life.augmented

AN4211
Application note

Guidelines for soldering MEMS microphones

By Alessandro Morcelli

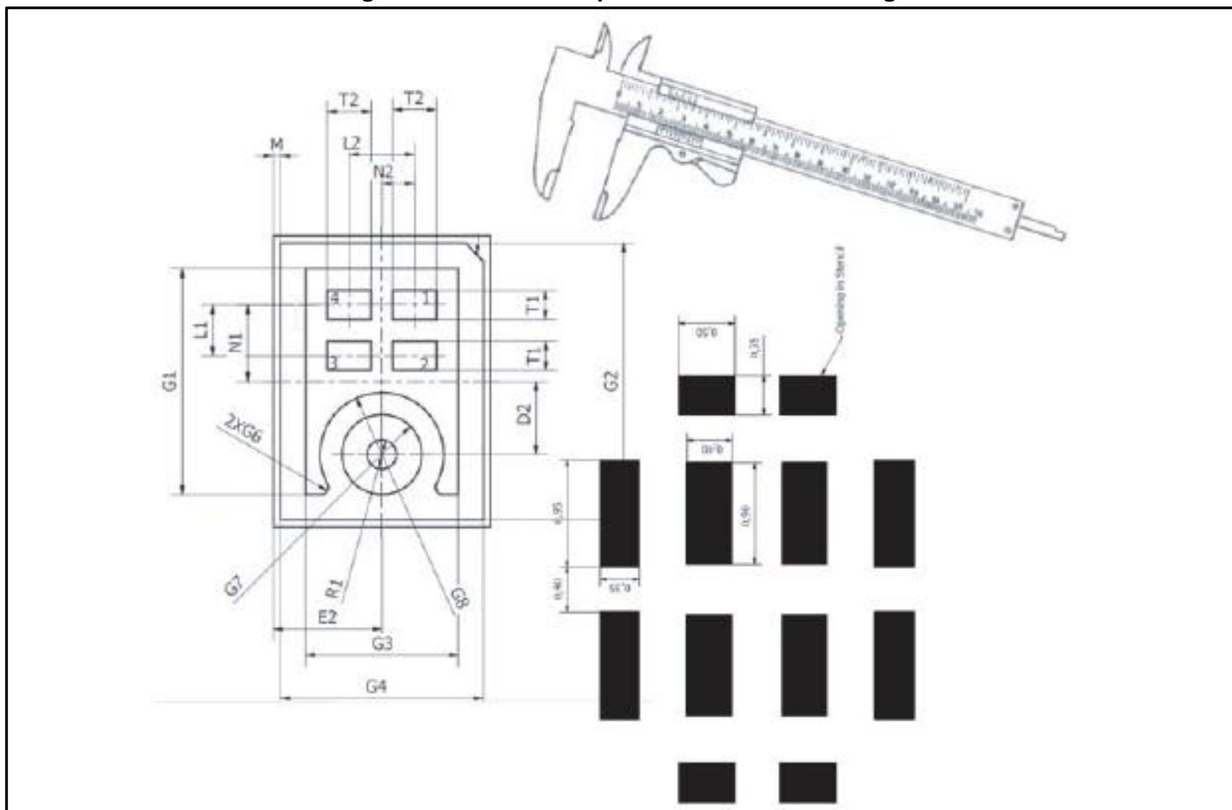
Introduction

This application note provides guidelines and recommendations for soldering MEMS microphones on application boards. Microphones, as pressure sensors, sense the mechanical stress coming from the PCB, hence this force should be kept to a minimum. Printing and stencil parameters, followed by device footprints/recommended land patterns, stencil openings, and process considerations are presented in this document.

The following guidelines have been developed for the MEMS microphone family (MP45DT02/-M, MP34DB01, MP34DT01/-M, MP33AB01/H, MP34DB02, and MP23AB02B) offered by STMicroelectronics.

Demonstration boards (STEVAL-MKI129Vx) for the evaluation and promotion of the MP45DT02, MP34DT01, and MP34DT01 are also available. Please refer to AN4184 "Microphone coupon boards STEVAL-MKI129Vx based on the MP45DT02, MP34DB01, and MP34DT01" on www.st.com for further information.

