**General description**

This document describes the attachment techniques recommended by Murata* for their HTSC silicon capacitors on the customer substrates. This document is non-exhaustive. Customers with specific attachment requirements or attachment scenarios that are not covered by this document should contact Murata. The solder printing is described in this document but other processes like solder jetting, pre-bumped capacitors… can also be used with the same recommendations.

**Handling precautions and storage**

Silicon die must always be handled in a clean room environment (usually class 1000 (ISO 6)) but the assembled devices don’t need to be handled in such an environment as the product is already well packed. The remaining quantities have to be repacked immediately after any process step, in the same conditions as before the opening (ESD bag + N2).

Store the capacitors in the manufacturer’s package in the following conditions without a rapid thermal change in an indoor room:

- Temperature: -10 to 40 degree C
- Humidity: 30 to 70%RH

Avoid storing the capacitors in the following conditions:

- (a) Ambient air containing corrosive gas. (Chlorine, Hydrogen sulfide, Ammonia, Sulfuric acid, Nitric oxide, etc.)
- (b) Ambient air containing volatile or combustible gas
- (c) In environments with a high concentration of airborne particles
- (d) In liquid (water, oil, chemical solution, organic solvents, etc.)
- (e) In direct sunlight
- (f) In freezing environments

*Murata Integrated Passive Solutions*
HTSC SiCap 400µm - NiAu finishing - Assembly by soldering

Rev. 2.0

To avoid contamination and damage like scratches and cracks, our recommendations are:

- Die must never be handled with bare hands
- Avoid touching the active face
- Do not store and transport die outside protective bags, tubes, boxes, sawn tape
- Work only in ESD environments
- Plastic tweezers or a soft vacuum tool are recommended to remove the silicon die from the packing.

Standard packing is tape & reel for die size larger than 0201 but silicon capacitors can be provided within waffle pack, gelpak or sawing frame. Please contact the Murata sales contact for drawing and references (mis@murata.com).

Pad opening

The top surface of the Murata silicon capacitors are protected with a mineral passivation. The finishing of the contact pads are in nickel gold (generally 5µm nickel and 0.2µm gold) conforming with the soldering process.

![Diagram showing pad opening](image)

Murata recommends having an opening on the board which matches the pad of the capacitor (size, position and spacing) – see figure 1. On the substrate, the metal layer can be larger than the varnish coating opening size but in this case, the varnish coating opening has to be mirror with the pad size of the capacitor. No need to change the metal landing pad of the PCB, only the opening in the varnish coating needs to be adjusted (see Figure 1). These recommendations will improve the die placement, tilting and will avoid the contact between the solder paste and the bare silicon die - see Figure 2.

![Solder paste after reflow diagram](image)

*Figure 1: Solder paste after reflow - Targeted*
Design of the board:

On the customer substrate, Murata recommends SMD (Solder Mask Design) to control the solder flowing on the tracks but NSMD (None Solder Mask Defined) can also be used with some precautions:
Nota: No varnish between two landing pads can be done. See figure below:

**Solder Mask Defined:**

![Example #1 of Solder Mask Defined (SMD) PCB](image1)

![Example #2 of Solder Mask Defined (SMD) PCB](image2)

**None Solder Mask Defined:**

![Example #3 of None Solder Mask Defined (NSMD) PCB](image3)

In case of NSMD, it is recommended to place a varnish in the metal track to limit the risk of contact between the solder paste and the capacitor side.

**Landing pad for the PCB and die pad dimensions for the Murata silicon die:**

<table>
<thead>
<tr>
<th>Silicon Capacitor Type</th>
<th>Capacitor size (µm²)</th>
<th>Capacitor thickness</th>
<th>A (µm)</th>
<th>B (µm)</th>
<th>C (µm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0201</td>
<td>800 x 600</td>
<td>100µm minimum</td>
<td>150</td>
<td>400</td>
<td>300</td>
</tr>
<tr>
<td>0402</td>
<td>1200 x 700</td>
<td></td>
<td>300</td>
<td>500</td>
<td>400</td>
</tr>
<tr>
<td>0603</td>
<td>1800 x 1100</td>
<td></td>
<td>400</td>
<td>900</td>
<td>800</td>
</tr>
<tr>
<td>0805</td>
<td>2200 x 1400</td>
<td></td>
<td>500</td>
<td>1200</td>
<td>1000</td>
</tr>
<tr>
<td>1206</td>
<td>3400 x 1800</td>
<td></td>
<td>600</td>
<td>1600</td>
<td>2000</td>
</tr>
<tr>
<td>1812</td>
<td>4700 x 3600</td>
<td></td>
<td>900</td>
<td>3400</td>
<td>2700</td>
</tr>
</tbody>
</table>
Process Flow

