refers to the viscosity of the mixture in its as-received condition, i.e. 120 s.

In addition to the reference viscosity, the temperature dependency must equally be considered.

**Fig. 2: Viscosity curve (flow time) for Elpemer®GL 2467**

The illustrated relationship is affected by process and equipment parameters. For this reason, the data in the diagram should be seen as a guideline only and, if necessary, be adjusted to the actual production environment.

# Precleaning

An oxide-free metallisation with an average peak-to-valley height of 2 µm (0.8–1.2 µm for flexible laminate) and a perfectly clean, dry substrate are prerequisites for optimum adhesion of the coating without dewetting or flaking in follow-up processes and for achieving the maximum electrical properties of the solder mask.

→ In our **Technical Information Sheet TI 15/13 "Precleaning in the pcb industry"** you will find detailed advice on the various precleaning methods. Coat as soon as possible after precleaning in order to avoid oxidation or contamination during temporary storage.

Particularly some flexible base materials tend to trap humidity.

→ If necessary, temper the base material before coating to avoid adhesion problems.

Since even slightest silicone residues may cause dewettings, tools and equipment must be meticulously cleaned when processing ink systems that contain silicones.

# Preheating

Owing to the well-balanced relation between thixotropy and flow characteristics of the **Elpemer®** solder resists, a preheating of the printed circuit board prior to coating is generally **not necessary**.

# Coating

The photoimageable **Elpemer®** solder resists have been developed for various applications and coating processes. If possible, the ink should be processed in air-conditioned clean rooms. The characteristics (density, viscosity, etc.) and processing parameters are contained in the corresponding Process data sheets and the specified final properties (electrical and physical data) are contained in the relevant Technical report.

The individual coating processes

- printing (horizontal or vertical)
- curtain coating
- screen spraying (conventional or electrostatic)

are described in the following.

Since the many different permutations make it impossible to evaluate the whole spectrum (parameters, reactions with materials used, chemical processes and machines) of processes and subsequent processes in all their variations, the parameters we recommend are to be viewed as guidelines only that were determined in laboratory conditions. We advise you to determine the exact process limitations within your production environment, in particular as regards compatibility with your specific follow-up processes, in order to ensure a stable fabrication process and products of the highest possible quality.

The product data specified in the technical reports is based upon standard processing conditions/test conditions of the mentioned norms and must be verified observing suitable test conditions on processed printed circuit boards.

Feel free to contact our application technology department (ATD) if you have any questions or for a consultation.

## Curtain coating

With curtain coating, the printed circuit board is accelerated to a speed of up to 90 m/min [300 ft/min], conveyed through the ink curtain and decelerated again. Since the ink curtain and pcb speed are almost identical, the solder resist precipitates on the printed circuit board as an even blanket.

The solder resist is continuously pumped from a reservoir into the pouring head and flows back to the reservoir via a collecting chute with an inclination of 30° in order to avoid air entrapment.

Air inclusion in the solder resist may lead to blistering on the printed circuit board; in addition, large air bubbles cause the curtain to tear which, in turn, leads to voids on the printed circuit board.

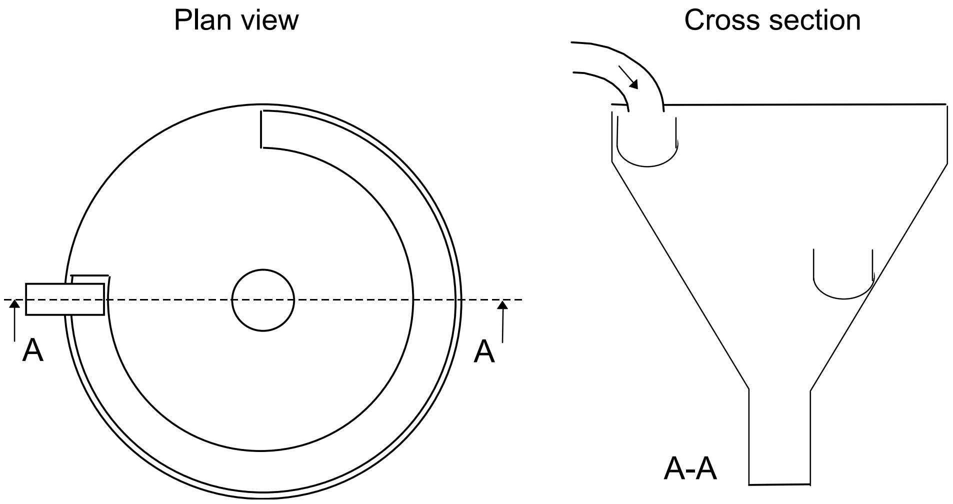


**Fig. 3: Design principle of an ink collecting chute**

In order to maintain constant process parameters, the viscosity of the solder resist is permanently monitored by a viscosity measuring system and kept within the specified viscosity tolerances by an automatic dispensing unit during the pumping process.

➔ Owing to the thixotropy of the **Elpemer**® solder resists, the viscosity should only be controlled and adjusted while the pump is in operation, because the shearing forces occurring during pump operation already have a viscosity-reducing effect.

The temperature of the solder resist is kept constant by means of a heating/cooling system in the ink circuit.

➔ Adjust the amount of ink applied by means of the pump capacity/speed.

Frequency-controlled pumps allow a relatively exact preadjustment.

- → Make fine adjustments by weighing the applied wet ink quantity on a test pcb.
  Prepare sections of base material of a defined size (for instance, 200 x 300 mm). Weigh one section on scales accurate to at least ± 0.1 g and then coat. Immediately after coating, weigh the board again. The difference between both readings is the wet ink weight in g/6 dm² / g/0.06 m² when the abovementioned format is used or, after division of the measuring result by 0.06, in g/m².
- → Ensure reweighing is effected **directly after** coating because otherwise the measuring result will be falsified by the evaporation of the solvents.
- → If the second printed circuit board side is coated immediately after the predrying process, turn the board through 180° prior to coating.

This turn will prevent the walls of the (plated-through) holes from being double-coated with solder resist when the second side of the pcb is coated. If the walls are double-coated, the developing time will have to be extended to avoid residues of the solder mask remaining in the holes. An extended developing time can lead to unnecessary deterioration of the resolution of the solder mask.

- → If necessary, reduce the distance between the pouring head and the printed circuit board, the flow rate of the pump/coating speed and/or adjust a higher ink viscosity (> 80 s according to DIN 53211) to avoid ink seeping through larger holes (≥ 5 mm) to the underside of the pcb.
- → When using a coating edge restriction, take into consideration that this will create drifts of ink along the uncoated edges of the pcb; these drifts will have a significantly higher coating thickness than the other printed circuit board areas.