Figure 1: Mechanical specifications for soldering



Contents

1 Printing and stencil parameters ... 3
2 Device footprint, land pattern and stencil openings ... 4
2.1 MP45DT02 and MP45DT02-M dimensions ... 4
2.2 MP34DB01 dimensions ... 5
2.4 MP34DT01 and MP34DT01-M dimensions ... 6
2.5 MP33AB01 and MP33AB01H dimensions ... 7
2.6 MP34DB02 dimensions ... 8
2.8 MP23AB02B dimension ... 9
3 Process considerations ... 10
4 Revision history ... 11

1 Printing and stencil parameters

The soldering paste thickness and pattern are important for the proper microphone soldering process. Some recommendations to be followed are listed below:

- Stainless steel stencils are recommended
- Stencil thickness of 75 - 100 µm (2.95 - 3.94 mils) is recommended for screen printing
- The final soldering paste thickness should allow proper cleaning of flux residuals and clearance between the sensor package and PCB
- Stencil aperture should have a rectangular shape
- The openings of the stencil for the signal pads should be between 70 - 90% of the PCB pad
- Optionally, for better solder paste release, the aperture walls should be trapezoidal and the corners rounded
- The fine IC leads pitch requires accurate alignment of the stencil to the PCB. The stencil and printed circuit assembly should be aligned to within 25 µm (1 mil) prior to application of the solder paste

2 Device footprint, land pattern and stencil openings

2.1 MP45DT02 and MP45DT02-M dimensions

Figure 2: MP45DT02 - device footprint and PCB land pattern

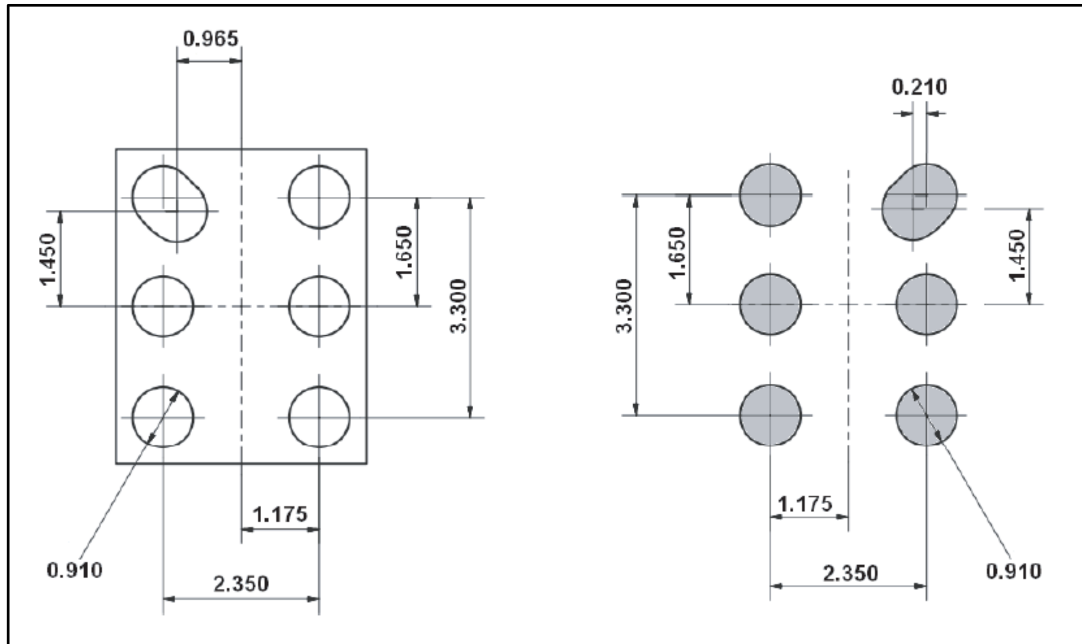
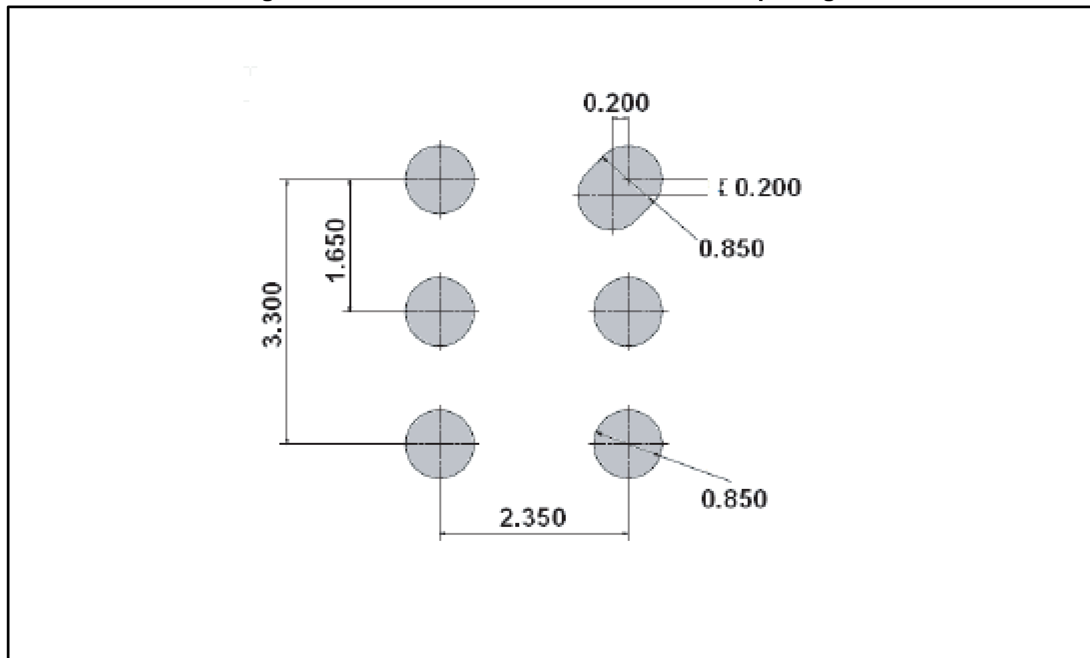


