Design-Rules and important information around the topic Solder mask

Albert Schweitzer
Fine Line Gesellschaft für Leiterplattentechnik mbH
Itterpark 4, D-40724 Hilden

16.09.2015 Vers. 1.0
Table of Content

1. General
2. IPC Standard
3. Application process
4. Thickness of Solder mask
5. Min. Solder mask web
6. Definition SMD / NSMD Pads
7. Why NSMD Pads?
8. Parameter NSMD Pads
9. Others
1 General
Solder mask is a protective layer for printed circuit boards with the following essential functions:

- Prevent of short circuits (solder bridges) during soldering process
- Reduction of solder past consumption during Solder process
- Additional insulation of the components from PCB
- Protection of the PCB against mechanical and chemical attack (corrosion)
- Increasing the breakdown voltage
- Prevent growth of metal whiskers
2 IPC Standard
In IPC standard “IPC-SM-840”, the following requirements are defined:

- Visual requirements
- Adhesiveness
- Soldering resistance
- Cleaning and flux resistance
- Layer thickness
- Curing
The test methods and conditions in IPC-SM-840C are based on end use and environmental reliability requirements for two classes of user:

*Class 2 is equivalent to Class T;*  
*Class 3 is the equivalent of Class H.*

**T** – Telecommunication (includes computers, telecommunication equipment, sophisticated business machines, instruments, and certain non-critical military applications.) Solder mask on boards in this class is suitable for high performance commercial and industrial products in which extended performance life is required, but for which interrupted service is not life-threatening.
**IPC-SM-840**

**H** – High Reliability/Military (includes that equipment where continued performance is critical, equipment down-time cannot be tolerated and/or the equipment is a life support item). Solder mask on boards of this class is suitable for applications where high levels of assurance are required and uninterrupted service is essential.
3 Application process
Solder mask Types

- Direct Imaging
  - Screen Print (UV or Thermal cure)
  - Ink Jet
- Indirect Imaging
  - LDI (Laser direct Image)
  - LPISM (Liquid Photoimageable Solder Mask)
  - Dry Film

Source: Electra
LPISM Application Method

We will concentrate on the Liquid Photoimageable Solder Mask (LPISM) Method. There are currently four main categories of LPISM available:

- Screen print
- Curtain coat
- Electrostatic spray
- Air spray
Air spray

Double sided scree print

Curtain coat

Electrostatic spray

Air spray
The curtain coating principle is used in various industries. It can apply liquid material at very high speeds with low material loss. Curtain coating has become more widely used in the circuits industry as PCBs have become more complex, with higher track densities and narrower track widths. A vertical falling curtain of coating material is produced by pumping the feedstock through a slit on the underside of a reservoir. The substrate to be coated is moved briskly through the resulting liquid curtain, collecting a thin and even deposit on the way. It is then passed on to separate machinery for baking.
Curtain coat
4 Thickness of Solder Mask
Thickness of Solder Mask

- Screen Print
- Curtain coat
- Spray

Source: Electra
Thickness of Solder Mask

A = Thickness above laminate
B = Thickness above line edge
C = Thickness above copper

<table>
<thead>
<tr>
<th>Copper Thickness</th>
<th>A</th>
<th>B</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>35µm</td>
<td>&lt;=50µm</td>
<td>5µm</td>
<td>15µm</td>
<td>10 - 25µm</td>
</tr>
<tr>
<td>50µm</td>
<td>&lt;=60µm</td>
<td>5µm</td>
<td>15µm</td>
<td>10 - 25µm</td>
</tr>
<tr>
<td>75µm</td>
<td>&lt;=75µm</td>
<td>5µm</td>
<td>15µm</td>
<td>10 - 25µm</td>
</tr>
<tr>
<td>105µm</td>
<td>&lt;=100µm</td>
<td>5µm</td>
<td>20µm</td>
<td>10 - 30µm</td>
</tr>
<tr>
<td>140µm</td>
<td>&lt;=150µm</td>
<td>5µm</td>
<td>20µm</td>
<td>10 - 35µm</td>
</tr>
</tbody>
</table>

Values for Screen Print process
Thickness of Solder Mask

IPC-SM-840C don‘t describe the thickness of the Solder Mask. It merely requires that the solder mask having a dielectric strength of 500VDC / 0,025mm.

