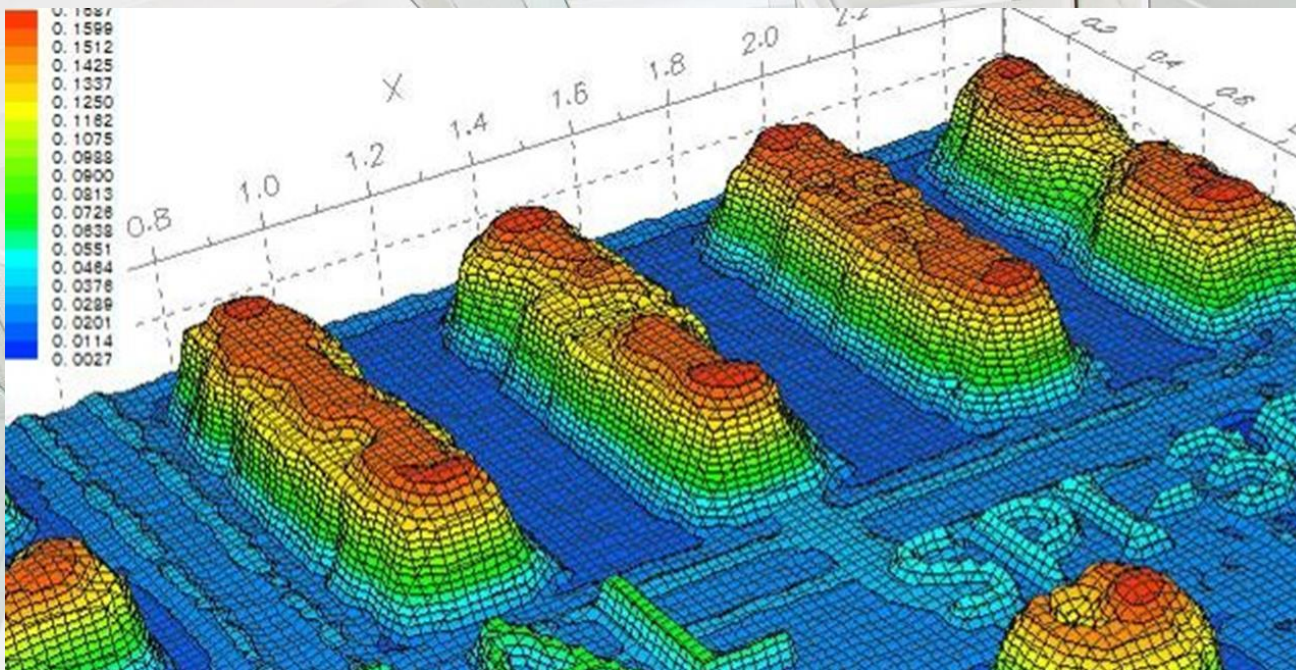


Litavimo pastos užtepimo defektai ir  
būdai jų išvengti.

TESTING of SOLDER PASTE PRINTING



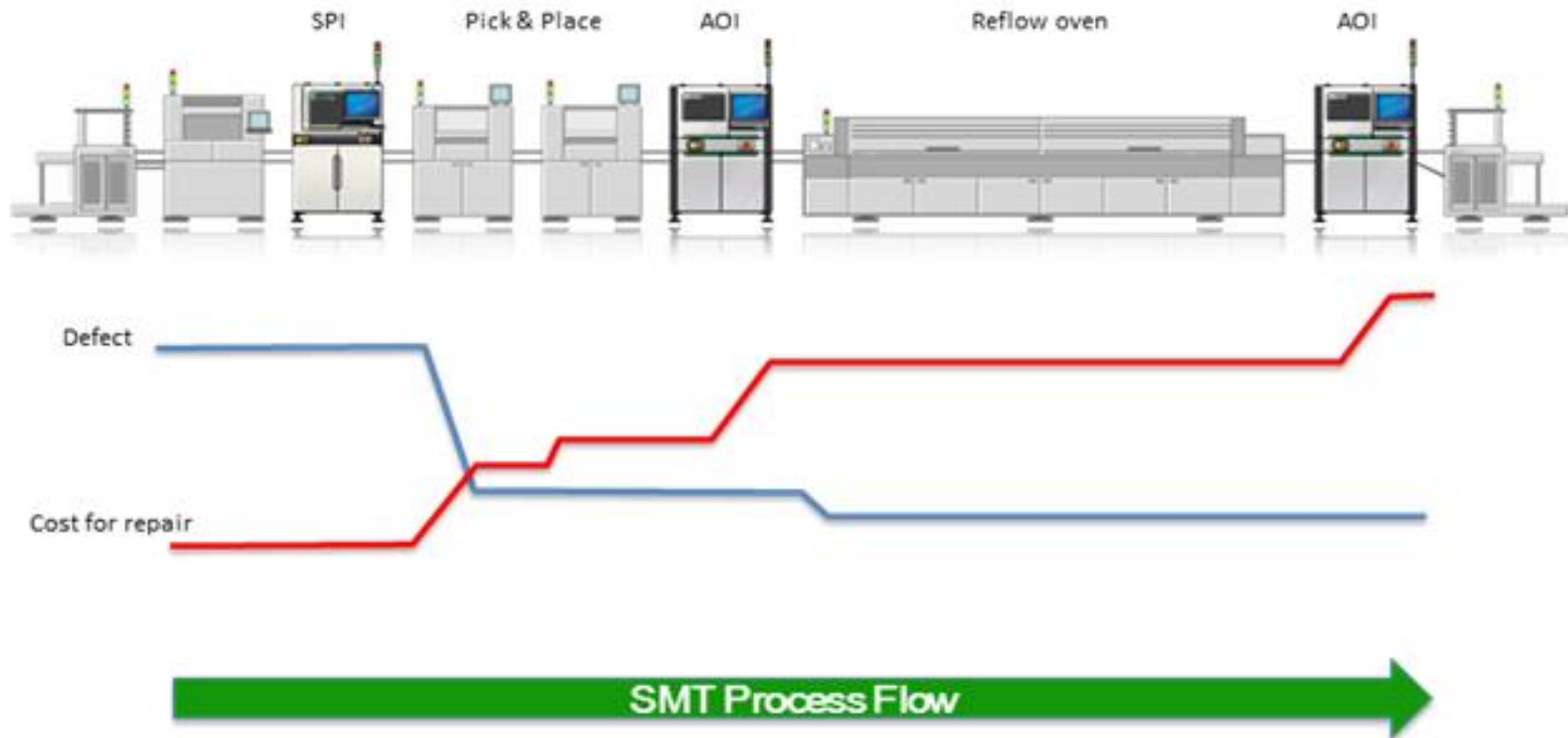
# Ivadas

# Introduction

One of the most important parts of the surface mount assembly process is the application of solder paste to the printed circuit board (PCB). The aim of this process is to accurately deposit the correct amount onto each of the pads to be soldered. This is achieved by screen-printing the solder paste through a stencil or foil but also can be applied by jet printing. It is widely believed that this part of the process, if not controlled correctly, accounts for the majority of assembly defects.