Figure 3: MP45DT02 - recommended stencil openings



2.2 MP34DB01 dimensions

Figure 4: MP34DB01 - device footprint and PCB land pattern

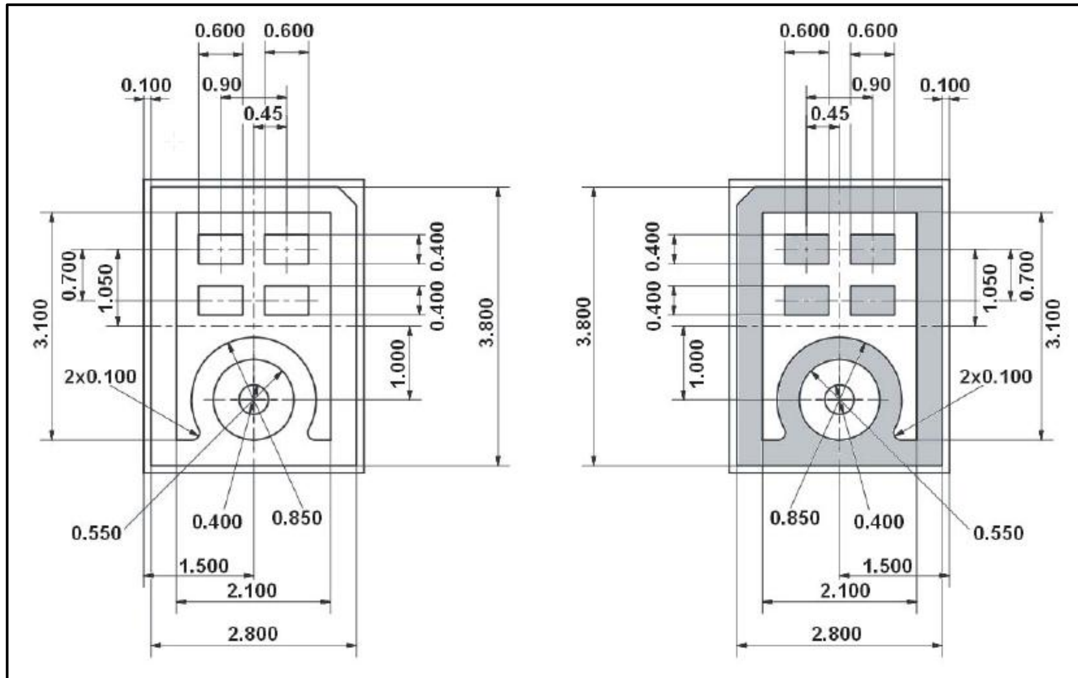
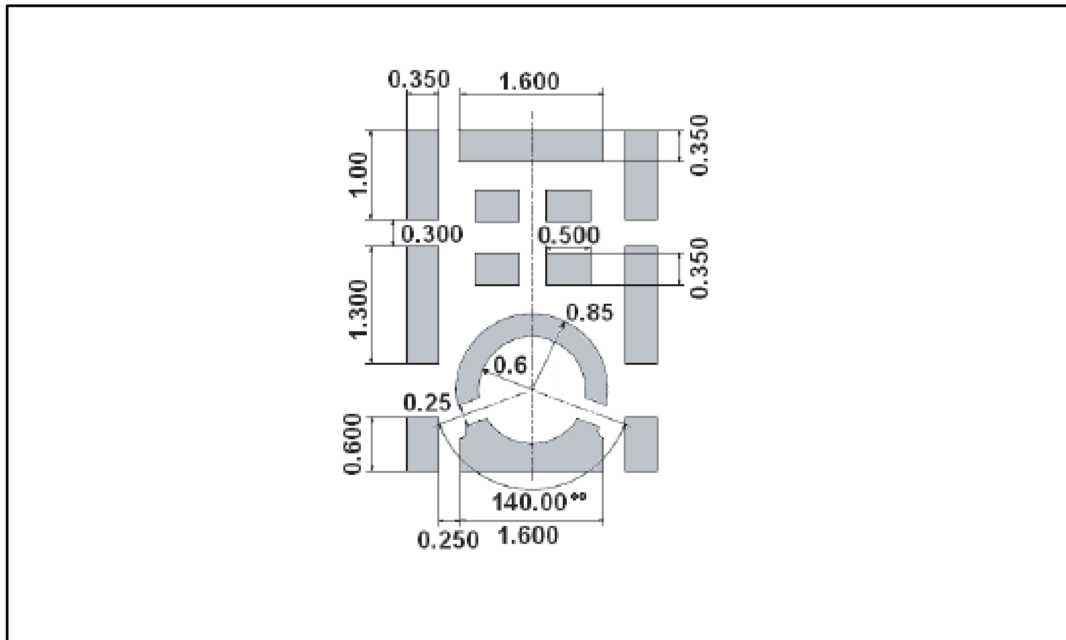