1- Solder printing of the substrate

Step A: Stencil

Step B: Substrate

Step C: Solder paste

Step D: 

2- Picking of the silicon die and placement

Rubber tip

Solder paste

3- Reflowing

Silicon capacitor

4- Cleaning

Silicon capacitor
Solder print material and stencil printing recommendations

Solder pastes SnPb63/37 or SAC305 are usually used and recommended but other materials compatible with the die pad finishing are also possible. In function of the die pad size, powder size could be adjusted. However, compared to type 3, type 6 limits the risk of tilting of the capacitor (see figure 2).

<table>
<thead>
<tr>
<th>ALLOY</th>
<th>COMPOSITION</th>
<th>SOLIDUS</th>
<th>LIQUIDUS</th>
<th>COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>SAC305</td>
<td>96.5Sn, 3Ag, 0.5Cu</td>
<td>217°C</td>
<td>217°C</td>
<td>Eutectic</td>
</tr>
</tbody>
</table>

Water soluble and no clean flux can be used. In case of water soluble flux, remove the flux immediately after reflow to avoid the potential issue of leakage current between pads.

Stencil design rules in function of the quality:

- INOX LASER: \([L \times W)/(2 \times (L + W) \times T)\] > 0.66 & \(W > 1.5 \times T\)
- NICKEL LASER: \([L \times W)/(2 \times (L + W) \times T)\] > 0.53 & \(W > 1.2 \times T\)
- ELECTROFORMED: \([L \times W)/(2 \times (L + W) \times T)\] > 0.44 & \(W > 1.0 \times T\)

And in all cases: \(W > 5 \times \) powder size

A solder joint thickness of 40 µm +/-10 is targeted to limit the risk of contact between the solder paste and the side of the capacitor. Such a contact would have a negative effect and would probably create a high leakage or a short circuit. Limited solder joint thickness will also avoid an excessive tilting of the capacitor.

For example, design of stencils done by Murata (SAC305 type 6 with 50% of flux):

<table>
<thead>
<tr>
<th>Silicon Capacitor Type</th>
<th>Stencil opening size µm²</th>
<th>Stencil thickness µm</th>
<th>Stencil quality</th>
</tr>
</thead>
<tbody>
<tr>
<td>0201</td>
<td>150 x 320</td>
<td>100</td>
<td>ELECTROFORMED</td>
</tr>
<tr>
<td>0402</td>
<td>260 x 369</td>
<td>125</td>
<td>NICKEL LASER</td>
</tr>
<tr>
<td>0603</td>
<td>300 x 768</td>
<td>125</td>
<td>NICKEL LASER</td>
</tr>
<tr>
<td>0805</td>
<td>400 x 960</td>
<td>125</td>
<td>NICKEL LASER</td>
</tr>
<tr>
<td>1206</td>
<td>500 x 1229</td>
<td>125</td>
<td>NICKEL LASER</td>
</tr>
<tr>
<td>1812</td>
<td>650 x 3012</td>
<td>125</td>
<td>NICKEL LASER</td>
</tr>
</tbody>
</table>
**Procedure for the solder joint measurement (After reflow):**

**STEP 1:**
- Silicon Capacitor Thickness: X µm
- Substrate

**STEP 2:**
- Silicon Capacitor
- Y µm (Measured value)
- 40 µm (Solder joint thickness)

**Pick and Place**

The most common approach is with automatic equipment using vision assist to correct placement after picking but manual placement can also be done.

Using a rubber tip is particularly preferred for the die manipulation.

A minimum pressure of 50 grams and a maximum of 150 grams is recommended for the die placement on the solder paste.

**Reflow soldering**

Murata recommends convection reflow but vapor phase reflow and infrared reflow could be also used.

The reflow must be carried out in accordance with the JEDEC standard.

*Figure 4: Generic reflow profile according to JEDEC J-STD-020-C*
Flux removes tarnish films, maintains surface cleanliness and facilitates solder spread during attachment operations. The flux must be compatible with the soldering temperature and soldering times. In case of water soluble flux, please refer to the solder paste supplier for the cleaning and flux removal. Flux residues could be responsible for current leakage or short circuits. For optimum results, clean the circuits immediately after reflow.