# Ivadas

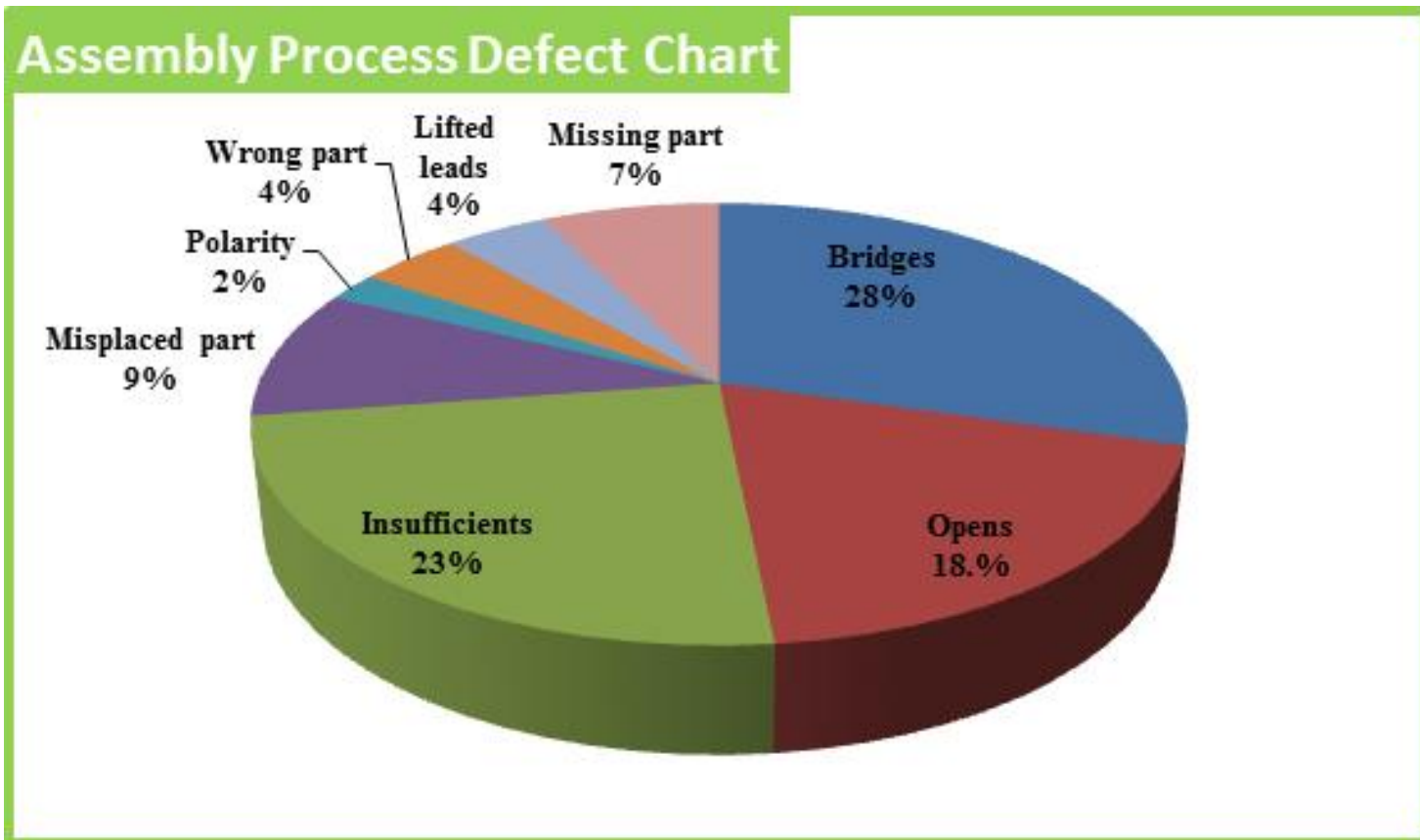
## Introduction



# Ivadas

## Introduction

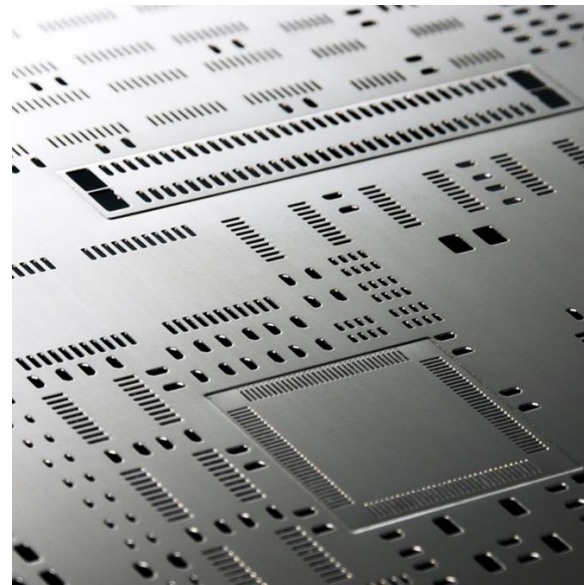
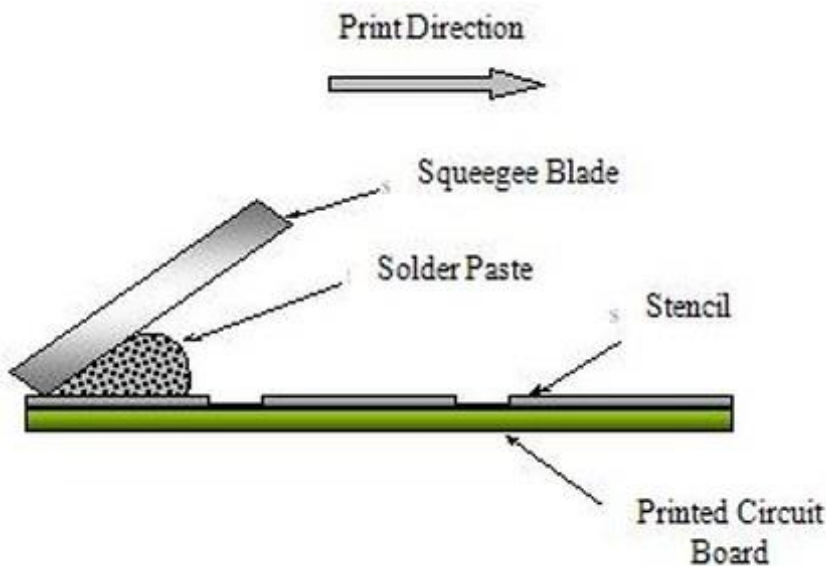
Viso gamybos proceso defektai





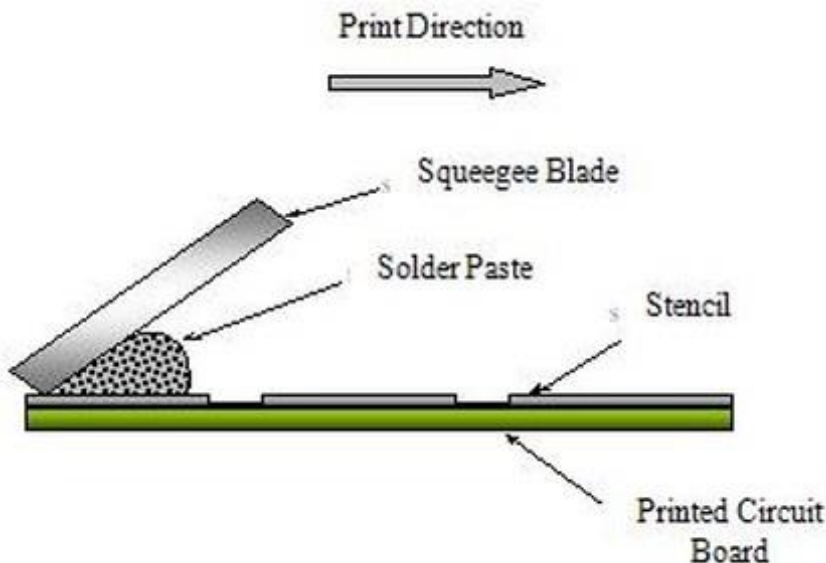
# Method of applying solder paste to PCB

The most common method of applying solder paste to a PCB using a stencil printer is squeegee blade printing – see images below. The squeegees are the tools used to apply the necessary force required to move the solder paste across the stencil and on to the PCB. They are usually made from metal but can also be made from polyurethane.