Figure 5: MP34DB01 - recommended stencil openings



2.4 MP34DT01 and MP34DT01-M dimensions

Figure 6: MP34DT01 - device footprint and PCB land pattern

Figure 7: MP34DT01 - recommended stencil openings

2.5 MP33AB01 and MP33AB01H dimensions

Figure 8: MP33AB01 and MP33AB01H - device footprint and PCB land pattern

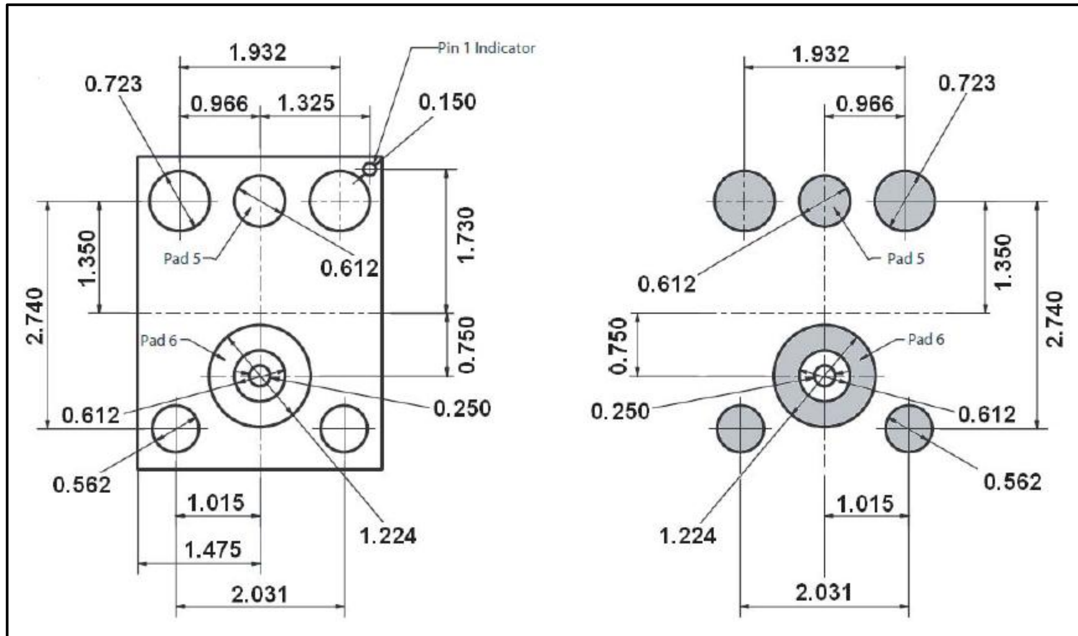
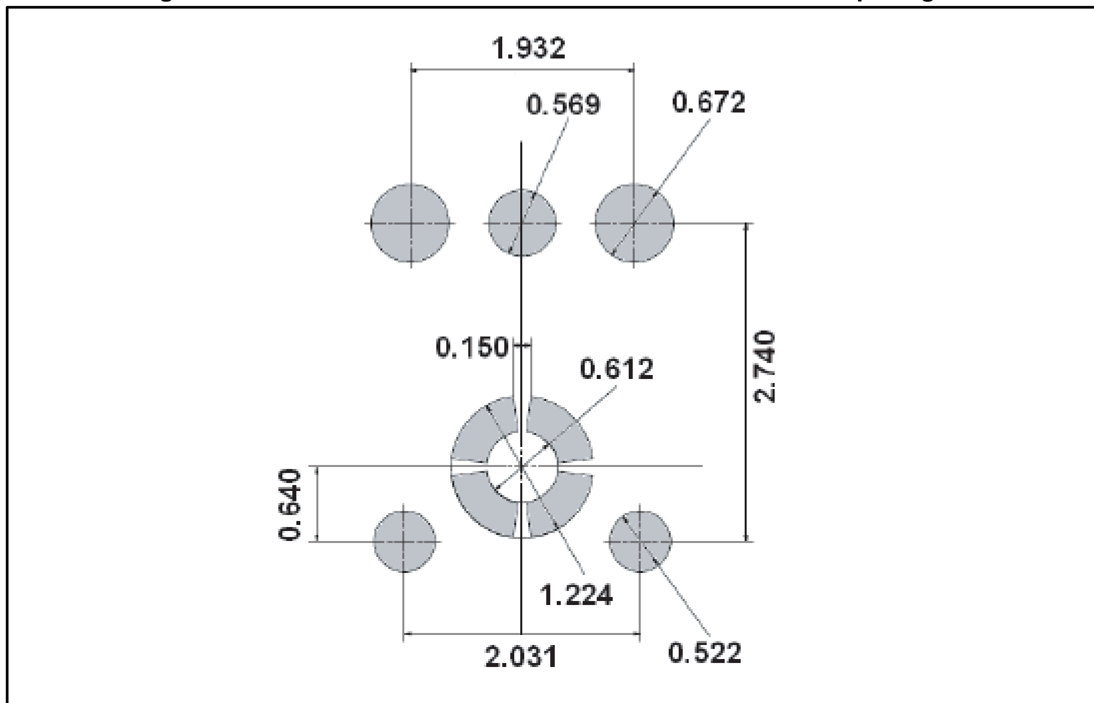