The shorter the distance between pouring head/coating edge restrictor and printed circuit board surface, the less pronounced the ink drifts.

- → Ensure that no large registration and location holes are positioned in the vicinity of the ink drifts in order to avoid seepage and that flash-off and predrying are adjusted to achieve as complete drying of the ink as possible, including the coating edge areas, without overdrying the ink in the other areas.

However, owing to the wide predrying window of the **Elpemer®** solder resists it is extremely unlikely that this will happen.

## Screen printing

Screen printing is a contact-printing process in which the solder resist is transferred onto the printed circuit board through a screen printing fabric (with or without stencil) by means of a squeegee.

The squeegee normally has a Shore A hardness of 65–75 and a right-angled profile. As a rule, the squeegee angle is 75–80°, or 20–30° for vertical screen printing.

- → If higher ink films are required, it is advisable to slightly round the edges.

Polyester screen fabrics or corresponding steel meshes of 32–100 to 54–64 (according to old nomenclature 32–54 T) with a screen tension of at least 25 N/cm (or as advised by the screen fabric manufacturer) are recommended.

- → With the exception of vertical double-sided screen printing, keep the squeegee pressure as low as possible in order to avoid oozing at the conductor edges.

All other screen printing parameters depend upon the pcb surface.

When coating flexible **Elpemer®** solder masks, adjust the flood speed, squeegee speed and screen snap-off as low as possible in order to avoid micro bubbles.

For good coverage, also of conductors that are located at right angles to the squeegee direction, a “wet-on-wet“ print is recommended.

The **Elpemer®** solder resists can be printed into so-called blank screens (i.e. without a stencil) or with a stencil that covers the holes. The diameter of the covers needs to be slightly larger than the hole diameters. A high stencil build-up is not necessary because the stencil is only required to fill the screen mesh. Covering the holes has the advantage that no ink plugs form in the plated-through holes that would prevent an adequate flow through the drill holes in the developing process and thus unnecessarily extend the developing time. The filling of holes can also be effectively avoided by off-setting the printed circuit boards in x-y direction after every print.

→ In order to guarantee the free development of the holes it may be necessary to extend the developing time and, in turn, the exposure time. Please note that the resolution grade may suffer slightly.

Also the processing method (single- or double-sided) as well as the predrying method (convection drying, IR drying or a combination of the two) influence the developing performance and thus the decision whether a stencil is required or not.

The following are fundamental advantages of the screen printing process over the curtain coating method:

- owing to the higher solids content, a substantially lower wet-ink weight is required
- the rheological properties (structural viscosity) prevent flow-off from the conductor edge so that a better edge coverage is achieved.

For further advantages of screen printing see section ”Double-sided screen printing (vertical)”.

### Single-sided screen printing

In the single-sided screen printing process, initially the first printed circuit board side is printed with solder resist, predried, exposed, developed and cured. Then the second side is printed and processed accordingly.

### Double-sided screen printing (horizontal)

Double-sided application by means of horizontal screen printing whilst ensuring a high degree of process safety can only be accomplished with special screen printing equipment. Since screen printing is a contact printing method, the underside of the printed circuit board is subjected to a pressure load during the printing process. This load can cause damage to the initially applied ink coat if this was predried only. If the first printed circuit board side is exposed and developed before the second side is coated, this may lead to increased undercut and thus to poorer resolution on the first side (by the duplicate developer action on the first side).

A double-sided “wet-on-wet” coating is only possible if the printed circuit board is fixed on adapter pins in the plated-through holes when printing the second side.

### Double-sided screen printing (vertical)

Vertical screen printing units allow the simultaneous coating of both printed circuit board sides. Semi- and fully automatic units are available for a variety of different printed circuit board thicknesses and panel sizes.

Semi-automatic units consist of a manual loading and unloading section and a fully automatic printing section.

Fully automatic units have the additional feature of loading the panels from a stacker or tray. After the printing cycle, the panels are stacked and transferred automatically into a vertical conveyorised dryer.

The main differences among the individual screen printing machines are the panel size and thickness that can be processed.

Depending upon the machine manufacturer, there are also process engineering differences such as:

- pneumatic or electromechanical drive
- printing cycle with or without separate flooding of the screen mesh
- "push-stroke" or "pull-stroke" squeegees
- fixed or variable squeegee angle.

The quantity of ink pressed into the holes depends strongly on the adjusted printing parameters.

In spite of these differences, all machines offered operate along the same basic principle:

- an absolutely uniform coating on both sides of the printed circuit boards is achieved in one pass owing to a relatively flat squeegee angle combined with a very high squeegee pressure of the self-adjusting squeegee units
- the special printing technique guarantees a uniform coat over and between the conductors. A reliable print without "skipping" can be achieved even with high conductors and very small conductor widths and spaces.

This application process boasts the following advantages:

- simultaneous coating of both printed circuit board sides; thus printing on two equally clean and oxide-free surfaces
- both printed circuit board sides are predried together; this means a lower energy consumption and no risk of overdrying such as can occur in the curtain coating process on the printed circuit board side coated first.

## Spraying (conventional or electrostatic)

In the spraying process, the solder resist is finely atomised by compressed air and transported to the printed circuit board by this air (conventional spraying) or additionally charged electrically (electrostatic spraying) and transferred to the earthed printed circuit board via the static charge in an electrical field and at a low air pressure.

The printed circuit board sides are coated (and predried) horizontally by means of an oscillating spray head that is positioned at right angles to the conveyor direction. As a rule, the spray nozzle and/or atomised air is heated to approx. 50–100 °C [122–212 F]. This leads to a direct reduction of the ink viscosity in the spray nozzles and enables easy atomisation of the solder resist.

Heating the spray nozzles enables the application of relatively high-viscosity ink systems, on the one hand making the use of high-solids ink possible and on the other hand ensuring fast predrying.

The spray application of **Elpemer®** solder resists boasts the following advantages:

- coating and predrying of both printed circuit board sides in one pass
- optimum flow and good edge coverage; no wetting problems (“skipping“) when coating printed circuit boards with high conductor densities and unfavourable layouts – even areas of the printed circuit board that are difficult to access can be safely coated due to the fine atomisation of the ink particles

- only small amounts of ink in (even very small) via holes
- excellent ratio between ink on conductor and ink on base material.