# Method of applying solder paste to PCB

The most common method of applying solder paste to a PCB using a stencil printer is squeegee blade printing – see images below. The squeegees are the tools used to apply the necessary force required to move the solder paste across the stencil and on to the PCB. They are usually made from metal but can also be made from polyurethane.



# The key parameters of an effective solder paste printing process are as follows:

- Squeegee speed
- Squeegee pressure
- Stencil separation speed
- Stencil cleaning
- Stencil and squeegee condition
- PCB support
- Print stroke
- Type, storage and handling
- Inspection (2D/3D)

# Squeegee speed



The speed of travel of the squeegee determines how much time is available for the solder paste to “roll” into the apertures of the stencil and onto the pads of the PCB. Typically a setting of **25mm per second** is used but this is variable depending on the size of the apertures within the stencil and the solder paste used.



# Squeegee pressure

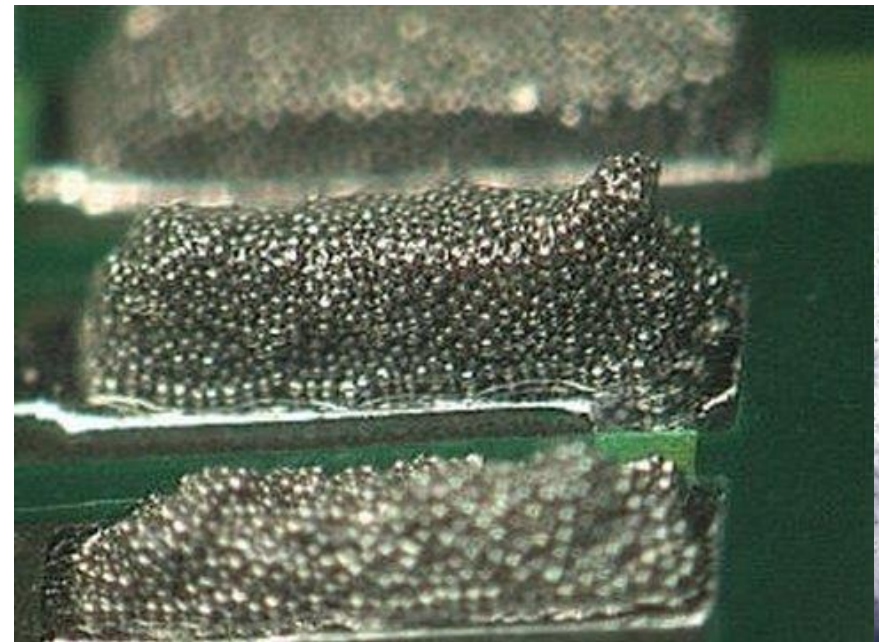
During the print cycle it is important to apply sufficient pressure across the entire length of the squeegee blade to ensure a clean wipe of the stencil.

Too little pressure can cause “**smearing**” of the paste on the stencil, **poor deposition**, and the **incomplete transfer** to the PCB.

Too much pressure can cause “**scooping**” of the paste from larger apertures, excess wear on the stencil and squeegees, and may cause “**bleeding**” of the paste between the stencil and PCB. A typical setting for the squeegee pressure is **0.5Kg of pressure per 25mm** of squeegee blade.

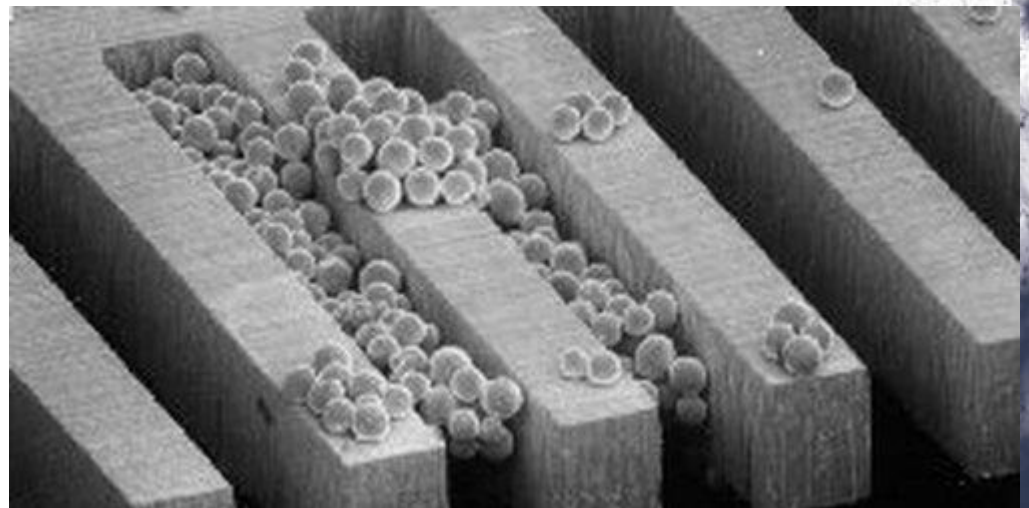
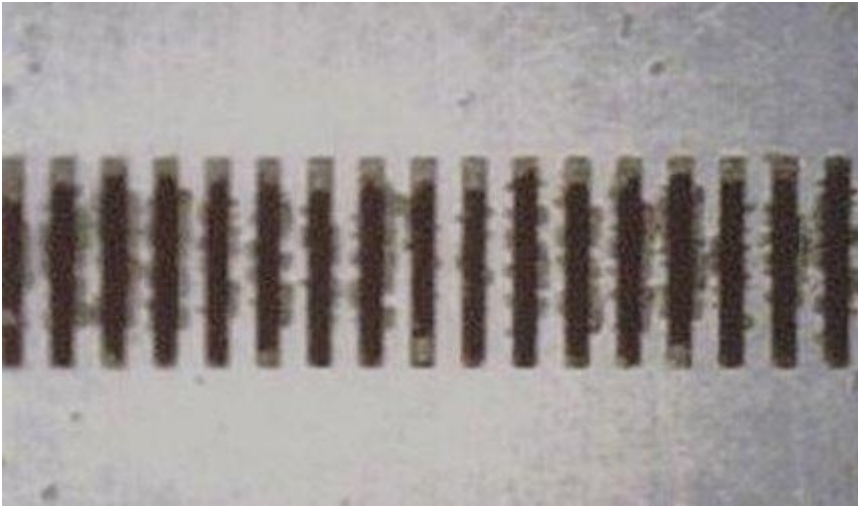
# Stencil separation speed

This is the speed at which the PCB separates from the stencil after printing. A speed setting of up to **3mm per second** should be used and is governed by the size of the apertures within the stencil. If this is too fast it will cause the solder paste to not fully release from the apertures and the formation of high edges around the deposits also known as “**dog-ears**” which can be seen below.



# Trafareto valymas

## Stencil Cleaning



# Trafareto valymas

## Stencil Cleaning

The stencil must be cleaned regularly during use which can be done either manually or automatically.

Many of the automatic printing machines have a system that can be set to clean the stencil after a fixed number of prints using lint-free material applied with a cleaning chemical such as **IPA**.