Figure 9: MP33AB01 and MP33AB01H - recommended stencil openings



2.6 MP34DB02 dimensions

Figure 10: MP34DB02 - device footprint

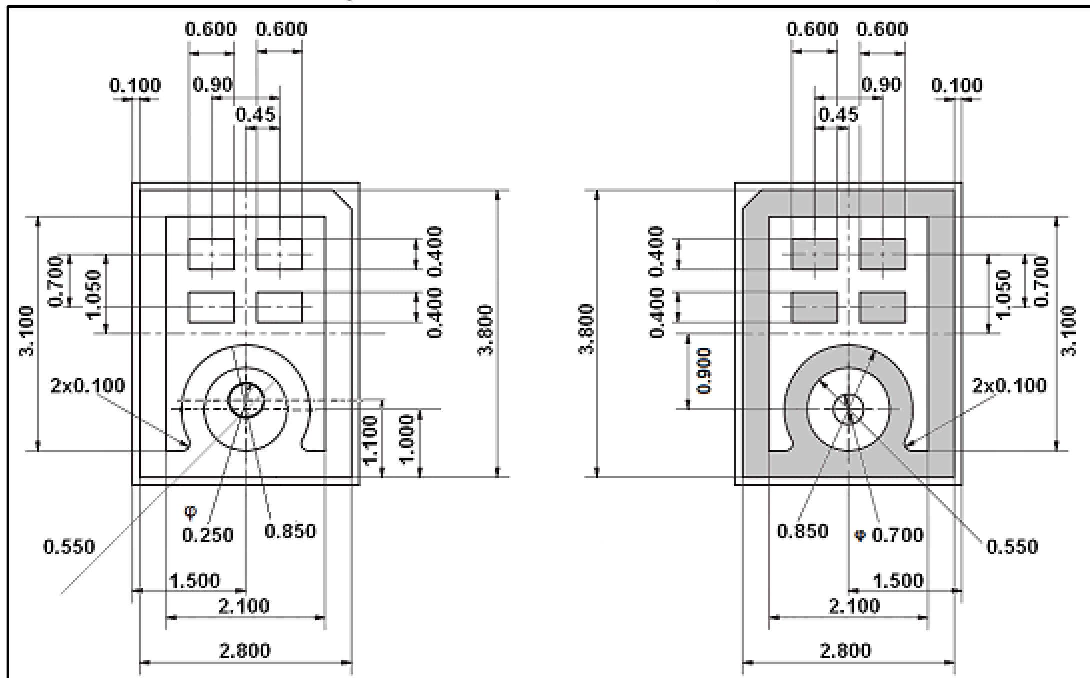
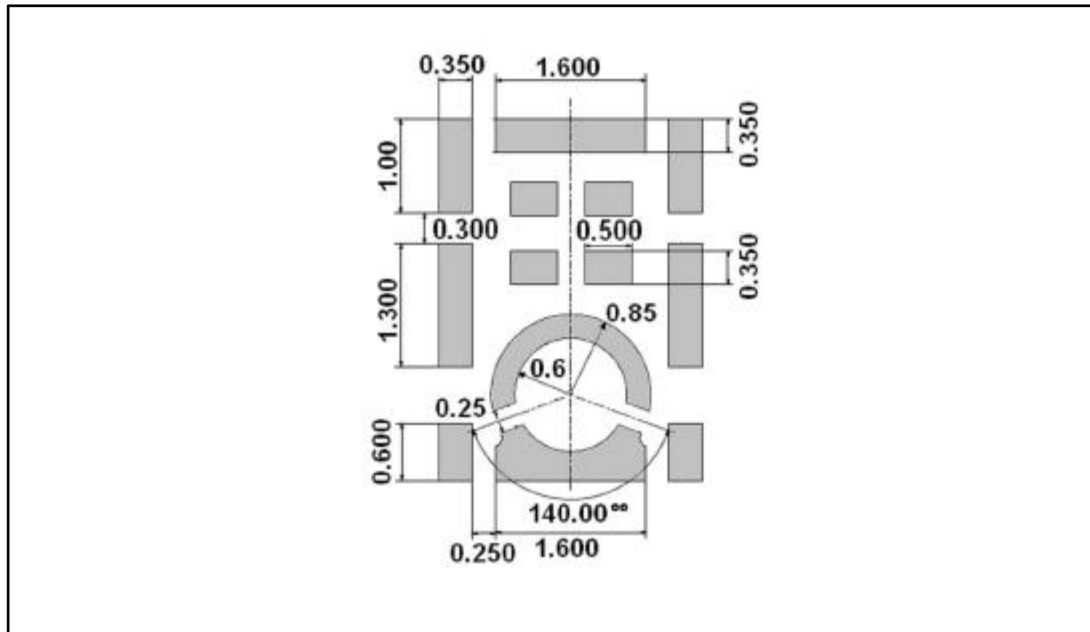