A serious disadvantage of this spraying technique is the overspray that (a) leads to an increased ink consumption and (b) to considerable soiling of the pcb handling systems. However, newest equipment with defined overspray recycling is expected to promote considerable savings in this respect.

## Double coating

On account of special customer demands with regard to edge coverage, dielectric strength, etc. double coating may be required. For adequate resistance to electroless nickel/gold or tin baths a conductor edge coverage of at least 10 µm should be aimed for. In principle, there are three possible processes that are described below.

➔ Perform pretests to secure optimum coating results.

**The second coat is applied directly after predrying the first coat**

The advantage of this process is that maximum possible productivity is reached because all further process steps, commencing with exposure, are performed together. The disadvantage or difficulty with this method is that the printed circuit boards pass through the predrying cycle four times which substantially reduces the process window of this step and may easily cause overdrying and thus incomplete free development. If the latter is not detected in time (prior to final curing), it can lead to irreversible defects (for instance, defective tinning) on the printed circuit boards.

**The second coat is applied after exposing and developing the first coat**

The main advantage of this method is also the relatively high productivity achieved for double coating, because the particularly time-consuming final curing of the inks is effected simultaneously.

The risk of overdrying in this case is not higher than with single coating; however, inadequate predrying of the first coat may cause problems. It is extremely hard for solvent inclusions and an excessively high residual solvent content of the first coat to escape from the coat after the cross-linking reaction triggered by the exposure. This may lead to these solvents explosively evaporating in the hot-air levelling process and thus causing delamination. In order to avoid this, observe the following:

➔ Select the wet-ink weight of the first coat as low as possible in order to ensure optimum drying; if necessary, this can be compensated for by a correspondingly higher wet-ink weight of the second coating which would leave the final coat thickness unchanged.

➔ For the same reason, set the predrying parameters of the first coat at the upper limit of the processing window, with particular attention being paid to an adequate air exhaust (solvent discharge).

If this advice cannot be followed for some reason, it may be possible to achieve an optimum result by means of a two-step final cure, in spite of solvent inclusions. In this case, the temperature in the first step is lower (for instance, 30–45 min/80–120 °C [175–250 °F]) than the final curing temperature so that solvent residues can escape.

**The second coat is applied after final curing the first coat.**

This relatively unproductive process is the overall safest method. Nevertheless, flaking in the hot-air levelling process may also occur in this case.

The cause are contaminations on the surface of the first coat which practically constitute a dividing layer between the two ink films. There are two possible backgrounds to this:

- Contaminations on account of the handling of the printed circuit board between the two coating processes.
  Therefore, care must be taken that after the first preliminary coat the boards are treated with the same precaution as precleaned, uncoated printed circuit boards. Finger prints, etc. must be avoided at all costs; if necessary, it may be advisable to clean the boards prior to applying the second coat.
- Low-molecular inkcomponents (condensate) precipitate on the first coat when it is final thermal cured.
  An inadequate exhaust air volume during final cure of the first coat may cause this "condensate" to precipitate on the printed circuit boards making it impossible to obtain an optimum ink-on-ink adhesion.

# Flash-off

Flash-off allows potential air inclusions between the conductors to escape. Furthermore, considerable amounts of solvent are removed from the **Elpemer**® solder resist in this process step. Flash-off takes place at temperatures of max. 50 °C [120 F]. A good exhaust air extraction is vital. In addition, it should be possible to direct the supply air exactly across the printed circuit boards right from the start of the flash-off process in order to avoid blistering – dewettings and/or accumulation of ink around the drill holes due to the stack effect – and to obtain optimum edge coverage.

The air quantities and temperature in the flash-off zone should be adjustable. The flash-off time must comply with the specifications in the Process data sheet.

➔ If the printed circuit boards are stored vertically during predrying sufficient air exchange should already be ensured during flash-off to promote fast drying, thus preventing the ink from dripping off (“teardrop formation“).

With a high Cu build-up (70 µm [2.8 mil] base copper), high ink film thicknesses may be encountered directly at the conductors. In the case of an inadequate flash-off time or an excessively steep temperature profile of the predrying process, this ink film thickness may lead to solvent inclusions that produce so-called "pin holes".

# Predrying

Predrying serves the purpose of removing the solvents still contained in the solder resist after flash-off and of drying the coat to such an extent that the coating of the second side or exposure is possible without sticking, imprints or other damage to the first side.

The predrying process takes place in a time/temperature window that varies depending on the ink system used. The lower limit is fixed by the mechanical strength of the solder mask. The upper range of the predrying process, both regarding temperature and time, must be chosen in such a manner that no polymerisation of the solder mask starts to take place and thus optimum development of the unexposed areas remains guaranteed.

➔ With double-sided processing remember that the side coated first may be predried twice.

Generally speaking, predrying can be performed by means of straight convection drying, if necessary with integrated flash-off, as a combination of convection and IR radiation or as a pure IR cure process.

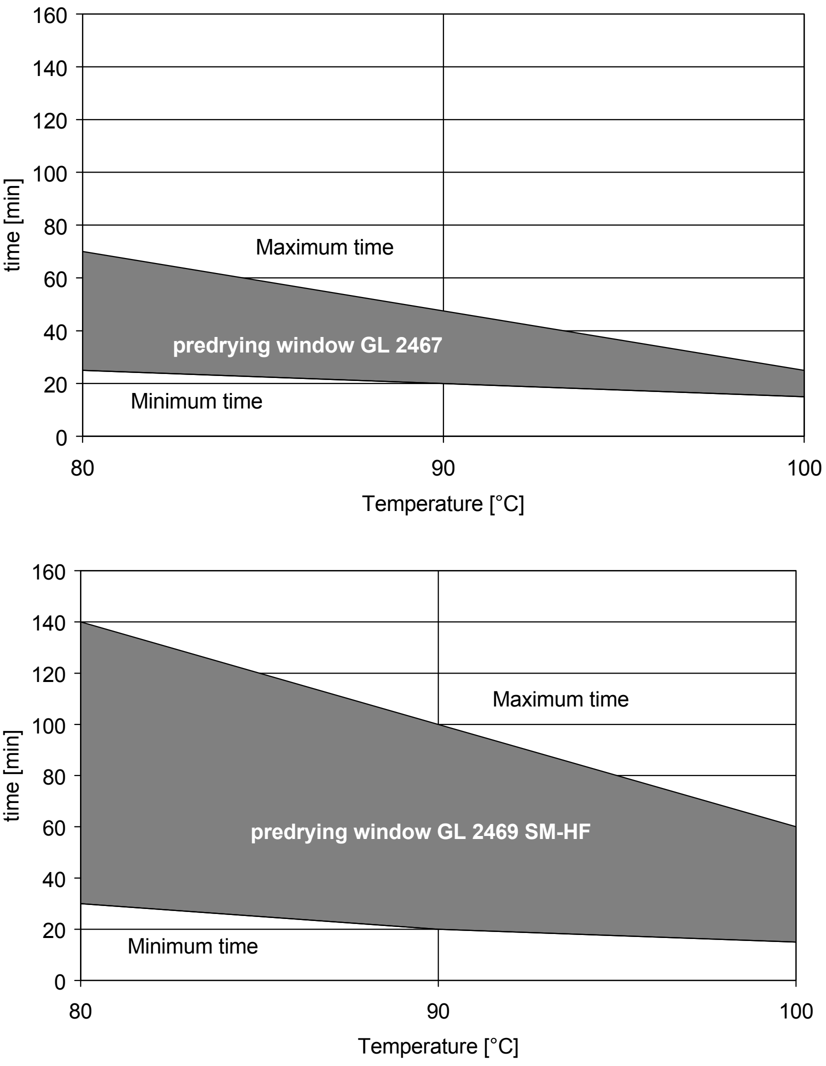
The pure convection drying process requires more floor space and has longer processing times. On the other hand, one can work at lower temperatures which offers the following advantages:

- the solder resist is dried more effectively and “gently”
- lower risk of large solvent quantities remaining in the deeper layers which negatively affect the resolution of the solder mask. This is of particular importance in the case of higher ink quantities
- problems with imprints caused by the transport system in the case of double-sided processing are reduced, especially with very thin laminate
- since the temperature load can be kept at a much lower level, this means a higher mechanical stability of the base material in the predrying process.

Besides adequate air ventilation for the effective discharge of the solvents in the predrying process, it is important to ensure sufficient cooling in the outlet of the predrying unit.

After cooling, the printed circuit board temperature should be lower than 30 °C [86 F] in order to avoid mechanical damage during transport and stacking, because at this stage the coat is still thermoplastic. In the case of double-sided processing, the second side can be coated after the predrying cycle.

For double-sided coating after the predrying phase, the entire transport system should have a V-shaped layout, or the printed circuit board transport should be designed in such a manner that the reverse side of the printed circuit board will not lie flat on its side, which will effectively avoid possible damage to the coat.



The lower limit defines the mechanical strength of the solder mask.

The upper limit shows the beginning of the polymerisation of the solder mask. The optimum development of the non-exposed areas is possible up to this limit.

It has to be considered that the predrying window is dependent upon a variety of parameters (e.g. the layer thickness, the exhaust air volume, the printed circuit board condition, the holding time after predrying).

**The illustrated relationship was determined under ideal laboratory conditions; it is strongly affected by process and equipment parameters.**

**Fig. 4: Example of a predrying window for Elpemer® GL 2467 (above) and GL 2469 SM-HF (below) when using convection drying**

- **Formation of condensate**

When predrying photoimageable solder resists, particularly those with extremely short exposure times, a certain formation of condensate may be encountered. This condensate formation can be more prominent when hot air/IR precuring are combined.

In order to prevent this condensate from being returned to the predrying oven, an adequate separation of warm and cold air and, if necessary, the installation of a condensate trap is required.

- **Holding time**

Attention must be paid to the fact that the maximum holding times between the process steps (predrying – exposing – developing) stated in the Process data sheet are not exceeded because otherwise a complete free development may no longer be guaranteed.

### Note on environmental protection

In the process steps “Flash off“ and “Predry“ the majority of the solvents in the corresponding ink system escape. In this regard the directive “on the limitation of emissions of volatile organic compounds due to the use of organic solvents in certain activities and installations“ (EU-VOC-RL) should be considered that aims for a reduction in the emission of volatile organic compounds (with a vapour pressure of 0.1 mbar at 20 °C [68 F]).

Beneficial for the solvents balance in the field of solder resists is

- the application of ink with high solids contents
- the utilisation of cleaning agents and developer media with a vapour pressure of > 0.1 mbar, e. g. **R 5899** / butyl carbitol (BDG) or aqueous-alkaline developers
- the recycling of used solvents.

Detailed information on the requirements of the VOC directive, limit values and effects on the manufacture of printed circuit boards can be found in the **Technical Information Sheet TI 15/110 "The EU-VOC regulation – Contents and consequences for the pcb industry"**.

## Exposure

The process steps “Exposure” and “Developing” must be considered in conjunction with each other because they have to be co-ordinated (see also Section “Developing”).

In the exposure process, those areas of the coated printed circuit board are exposed to UV light that are to be masked by a solder resist on the finished printed circuit board. Therefore, the areas to be developed free are covered with a suitable photo tool during the exposure process (see also Section “Photo tool”).

As a rule, for the exposure process exposure units are employed with an Fe-doped mercury vapour lamp that has a power of 5–10 kW and a maximum emission at 365 nm; for white **Elpemer**® solder resists you may also use Ga-doped lamps (see Section ”UV lamps”). A higher power will reduce the exposure time and enable a higher resolution. The exposure units should be equipped with adequate cooling so that the printed circuit boards do not exceed a temperature of about 25 °C [77 F] in the exposure process in order to avoid sticking of the photo tool.

The units must be equipped with an operating hourmeter because the service life of the UV lamps is only about 1000 to 1500 hours, after which they will have to be replaced. As the lamps grow older, the wave length range is displaced. The integrators installed in the exposure units ensure that the corresponding energy in the correct wave length range continues to be emitted. However, the wave length displacement extends the exposure times and increases the thermal stress on the photo tool (photo tool sticks to ink, dimensional stability decreases).

Increased heat radiation in the exposure process can also be caused by an incorrectly doped or defective lamp and possibly create the following problem:

Owing to the higher thermal energy, the UV light-activated photo reaction with the sensitisers that react to long-wave light continues. This reaction is sufficient to represent an adequate Stouffer step. However, the cross-linking reaction/photopolymerisation of the ink surface that is achieved especially by means of short-wave light is not sufficient for an optimum resistance of the exposed areas in the development process so that the ink film swells in the development process. This may lead the ink to flake in thinly applied areas (on narrow conductors, at the conductor edges).

The exposure frame must render a vacuum between printed circuit board and frame possible. Any remaining air should be stroked out in order to avoid air entrapment between the photo tool and printed circuit board.