The system performs two functions, the first being the cleaning of the underside of the stencil to stop smudging, and the second is the cleaning of the apertures using **vacuum to stop blockages**.

# Trafareto ir menteles būklė

## Stencil and squeegee condition



Both stencils and squeegees need to be carefully stored and maintained as any mechanical damage to either can lead to undesired results.

Both should be checked before use and thoroughly cleaned after use, ideally using a automated cleaning system so that any solder paste residue is removed.

If any damage is noticed to squeegees or stencils they should be replaced to ensure a reliable and repeatable process.



# PCB atramèlès

## PCB support

This is an important factor to ensure the PCB is held flat against the stencil during the printing process.

If the PCB is not fully supported it can lead to printing defects such as a **poor paste deposit and smudging**.

PCB supports are generally supplied with printing machines which are a fixed height and have programmable positions to ensure a consistent process.

There are also **adaptable PCB supports** available of varying designs which mould themselves to the PCB and are useful for double sided assemblies.



# Mentelès prispaudimo jèga

## Print stroke



This is the distance the squeegee travels across the stencil and is recommended to be a minimum of **20mm past the furthest aperture.**

The distance past the furthest aperture is important to allow enough space for the paste to roll on the return stroke as it is the rolling of the solder paste bead that generates the downward force that drives the paste into the apertures.

# Litavimo pastos tipas, laikymas, apdorojimas

## Solder paste type, storage and handling

Solder paste is essentially powdered solder suspended in a thick medium called **flux**. The flux acts as a temporary adhesive, holding the components in place until the soldering process melts the solder and forms the electrical/mechanical connection.

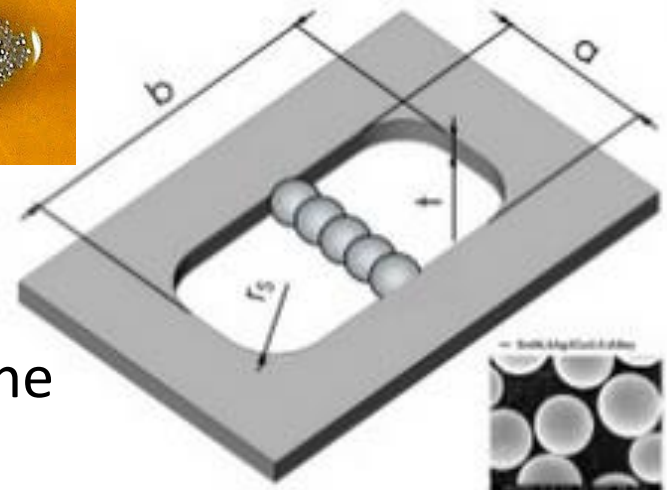
| Particle size in microns | Particle type |
|--------------------------|---------------|
| 75-45                    | 2             |
| 45-25                    | 3             |
| 38-20                    | 4             |
| 25-15                    | 5             |
| 15-5                     | 6             |

# Litavimo pastos tipas, laikymas, apdorojimas

## Solder paste type, storage and handling



There is a '**5 ball rule**' which says ideally a minimum of 5 solder particles should span the width of the smallest aperture.



5 Solder Ball Rule

# Litavimo pastos tipas, laikymas, apdorojimas

## Solder paste type, storage and handling



Solder paste should be refrigerated while being stored to maintain its shelf life but must be brought to room temperature for a minimum of **eight hours before use** to maintain quality.

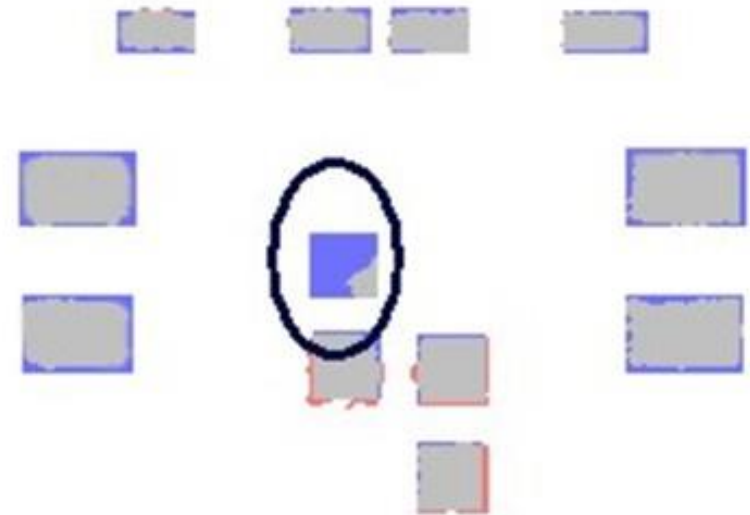
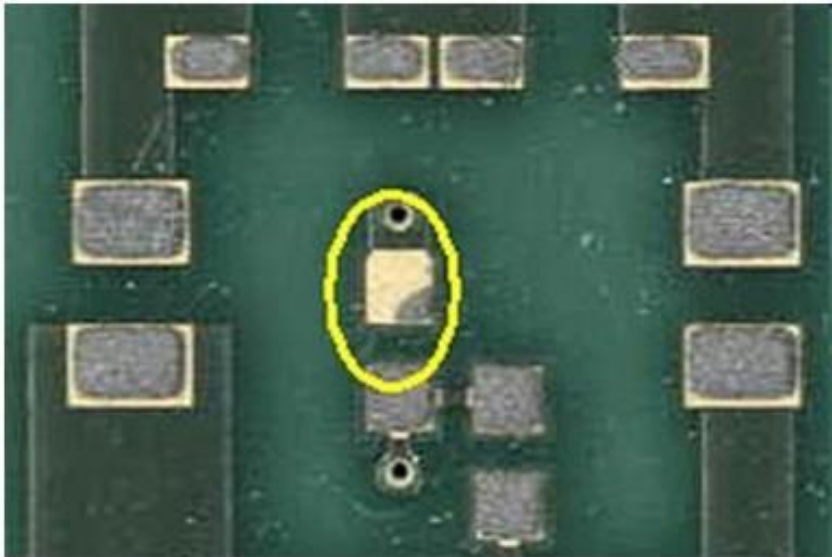
The solder paste should be mixed before use to ensure even distribution of any separated material throughout the paste. **Mixing can be carried out manually or automatically for a duration of between one to three minutes.**

As a general rule, solder paste that has been in use for **more than 8 hours should be disposed of**. Solder paste which has been in use for up to 4 hrs can be stored for up to 24 hours in a sealed container at room temperature before being re-used. The working environment (ambient temperature and relative humidity) will affect the performance and so to be sure of the condition of the solder paste a simple coalescence test can be carried out.