Figure 11: MP34DB02 - recommended stencil openings



2.8 MP23AB02B dimension

Figure 12: MP23AB02B - device footprint

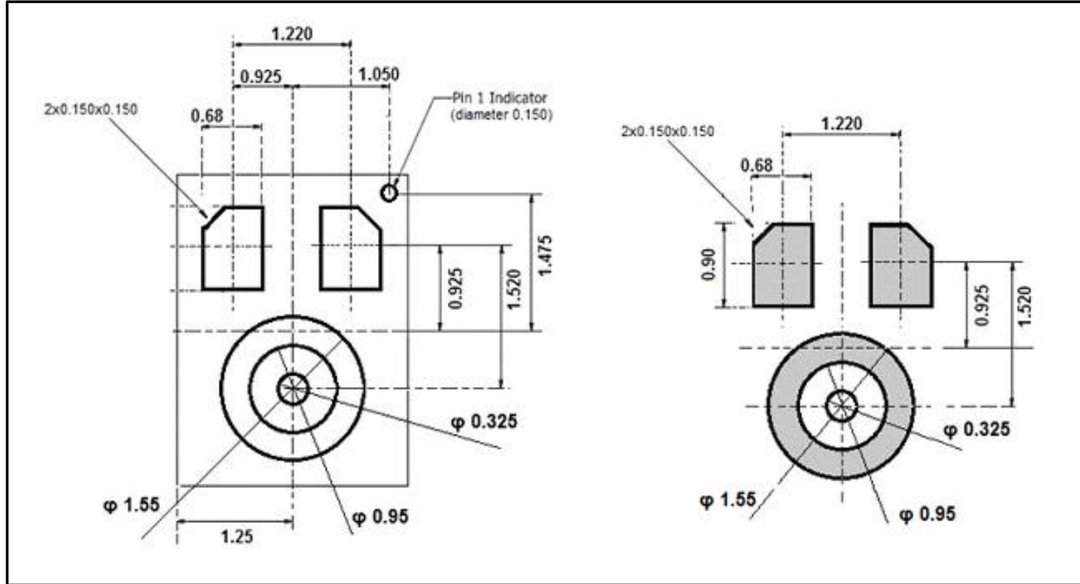
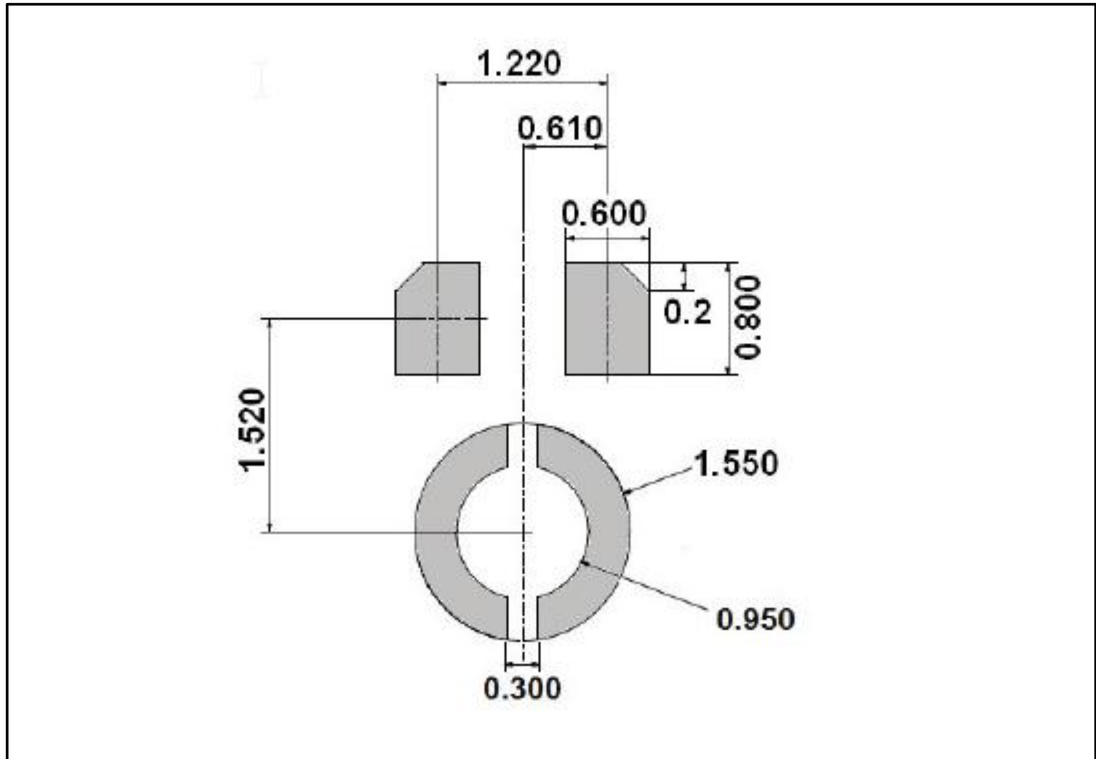