The thickness of the solder mask can only predicted accurately in limits and depends on used process. It differs also from manufacturer to manufacturer and used chemistry.

In case a customer ask for a specific thickness, it must be stated in order documents.
5 Min. Solder Mask dam
Solder Mask dam

Min. Solder Mask Dam / Min. Lötstoppmaskensteg

Solder Mask Clearance / Freistellung

Min. Solder Mask Dam / Min. Lötstoppmaskensteg = > 150µm

Solder Mask Clearance / Freistellung = > 75µm

Solder mask dam is different for non-green SM

Source: Fineline
Misregistration

Source: Fineline
Solder mask defined (SMD) and non solder mask defined (NSMD) Pads
Definition Solder Mask defined (SMD) and non Solder Mask defined (NSMD) Pads

The difference between:

**Solder Mask Defined, SMD Pads:**

**Non Solder Mask Defined, NSMD Pads:**
Difference between SMD and NSMD Pads

Source: TopLine
Typically dimensions of SMD Pad

<table>
<thead>
<tr>
<th>Pitch</th>
<th>Ball Durchmesser</th>
<th>Ø Lötbare Pad</th>
<th>Ø Pad Kupfer</th>
<th>Leiterbahnbreite</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5mm</td>
<td>0.76mm</td>
<td>30mil</td>
<td>0.61mm</td>
<td>0.25mm</td>
</tr>
<tr>
<td>1.27mm</td>
<td>0.76mm</td>
<td>30mil</td>
<td>0.61mm</td>
<td>0.25mm</td>
</tr>
<tr>
<td>1.00mm</td>
<td>0.634mm</td>
<td>25mil</td>
<td>0.51mm</td>
<td>0.2mm</td>
</tr>
<tr>
<td>1.00mm</td>
<td>0.5mm</td>
<td>19.7mil</td>
<td>0.4mm</td>
<td>0.2mm</td>
</tr>
<tr>
<td>0.8mm</td>
<td>0.5mm</td>
<td>19.7mil</td>
<td>0.4mm</td>
<td>0.15mm</td>
</tr>
<tr>
<td>0.8mm</td>
<td>0.45mm</td>
<td>17.7mil</td>
<td>0.36mm</td>
<td>0.15mm</td>
</tr>
<tr>
<td>0.8mm</td>
<td>0.4mm</td>
<td>15.7mil</td>
<td>0.32mm</td>
<td>0.15mm</td>
</tr>
<tr>
<td>0.75mm</td>
<td>0.35mm</td>
<td>13.8mil</td>
<td>0.28mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.65mm</td>
<td>0.35mm</td>
<td>13.8mil</td>
<td>0.28mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.50mm</td>
<td>0.3mm</td>
<td>11.8mil</td>
<td>0.24mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.40mm</td>
<td>0.25mm</td>
<td>9.8mil</td>
<td>0.20mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.30mm</td>
<td>0.2mm</td>
<td>7.9mil</td>
<td>0.15mm</td>
<td>0.10mm</td>
</tr>
</tbody>
</table>