# Inspekcija (2D/3D)

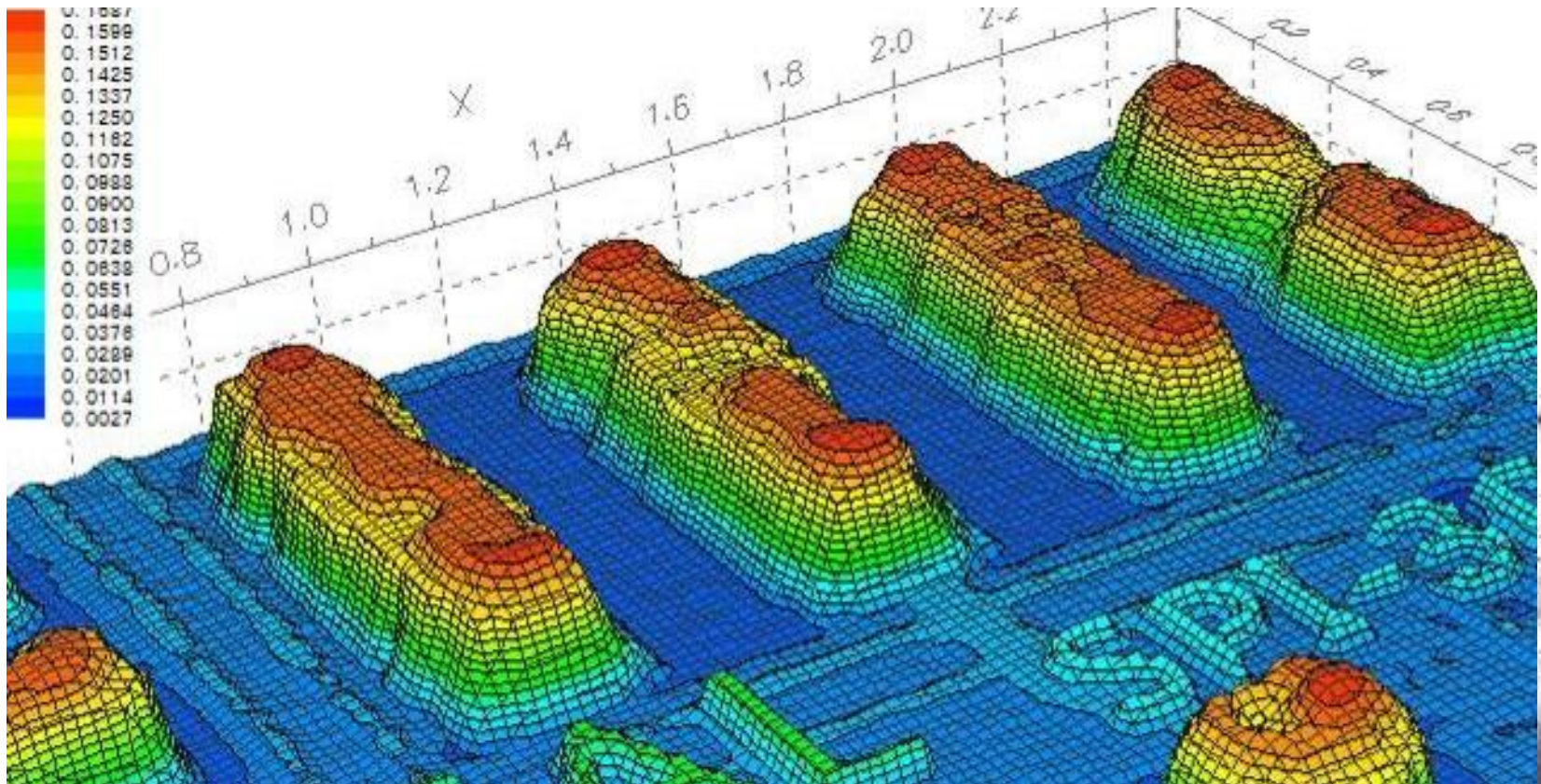
## Inspection (2D/3D)



Example of fault found by 2D inspection

# Inspekcija (2D/3D)

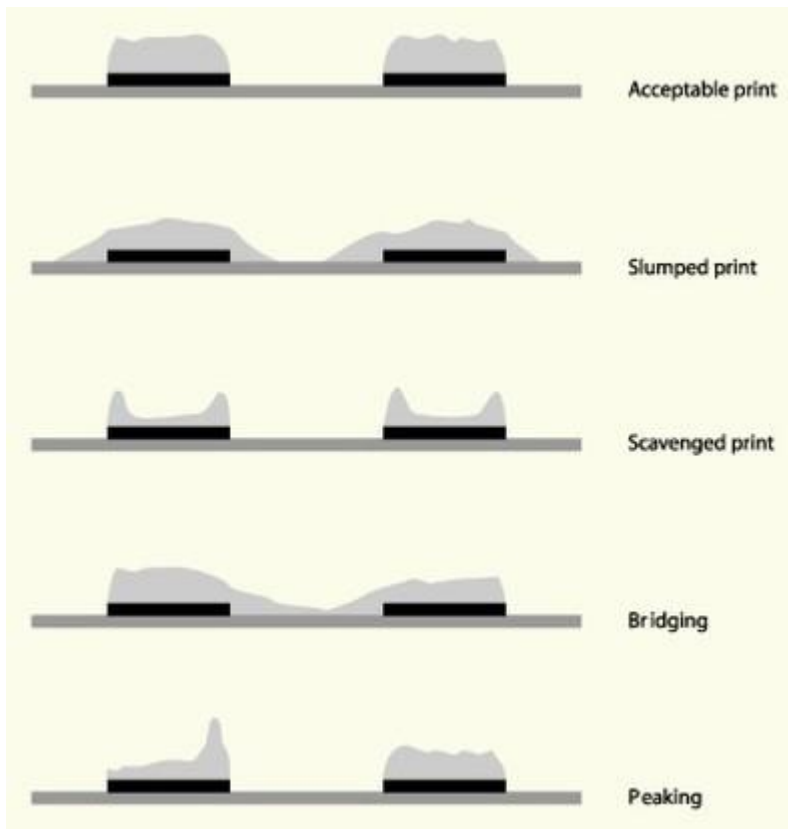
## Inspection (2D/3D)



Example of fault found by 3D inspection

# Inspekcija (2D/3D)

## Inspection (2D/3D)



Example of fault found by 2D inspection This is the target condition.

The 'Slumped print' typically occurs in a process that is run at temperatures above the recommend level.

The 'Scavenged print' result is what can be seen if the squeegee pressure is set too high and 'Scooping' occurs.

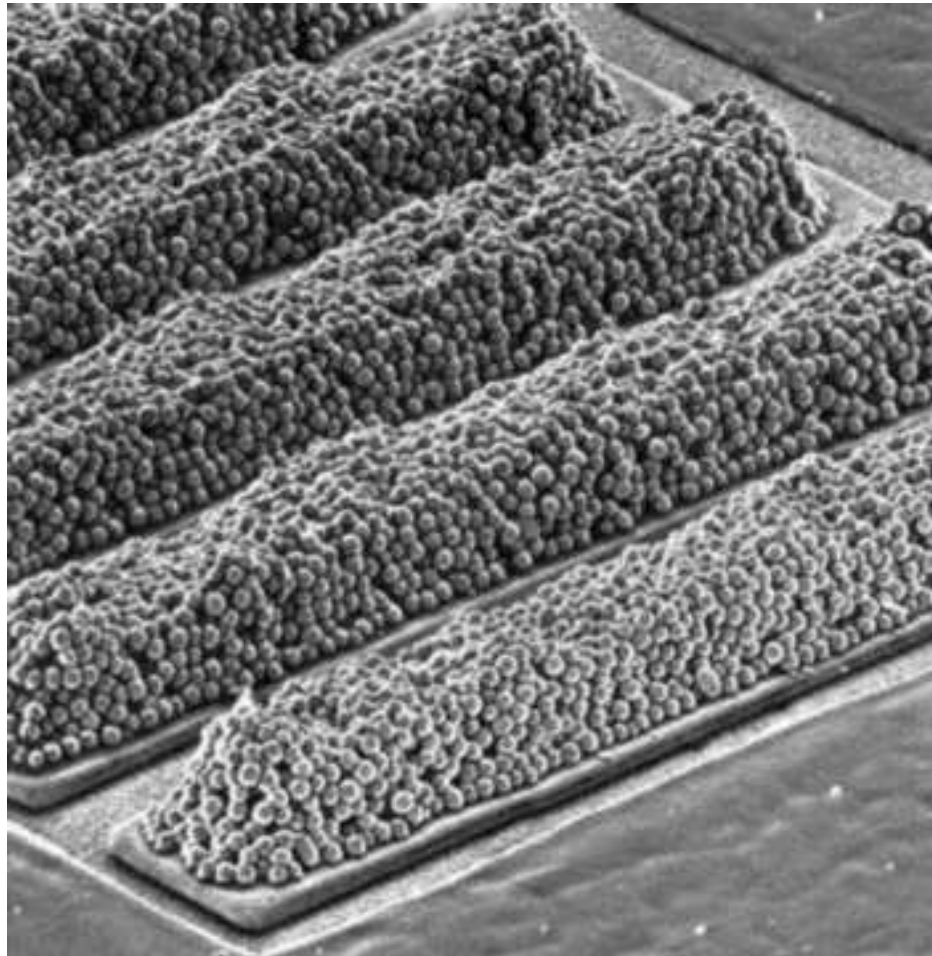
'Bridging' can be as a result of poor board support or stencil condition/cleanliness.