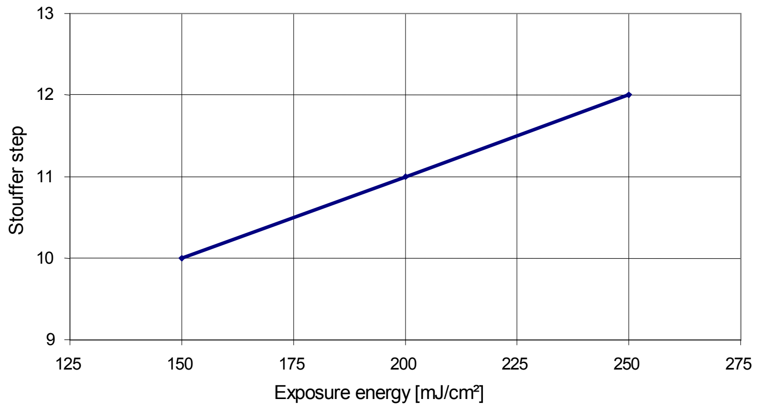
- **Glossy spots**

Glossy spots on the solder resist surface are frequently caused by too short a cooling phase of the coated printed circuit board between predrying and exposure. The ink that is still warm right after the predrying process is strongly thermoplastic and deformed by the superimposed film and the vacuum produced in the exposure process to such an extent that film sticking to the ink surface creates isolated glossy areas.

Another cause for the glossy spots is an excessive heating of the exposure unit surface during the exposure process. Owing to the deposited film and the vacuum produced, this situation can also cause glossy areas to develop on the printed circuit board due to the thermoplasticity of the ink that has not yet been final cured.

In order to avoid those glossy areas, the printed circuit board should be cooled down to room temperature prior to exposure.

Furthermore, the temperature must not rise substantially above room temperature during the exposure process. This can be ensured by means of an effective water cooling of the lamp. By reducing the vacuum, film sticking and the resulting glossy spots can be avoided.



The exposure energy refers to a 5 kW exposure unit with Fe-doped mercury vapour lamps at an emission maximum of 365 nm, measured through the photo tool at a transparent point with ORC UV model UV-350.

The Stouffer step reached is dependent upon a variety of parameters, e.g. the developing speed, the spraying pressure, the developer temperature and potentially the developer medium.

**Fig. 5: Graph of the exposure energy in relation to the Stouffer step of Elpemer® GL 2467**

**The illustrated relationship was determined under ideal laboratory conditions; it is strongly affected by process and equipment parameters.**

## UV lamps

For the exposure of the **Elpemer®** solder resists, Fe-doped mercury vapour lamps with an emission maximum at 365 mm are suitable. For white **Elpemer®** solder resists you may also use Ga-doped lamps with a maximum emission of 405/420 nm; they may shorten the required exposure time by up to 30 % and reduce undercut. Mix-doped and undoped lamps are not suitable on account of their emission spectrum).

When changing the solder resist used, attention must be paid to the fact that not only the lamp, but also the integrator that measures the radiation energy and extends the exposure time if the exposure energy decreases, may have to be replaced.

The lamp reflectors must be designed in such a manner that the entire exposure frame is illuminated with the same light energy (± 10%).

With water-cooled lamps (recommended in order to achieve a low temperature in the exposure process), the glass cladding must be checked for discolouration and contamination when replacing the lamp in order to avoid an energy drop.

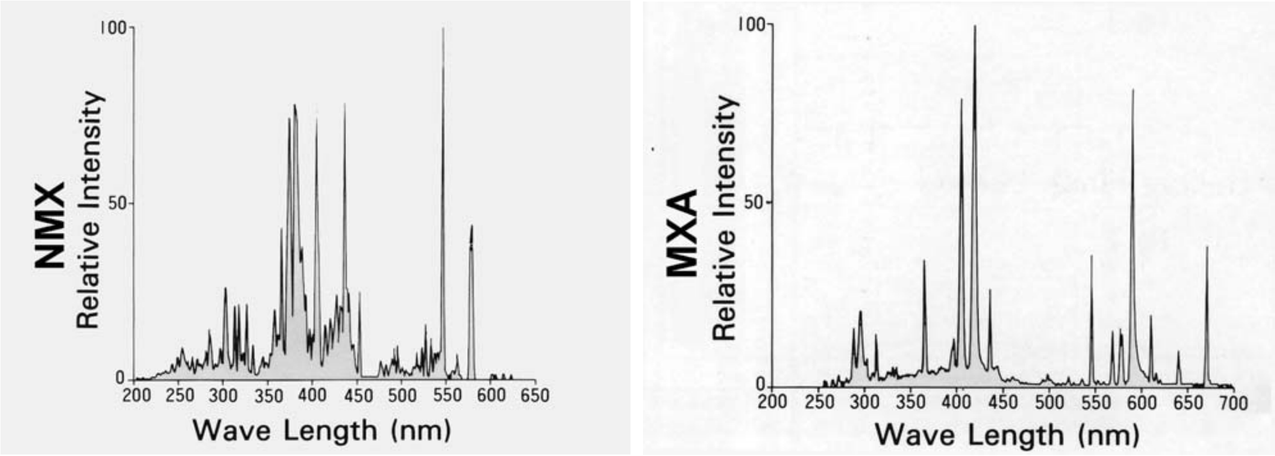


**Fig. 6 left: Spectrum of an Fe-doped lamp, peak at 365 nm (type NMX 7000H)**
**right: Spectrum of a Ga-doped lamp, peak at 405/420 nm (type MXA 7000H)**
**(Source: ORC Manufacturing Vertriebs GmbH, Düsseldorf, www.o-r-c.de )**

## Photo tool

Diazo or silver halide films can be used as photo tools. The advantages and disadvantages of the photo tools have to be weighed against each other:

- with visual registration, diazo films are the better choice because one can see through the spots impervious to UV and thus better align the layout with the photo tool.
- among others, the resolution of the solder mask is also dependent on the distance between the photo tool and the solder mask in the exposure process (stray light). Diazo films are more flexible than silver halide films and cling to the conductor structure better but they are thicker than silver halide films which partly counteracts this advantage.
- since silver halide films are black, they heat up the solder mask more intensively in the exposure process – especially with very large covered areas – and can increase sticking to the solder mask (only in the case of poorly cooled lamps).
- in the exposure process, the photo tools absorb part of the UV energy; therefore a film should always be applied when the exposure energy is measured. Diazo films absorb about 50% of the UV energy, silver halide films about 25%.