Source: TopLine
Typically dimensions of NSMD Pad

<table>
<thead>
<tr>
<th>Pitch</th>
<th>Ball Durchmesser</th>
<th>Ø Lötbares Pad</th>
<th>Ø Pad Kupfer</th>
<th>Leiterbahnbreite</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5mm</td>
<td>0.76mm</td>
<td>30mil</td>
<td>0.63mm</td>
<td>0.20mm</td>
</tr>
<tr>
<td>1.27mm</td>
<td>0.76mm</td>
<td>30mil</td>
<td>0.63mm</td>
<td>0.20mm</td>
</tr>
<tr>
<td>1.00mm</td>
<td>0.634mm</td>
<td>25mil</td>
<td>0.63mm</td>
<td>0.15mm</td>
</tr>
<tr>
<td>1.00mm</td>
<td>0.5mm</td>
<td>19.7mil</td>
<td>0.45mm</td>
<td>0.15mm</td>
</tr>
<tr>
<td>0.8mm</td>
<td>0.5mm</td>
<td>19.7mil</td>
<td>0.45mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.8mm</td>
<td>0.45mm</td>
<td>17.7mil</td>
<td>0.40mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.8mm</td>
<td>0.4mm</td>
<td>15.7mil</td>
<td>0.35mm</td>
<td>0.12mm</td>
</tr>
<tr>
<td>0.75mm</td>
<td>0.35mm</td>
<td>13.8mil</td>
<td>0.30mm</td>
<td>0.10mm</td>
</tr>
<tr>
<td>0.65mm</td>
<td>0.35mm</td>
<td>13.8mil</td>
<td>0.30mm</td>
<td>0.10mm</td>
</tr>
<tr>
<td>0.50mm</td>
<td>0.3mm</td>
<td>11.8mil</td>
<td>0.25mm</td>
<td>0.10mm</td>
</tr>
<tr>
<td>0.40mm</td>
<td>0.25mm</td>
<td>9.8mil</td>
<td>0.20mm</td>
<td>0.10mm</td>
</tr>
<tr>
<td>0.30mm</td>
<td>0.2mm</td>
<td>7.9mil</td>
<td>0.15mm</td>
<td>0.07mm</td>
</tr>
</tbody>
</table>

Source: TopLine
Why NSMD Pads?
Difference between SMD and NSMD Pads

**Important!**

Today all Semiconductor manufacturers are recommending NSMD pads.

The reason for that:

NSMD pads offer more space for routing (e.g., additional conductor paths between the pads) and lead to a higher reliability of the solder joint.
Parameter NSMD Pads
Non solder mask defined (NSMD) BGA pads

Spacing / Leiterbahnabstand

Coverage / Abdeckung

Solder Mask Clearance / Freistellung

Coverage / Abdeckung  = >50µm
(Advanced > 38µm)

Spacing / Leiterbahnabstand  = >= 100µm
(Advanced >= 75µm)

Solder Mask Clearance / Freistellung  = > 50µm
(Advanced > 38µm)

Source: Fineline
Ti Design Rekommandation for 0.5 pitch BGA

Pad Type: NSMD

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>A (Pad Pitch)</td>
<td>500µm</td>
</tr>
<tr>
<td>B (Pad Size)</td>
<td>250µm</td>
</tr>
<tr>
<td>C (Mask Opening)</td>
<td>50µm around Pad</td>
</tr>
<tr>
<td>D (Mask Web)</td>
<td>150µm</td>
</tr>
<tr>
<td>E (Trace Width)</td>
<td>82µm</td>
</tr>
<tr>
<td>E (Pad to Trace Clearance)</td>
<td>82µm</td>
</tr>
</tbody>
</table>

Source: Fineline
Ti Design Recommendation for 0.5 pitch BGA

Pad Type: NSMD

- Pad Size: $B = 250 \mu m$
- Via pad size: $B = 254 \mu m$
- Via Drill size: $F = 127 \mu m$
- Pad to Via clearance: $G = 72 \mu m$
- Pad to Via Trace: $H = 82 \mu m$
- Length of Connecting: $I = 35 \mu m$
9 ▶ Others
Why green Solder Mask?

Why is the green color for solder mask an industry standard?

The right placement of components in SMD Lines are often controlled by IR or visible-red light and monochromatic cameras. The contrast is in case of non-red color paints is the highest one and cameras typically operate as a quasi- “B / W" camera.

There are also brown and blue colored SM, but it is more difficult to cure this colors with UV light.
THANK’S FOR LISTENING

Danke für’s Zuhören