'Peaking' is generally noticed when the stencil separation speed is set too high.



# Faulty application of solder paste

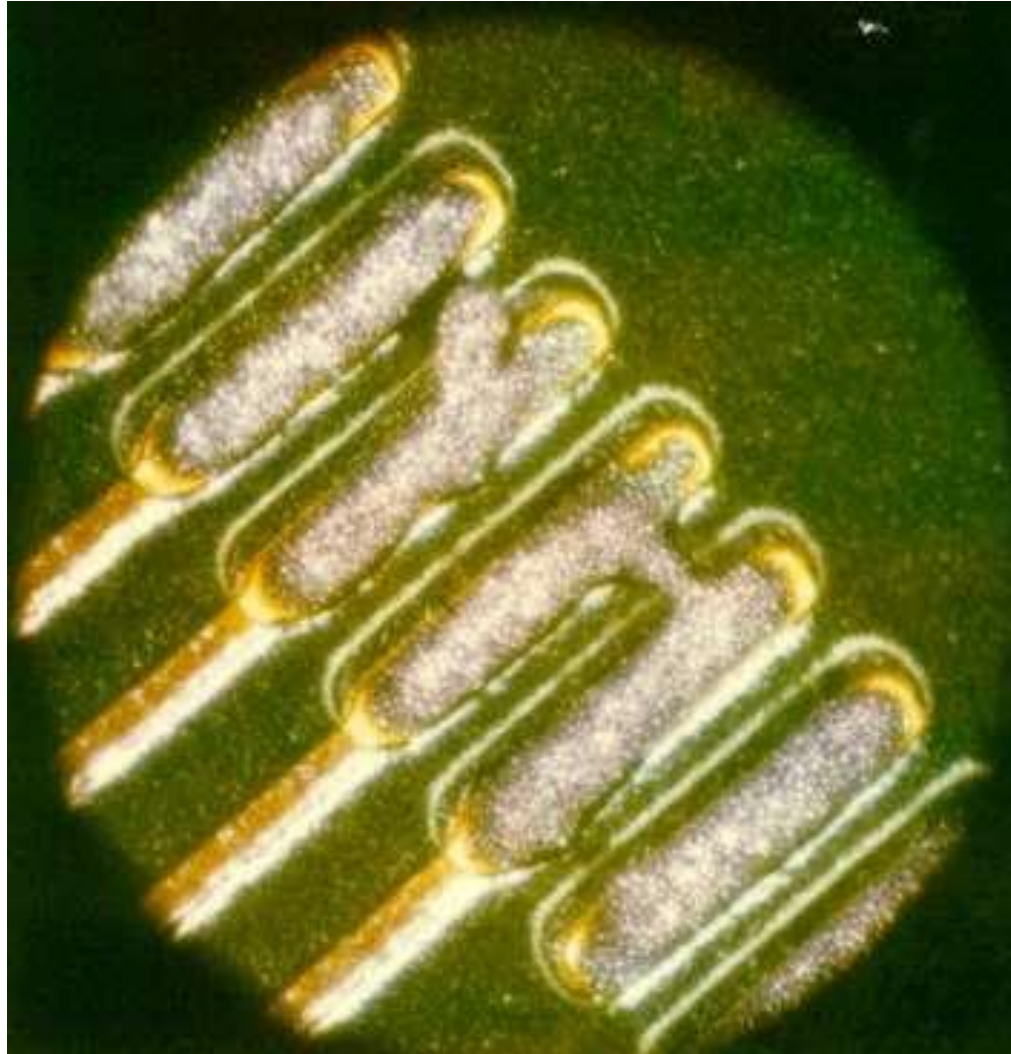
Idealus pastos užnešimo rezultatas  
Best possible solder paste application



# Faulty application of solder paste

Pastos tilteliai

Bridge due to solder paste





# Faulty application of solder paste

Pastos tilteliai

Bridge due to solder paste

## Description:

Bridging between the solder paste depots. There is a high risk that bridges also form between the contacts during soldering. Rework necessary.

## Causes/Remedy:

- unsuitable solder paste
- processing conditions ignored: temperature, best-before date exceeded
- poor printing process
- lifting-off step takes solder paste with it
- contaminated stencil

**Source:** visual, optical inspection - general view

# Faulty application of solder paste

Nevienodas pastos kiekis

Uneven application of solder paste



# Faulty application of solder paste



Nevienodas pastos kiekis

Uneven application of solder paste

## **Description:**

Uneven application of solder paste. The amount of solder paste applied is clearly lower on the right side. This risks on non-soldering or insufficient solder. Rework necessary.

## **Causes/Remedy:**

- unsuitable solder paste

- processing conditions ignored: temperature, best-before date exceeded

- poor printing process

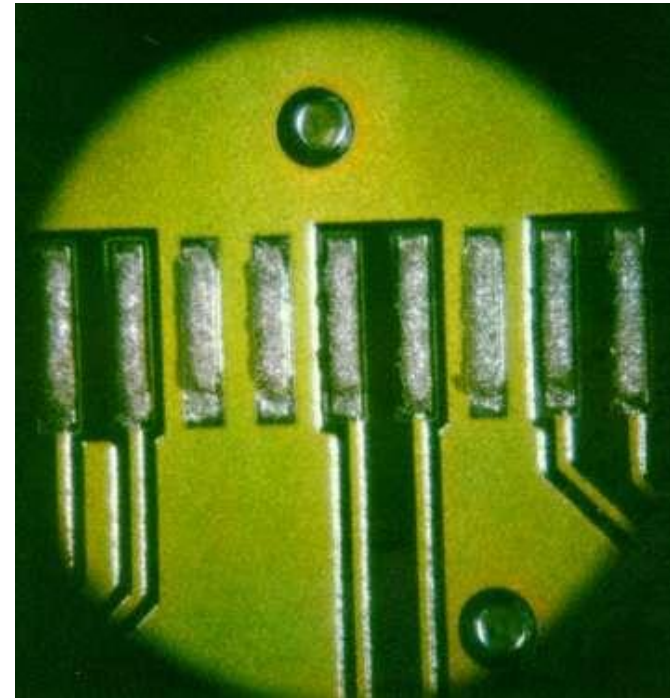
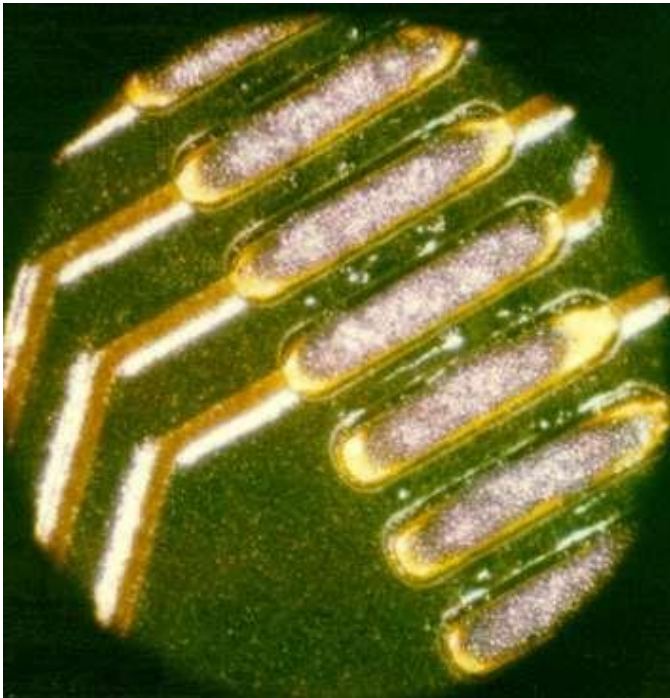
- lifting-off step takes solder paste with it