The use of so-called protective films that are laminated onto the film layer in order to extend the service life of the photo tools has to be checked very carefully. These protective films additionally absorb UV energy and increase the distance to the solder mask. This reduces the resolution performance of the solder mask due to stray light and extends the exposure time.

➔ Cut the photo tool smaller than or, at the most, as large as the printed circuit board panel so that a vacuum can be built up between the photo tool and the solder mask/no misregistration in the edge area is caused by the photo tool being pushed up at the edge by the vacuum. The photo tool is always placed onto the solder mask coated side down.

## Exposure energy

The exposure energy required for the **Elpemer®** solder resist is specified in the corresponding Process data sheet. The required wave length of the UV lamp (in nm) and the exposure energy in mJ/cm² as well as the grey level (Stouffer step) resulting after the development on a copper surface are indicated.

When using a measuring device to determine the exposure energy it must be ensured that – depending on the measuring gauge employed – the measuring unit is calibrated because the instruments of different manufacturers indicate substantially varying values. Generally, care must be taken that the measuring unit is suitable for the corresponding wave length range.

When determining the grey level, a Stouffer 21-step wedge, i.e. a film strip increasing in opacity, is placed on the solder mask and exposed at different exposure energies. When exposing test panels, corresponding film material **must** be placed on the grey step wedge because the film absorbs quite a substantial amount of UV light.

After the development process, the Stouffer step is evaluated that has just been washed off/developed. An exposure time is selected where the grey level is reached that is indicated in the corresponding Process data sheet. This is the method to set the exposure time for printed circuit boards with plated-through holes of up to 4 mm [0.16 inch]. Consider that the achieved Stouffer step is additionally dependent upon the developing speed, the spraying pressure, the developer temperature and the developer medium.

For printed circuit boards that require an extended development time because of very large holes or slots, the time required for the free development of such holes/slots has to be determined first. The exposure time to be applied is determined by means of exposure tests with the Stouffer wedge. In practical operation, a combination of both methods has proven to be useful in application processes where holes and plated-through holes are coated with solder resist.

As far as possible, the exposure energy should always be kept within the limits stated in the Process data sheet because underexposure leads to increased undercut and overexposure to stray light seeping under the photo tool. Both effects negatively affect the resolution. Furthermore, the risk of ghost-imaging on the second printed circuit board side will increase in the case of overexposure.

With thin base material (< 1 mm [< 0.04 inch]), an exposure and development test must be performed to ensure that ghost-imaging will not occur when the specified exposure parameters are used. Double-sided SMD (surface mounted device) layouts where the pads on both sides are not aligned are especially critical. In the case of ghost-imaging, these printed circuit boards must be processed single-sided or base material has to be used that is impervious to light.

Optimum exposure, in conjunction with optimum development, is always achieved when the widths of the dams on the photo tool are identical with the widths of the dams on the printed circuit board.

Fig. 7.3 shows an optimum exposure/development result.

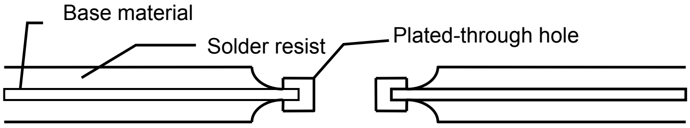


Fig. 7.1: Overexposure or underdevelopment

Fig. 7.2: Underexposure or overdevelopment (undercut)

Fig. 7.3: Optimum balance of exposure and development

**Fig. 7: Diagram showing microsections (cross-sections) of free-developed plated-through holes**

# Holding time

Depending on the ink system, a holding time of 10 min for instance is recommended after the exposure process. This process step serves the purpose of optimising the photochemical reaction activated during exposure.

# Developing – Rinsing – Drying

The process steps "Exposure" and "Developing" must be considered in conjunction with each other as they have to be co-ordinated (see also Section “Exposure”).

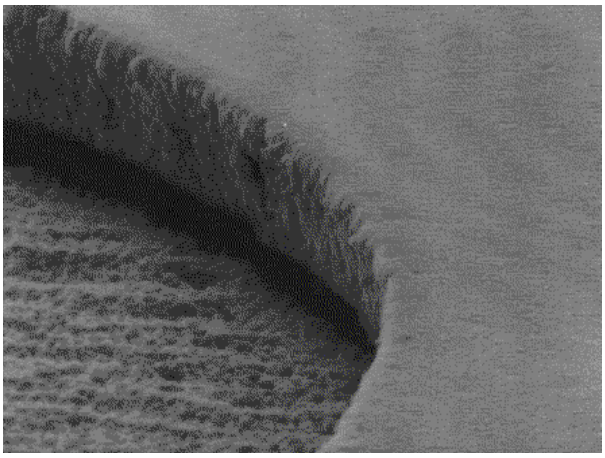
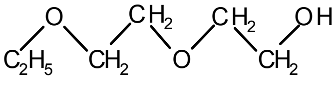


**Fig. 8: Scanning electron beam microscope image of an Elpemer® ink sidewall (700-fold magnification)**

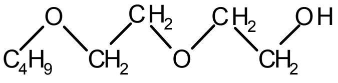
## Developer media

According to the solubility of the resins used, the developer media for the various **Elpemer® solder resists** differ. The following developer media are used:

- For the polyalcohol developable solder resists,



EDG: Carbitol ($C_6H_{14}O_3$)



BDG: Butyl carbitol ($C_8H_{18}O_3$)

- For the aqueous-alkaline developable solder resists alkalis, such as sodium carbonate solution $Na_2CO_3$

### Polyalcohol development

The development in polyalcohols is a physical solution process. The ink that was tack-dried in the predrying process and not exposed in the exposure process is dissolved again. The developer medium, temperature, spraying pressure and loading are responsible for the dissolving velocity. Developing processes in polyalcohols should be effected at a relatively cool temperature (about 25–30 °C [77–86 F]), because the development process takes place quite rapidly, even at low temperatures, and higher temperatures negatively influence the result at the edges (undercut). The polyalcohols used for developing have a flow behaviour that causes the medium to heat up relatively quickly when the developer solution is pumped. In order to keep the development temperatures low, adequate cooling must be provided. In order to terminate the influence of the developer on the solder resist, a water-rinsing process directly after the developing zone is mandatory.

The excellent suitability of the developer medium carbitol, compared to butyl carbitol, was determined in comprehensive tests.