Figure 13: MP23AB02B recommended stencil openings



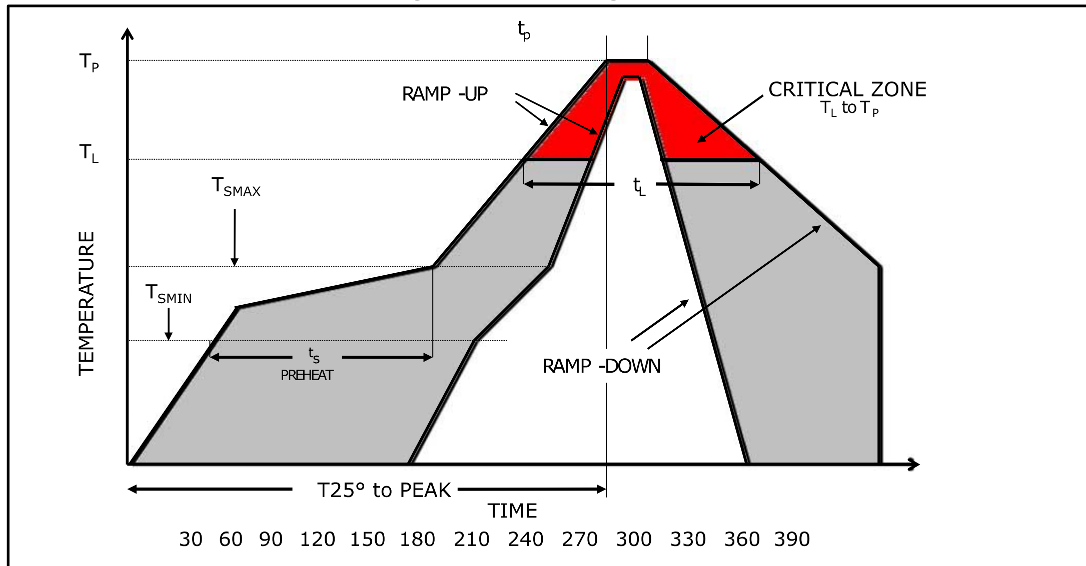
3 Process considerations

If self-cleaning solder paste is not used, proper cleaning of the board after soldering must be done to remove any possible source of leakage between pads due to flux residues. For the soldering profile, refer to the table and figure below.

Table 1: Soldering profile

Description	Parameter	Pb free
Average ramp rate	T_L to T_P	3 °C/sec max
Preheat Minimum temperature Maximum temperature Time (T_{SMIN} to T_{SMAX})	 T_{SMIN} T_{SMAX} t_S	 150 °C 200 °C 60 sec to 120 sec
Ramp-up rate	T_{SMAX} to T_L	3 °C/sec max
Time maintained above liquidus temperature Liquidus temperature	t_L T_L	60 sec to 150 sec 217 °C
Peak temperature	T_P	260 °C max
Time within 5 °C of actual peak temperature		20 sec to 40 sec
Ramp-down rate		6 °C/sec max
Time 25 °C (t25 °C) to peak temperature		8 minutes max

Figure 14: Soldering profile



No solder material reflow on the side of the package is allowed since LGA packages have metal traces on the sides of the package.

4 Revision history

Table 2: Document revision history

Date	Revision	Changes
12-Mar-2013	1	Initial release.
21-Jul-2014	2	Added *Section 2.5: "MP34DB02 dimensions"* and *Section 2.6: "MP23AB02B dimension"*.
08-Aug-2014	3	Modified *Figure 9: "MP33AB01 and MP33AB01H - recommended stencil openings"*.
15-Jun-2015	4	Updated *Section "Introduction"* Updated *Section 2: "Device footprint, land pattern and stencil openings"*

IMPORTANT NOTICE – PLEASE READ CAREFULLY

STMicroelectronics NV and its subsidiaries (“ST”) reserve the right to make changes, corrections, enhancements, modifications, and improvements to ST products and/or to this document at any time without notice. Purchasers should obtain the latest relevant information on ST products before placing orders. ST products are sold pursuant to ST’s terms and conditions of sale in place at the time of order acknowledgement.

Purchasers are solely responsible for the choice, selection, and use of ST products and ST assumes no liability for application assistance or the design of Purchasers’ products.

No license, express or implied, to any intellectual property right is granted by ST herein.

Resale of ST products with provisions different from the information set forth herein shall void any warranty granted by ST for such product.

ST and the ST logo are trademarks of ST. All other product or service names are the property of their respective owners.

Information in this document supersedes and replaces information previously supplied in any prior versions of this document.

© 2015 STMicroelectronics – All rights reserved