- contaminated stencil

**Source:** visual, optical inspection - general view

# Faulty application of solder paste

Nevienodas pastos užnešimas  
Uneven printing of solder paste



# Faulty application of solder paste



Nevienodas pastos užnešimas  
Uneven printing of solder paste

## Description:

Again uneven printing of solder paste, but clearly less risky as far as insufficient solder or opens are concerned. Acceptable to a certain extent

## Causes/Remedy:

- unsuitable solder paste
- processing conditions ignored: temperature, best-before date exceeded
- poor printing process
- lifting-off stop takes solder paste with it

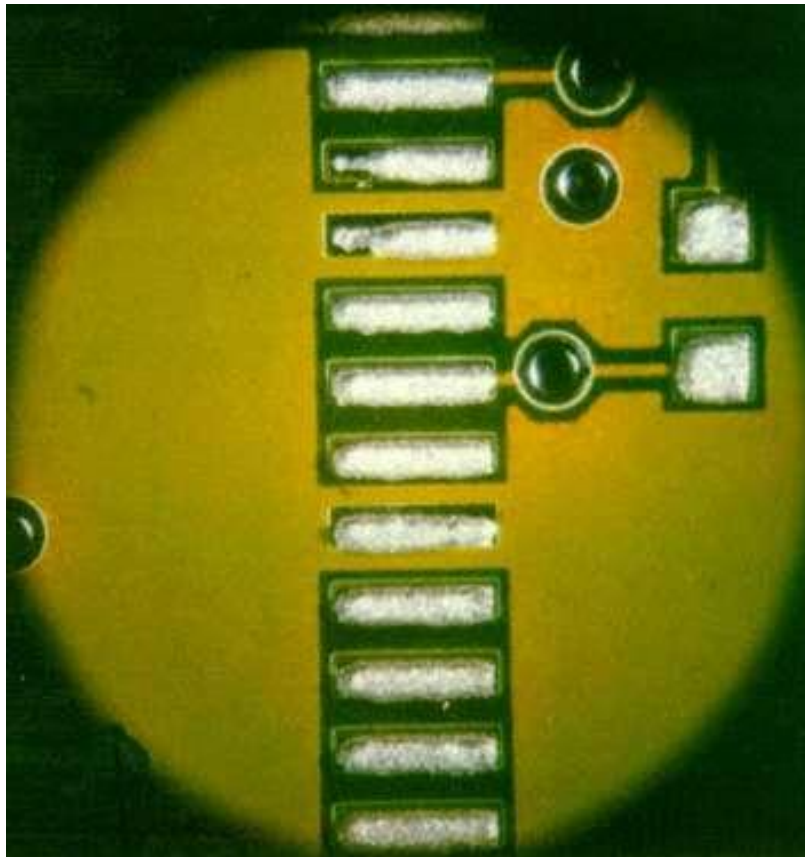
**Source:** visual, optical inspection - general view



# Faulty application of solder paste

Pastos pakėlimas

Pulling up of solder paste



# Faulty application of solder paste

## Pastos pakėlimas

### Pulling up of solder paste

#### Description:

Pulling up of solder paste, forming of studs. Acceptable to a certain extent, as long as no bridges generate or the amount of solder paste is drastically reduced.

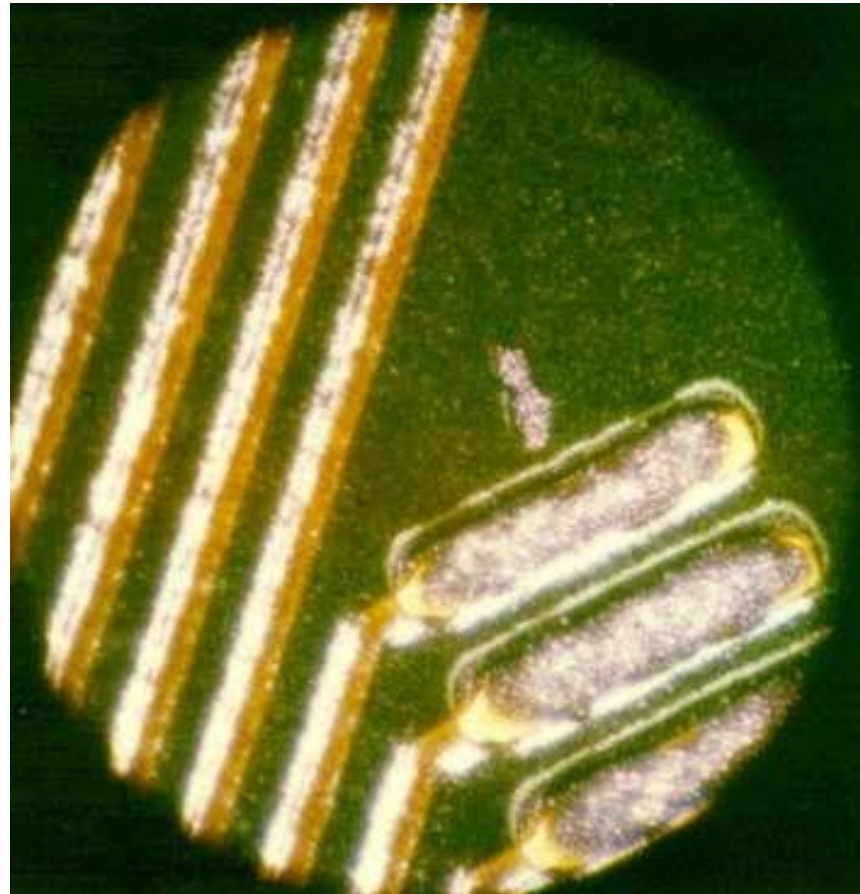
#### Causes/Remedy:

- unsuitable solder paste
- processing conditions ignored: temperature, best-before date exceeded
- poor printing process
- lifting-off stop takes solder paste with it

**Source:** visual, optical inspection - general view

# Faulty application of solder paste

Išteptas PCB paviršius pasta  
Smeared solder paste



# Faulty application of solder paste



## Išteptas PCB paviršius pasta Smeared solder paste

### Description:

Smeared solder paste. Solder paste is evident on the solder stop on the side of the printing area. Danger of microballs forming