The developing time can be decreased by about 25% without changing the exposure times or observing a poor free development. Merely the sensitivity of the ink in the high-pressure pump zone is slightly higher with carbitol. However, this can be controlled by a corresponding adjustment of the spray pressure.

When regarding the undercut, it was also found that the development in carbitol is comparatively favourable. When realising 100 µm [4 mil] lands, an undercut of max. 10 µm [0.4 mil] was measured which means a reduction of more than 30% compared to butyl carbitol.

When selecting the developer medium it should also be considered that carbitol falls under the VOC directive while butyl carbitol is not classified as a VOC (see also Section “Note on environmental protection”).

### Aqueous-alkaline development

The term "aqueous-alkaline" is widespread, but actually incorrect. According to the general formula

acid + lye => salt + water

during the development process the carboxylic group in the resin of the ink is converted with soda into a water-soluble salt. This salt is then dissolved in water and flushed off. With aqueous-alkaline solder resists the reaction basically takes place as follows:

$2\ R\text{-}COOH + Na_2CO_3 \rightarrow 2\ R - COONa + H_2O + CO_2$ (R = resin backbone)

This development process requires a certain reaction temperature. As a rule, the development process should be performed at temperatures of 32-38 °C [90-100 F]. Generally, aqueous-alkaline development processes are faster than those using polyalcohol. Water rinsing is also necessary to stop the process. Rinsing off the soda solution prevents any residual soda solution from further dissolving/developing the resist and the formation of a strong undercut.

## Developing

In the development process, some fundamental considerations are independent of the developer medium. The developer shall be provided as multiple cascade, both in the actual developing area and in the rinse area. Fresh developer solution or fresh water shall always be added to the last chamber from the direction of flow, while the corresponding quantity shall be pumped from the first chamber; this way, the feeding of the developer medium and/or the rinse water will decrease in the direction of flow.

### Edge coverage

In the developing process the solder mask is to be removed from those areas of the printed circuit board that were covered by the photo tool in the exposure process. First of all, the question arises how the edge areas of the printed circuit board or pcb panel should be treated. As a rule, the edge is exposed completely because otherwise the developer would be loaded with stripped ink faster and a free-developed edge would have to be treated in subsequent steps (hot-air levelling, electroless Ni/Au or other processes).

➔ It is vital to ensure that exposure or non-exposure of the edge is performed unambiguously.

An edge that is not covered light-tight can create a considerable problem. In the development process, an ink "smear" will form that cannot be developed off completely, deposits itself on the transport system of the developer and is stamped on to free developed areas causing spots that are unwettable in subsequent processes.

### Development from plated-through holes

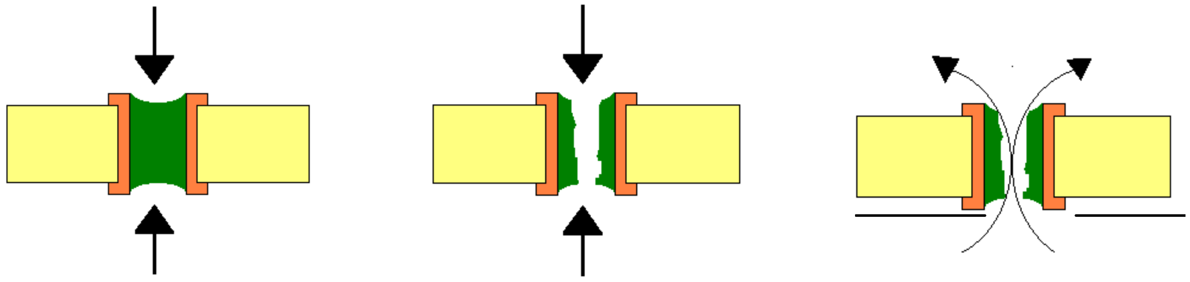
Irrespective of the application method, ink is deposited in drill holes, plated-through holes, cut-outs or similar during the coating process. In the curtain coating process mainly large holes are affected while small holes, as a rule, are tented by a "ink skin" that will tear during the flash-off process. In the screen printing process, particularly in double-sided screen printing, it is more the small holes that are filled with ink; this can result in the formation of actual ink plugs.

For the free development of large holes it is important to operate with a relatively high spray pressure (about 4 bar [58 psi]) and flat-cone nozzles. The nozzles must be inclined at an angle of about 60° to the printed circuit board surface so that the developer medium can attack the ink in the holes as long as possible and with a high pressure.

The associated developing time depends on the time that is required to remove the ink from the holes. These development conditions inevitably create a more substantial undercut of the ink edges because these areas are developed too long and thus attacked excessively.

If, with the type of application chosen, only a small quantity of ink has penetrated the holes (this especially applies to horizontal screen printing with hole tenting/spray application) it may be possible to considerably reduce the developing time and the spray pressure can be decreased to about 2 bar [29 psi]. These measures reduce undercut and much higher resolutions can be achieved.

In order to achieve a reliable free development of small holes (≤ 0.4 mm [≤ 0.016 inch]), developer must be transported to the area to be developed as frequently as possible. However, this is not possible without problems in the case of clogged holes. In the first third of the development process the plated-through holes must be sprayed at high pressure and with vertical nozzles in order to render them permeable for the developer solution. In the further course of the development process, a thorough flow through the plated-through holes at low pressure should be effected in order to channel the developer to the ink in the holes. The use of flood nozzles has proven beneficial.



Step 1
90° spray angle to ink-filled hole

Step 2
Opening of the ink-filled hole

Step 3
Free development of holes with flood nozzles

**Fig. 9: Schematic of a free development using flood nozzles**

### Filtration of developer solution

The developer solution is substantially loaded with dissolved ink components, especially in the first developer zone. In order to avoid these ink residues being squeezed onto areas already developed free, the developer should always be directed through a suitable filter system. It is also advisable to duct the developer through a centrifuge in the first developer chamber. This will separate solid components to a large extent and substantially minimise the cleaning effort of the developer.

### Replacement of the developer solution

For a stable process control it is advisable to permanently replace (bleed and feed) the developer solution (controlled by a pcb feeding sensor).

As a rule, the degree of saturation of the developer is stated in percent. The average area developed can be estimated statistically and the quantity of developer solution calculated. In any case, a parallel process control of the loading should be effected. Many years of practical experience have proven that a safe fabrication, irrespective of whether butyl carbitol or carbitol is used as the developer medium, is guaranteed up to a loading degree of 2% of solids. In many cases, a very fast and conclusive check can be effected by determining the specific gravity of the developer solution. Consider that the specific weight is strongly dependent upon the temperature.