### Causes/Remedy:

unsuitable solder paste

processing conditions ignored: temperature, best-before date exceeded

poor printing process

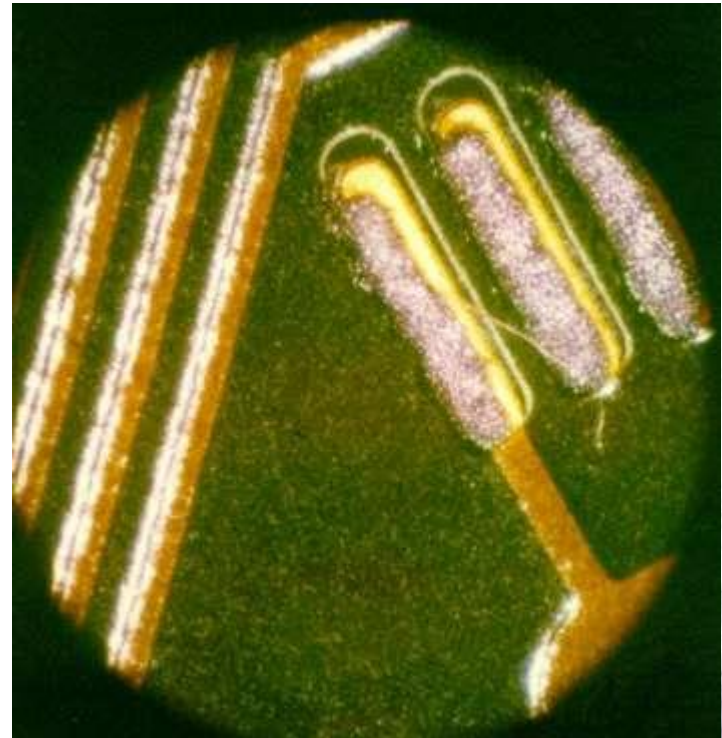
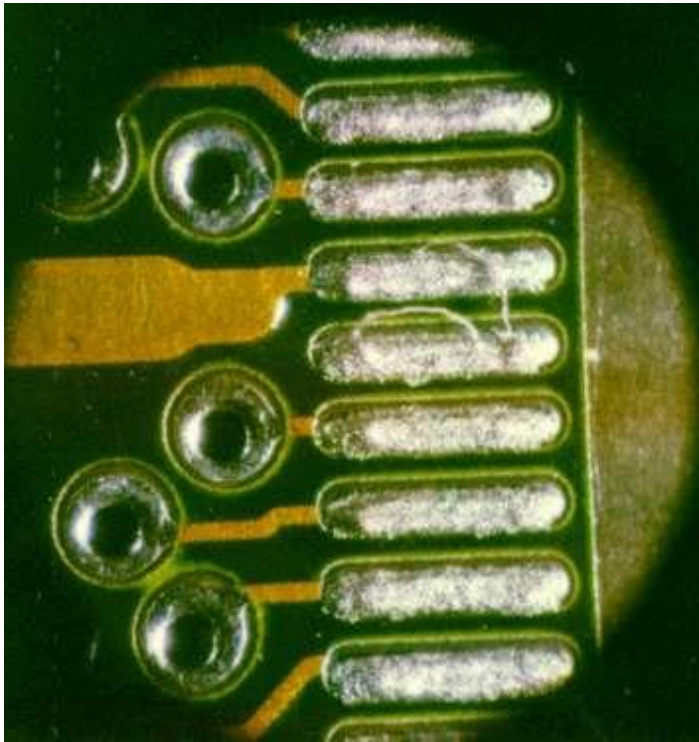
lifting-off stop takes solder paste with it

**Source:** visual, optical inspection - general view

# Faulty application of solder paste

Nešvari pasta

**Contamination of the solder paste depot**





# Faulty application of solder paste

Nešvari pasta

## Contamination of the solder paste depot

### Description:

Contamination of the printed solder paste depot. Textile fibres are lying on the solder paste. It could be proven that the fibres stem from textile material used to clean the stencil.

### Causes/Remedy:

unsuitable solder paste

processing conditions ignored: temperature, best-before date exceeded

poor printing process

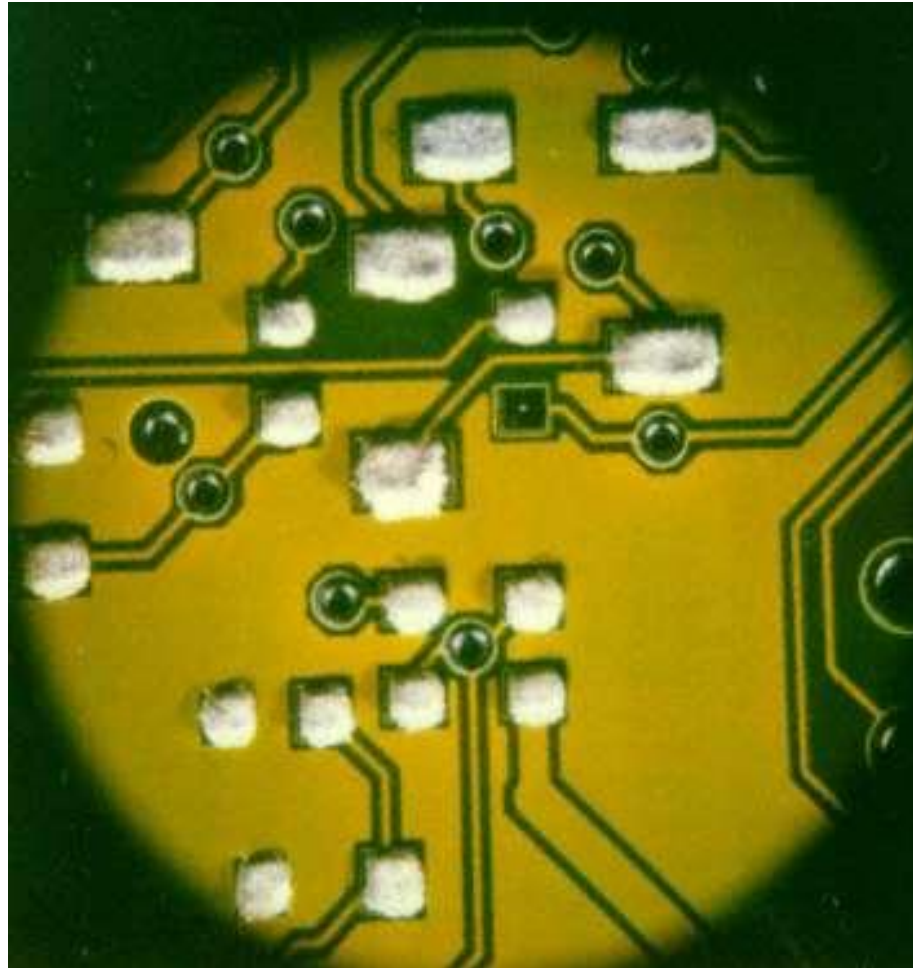
lifting-off step takes solder paste with it

**Source:** visual, optical inspection - general view

# Faulty application of solder paste

Pastos perteklius

**Excess solder paste**



# Faulty application of solder paste

Pastos perteklius

## Excess solder paste

### Description:

Excess solder paste on the pads (excess printing). The use of unsuitable printing stencils led to this excess printing. In some rare cases this is even intended for special types of components. For fine-pitch components there is the danger of solder bridges forming, however.

### Causes/Remedy:

- unsuitable solder paste
- processing conditions ignored: temperature, best-before date exceeded
- poor printing process
- lifting-off step takes solder paste with it

**Source:** visual, optical inspection - general view