The consumption of butyl carbitol/carbitol can be calculated as follows:

| | | |
|---|---|---|
| Calculation bases: | wet coating quantity ink | 100 g/m$^2$ |
| | dry matter ink after predrying (solids content 64%) | 64 g/m$^2$ |
| | percentage share of area to be developed (average value) | 10% |
| This results in an ink quantity to be dissolved of | | 6.4 g/m$^2$ |
| With a loading of 2% this means a theoretically required developer quantity of | | 320 g/m$^2$ |

Thus, about **640 g** of butyl carbitol/carbitol are needed for 1 m$^2$ of double-side coated pcb.

Since the share of the area to be developed is a fluctuating quantity, a "safety reserve" should be provided in order to increase the process safety.

For the process control of aqueous-alkaline developer media, especially pH value determinations, the titration at regular intervals with 0.1 n HCl against phenolphthalein, the conductance value determination but also the colorimetric measurement have proven useful.

When calculating the developer quantity required, it must equally be considered whether also incorrectly coated printed circuit boards are washed in the developer since this process will feed a large quantity of ink to the developer. Generally, washing in the developer is not recommended because the transport system will become badly soiled by such activities. If necessary, the boards to be cleaned off can be collected and washed together prior to the next cleaning cycle.

### Transition zone developer – water rinse

The transition zone between development zone and water rinse is regarded as a very critical area in the entire development process. The most important requirement is that no developer medium must be dragged into the water zone. For this reason, squeegee rollers and air knives are used. The air knives installed must not spray the developer medium into the water zone; the developer solution blown out of the plated-through holes must not be conveyed into the water rinse return either.

The squeegee rollers should never run dry but must always be moistened with developer medium on the developer side. Partially dissolved ink residues are trapped between the free-developed ink edges. By means of the squeegee rollers, these residues are squeezed out and rinsed off together with the excess developer medium. If these residues get stuck on the dry squeegee rollers, they are stamped at random onto another spot of the printed circuit board at a later time and cause voids in the subsequent metal plating processes.

The squeegee rollers at the water rinse inlet are even more critical. These rollers should be sprayed with water in such a manner that a "puddle" forms on the printed circuit boards entering the water zone. This water will wash off dissolved ink particles.

After rinsing the printed circuit boards with demineralised water in a triple cascade, water residues initially are blown off over air knives. The boards are then hot-air dried in a dryer in such a manner that they leave the developer unit without any water residues.

### Developer unit maintenance

The extent of maintenance for the developer unit largely depends on the loading. A complete clean per week is recommended. The transport rollers should be cleaned at least at the change of shifts, the squeegee rollers described above even more frequently, if necessary. In addition, the nozzle assembly should be checked for clogged nozzles at the change of shifts.

## UV bump (optional)

A UV bump (UV post-crosslinking at 500-2000 mJ/cm²) before and/or after thermal curing has a positive effect with regard to

- the ionic contamination by HAL fluxing agents
- the prevention of whiteish discolourations by subsequent cleaning processes
- the prevention of spotting in subsequent finish-processes.

Moreover, the UV bump **before** final curing offers the advantage that condensate formation during final curing is reduced.

→ Ascertain the need for a UV bump by means of pretests as the resistance in subsequent chemical processes and the adhesion of subsequent coatings may be affected both positively and negatively.

→ Please consider the recommendations on the UV dosage in the respective process data sheet.

With conveyorised final curing ovens the UV bump can be integrated in-line directly behind the oven. A regular control of the lamp power and age by means of an hourmeter are imperative.

→ Do not carry out a UV bump when using flexible **Elpemer**® solder resists as these inks become brittle as a result.

# Visual inspection

Before the curing process a visual control of the free development, particularly from the plated-through holes, registration accuracy and the presence of voids should be carried out. At that time printed circuit boards that have one or more of the above imperfections can be washed off in 5% NaOH or KOH or special stripper **HP 5707**. After final curing, the ink film is so stable that stripping is practically no longer possible.

# Thermal curing (final curing)

In the final curing process the chemical cross-linking of the ink components is carried out. This process is responsible for the mechanical, chemical and electrical properties of the solder mask. Therefore, the observance of the parameters in accordance with the specifications in the Process data sheet is imperative.

The quality is irrespective of whether the final curing is effected in a box oven or a conveyorised convection oven.

It is important to observe that

- the temperature specified in the Process data sheet is reached evenly in the entire oven chamber
- particularly an optimum control of circulating and exhaust air in the initial phase of the thermal curing process is ensured in order to avoid condensation (see also the advice on “Condensate” in the Section ”Predrying”).
- the net burn-in time does not begin until the printed circuit boards have reached the specified temperature (object holding time); attention must be paid to the fact that the printed circuit board thickness substantially influences the heat-up time.

For a trouble-free handling, printed circuit boards cured in conveyorised ovens should be cooled down to temperatures slightly above room temperature at the end of the burn-in process.

In the case of stationary ovens, it is advisable to lock the oven during the entire burn-in process: Within the scope of quality assurance, this will ensure that the oven cannot be opened in between which would mean an uncontrolled reduction of the specified burn-in-temperature.

**A ramp formed curing curve must be adjusted when filling via holes in order to expel possible trapped residual solvents, e. g. 1-3 h at 110 °C [230 F]; subsequent final curing according to the Process data sheet.**
**The curing conditions also depend on the used quantity of solder resist. Perform pretests to determine the optimum parameters.**

Basically, a pure IR final cure is possible, but requires an exact process review. Our Application Technology Department (ATD) will gladly assist you in case of any queries.

➔ Determine suitable IR curing parameters by means of pretests while ensuring that cross-linking is completed and the final properties are achieved.

# Literature

In addition to the recommendations given in this technical report, we can provide technical papers and information sheets written and compiled by members of our staff which are available for download on our website **http://www.peters.de.** We also recommend for further reading:

**Dr. Manfred Suppa, Publisher Werner Peters: “Conformal Coatings for Electronics Applications”**, 1st edition 2012, Lackwerke Peters GmbH + Co KG, ISBN 978-3-00-039856-8

# Disclaimer

Any questions?

We would be pleased to offer you advice and assistance in solving your problems. Samples and technical literature are available upon request.

**Lackwerke Peters GmbH & Co. KG**
Hooghe Weg 13, 47906 Kempen, Germany

Internet: www.peters.de
E-Mail: peters@peters.de

Phone +49 2152 2009-0
Fax +49 2152 2009-70

peters
Coating Innovations
for Electronics