BL-Cellular Systems, MST RF Power
Basestations DEV&QAS

Title:	Solder mounting recommendations for Ldmos Power Amplifiers
Author:	Josselin FAY, Michael ASIS, Jose SALTA
Doc. Nr.	
Date:	2008/02/14

General solder mounting recommendations for Ldmos Power Amplifiers

TABLE OF FIGURES

1 Solder mounting method: introduction

1.1 General

RF power semiconductors assembled in air cavity plastic packages like the SOT896B and ceramic packages like the SOT502B are mounted onto a PCB or heatsink by soldering.
The package is composed of several components. The flange is the lower part of the component used as a heat spreader. The transistors and capacitors (dies) are attached on the plated flange. The flange is mounted via bolt down or soldering mounted in a recessed area onto a metal support or heatsink.

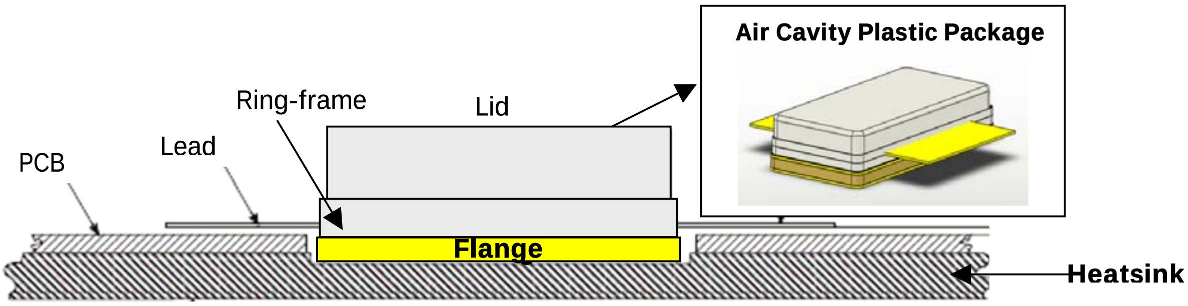


Figure 1 – Presentation of the device

Above is a drawing of the device. On the flange, a plastic package (or ceramic package) is “used”. It is composed of a ring-frame glued on the flange (soldered for ceramic packages). The ring-frame supports the leads that are soldered on the PCB[1]. A lid is glued on the ring-frame to protect the capacitors and transistors.

An Air Cavity Package has thus four main functions:

- Provide electrical interconnection between the silicon LDMOS chip and the external circuit;
- Protect the devices from chemically aggressive agents, for long term reliability (protection of the Power Amplifier Module);
- Mechanical support to the LDMOS die to make handling easier;
- Provide thermal conductive path to transfer the heat generated in operation from the silicon LDMOS die to ambient or to the heatsink.

There are two different mounting flanges shown in the pictures below:

- A flange without screws, mounted onto the heatsink by soldering: i.e. NXP’s SOT[2]896B, SOT 502B packages;

- A flange fixed by using two screws or bolt down mounting: i.e. NXP’s SOT895A, SOT539A, SOT979A packages.

This document will only discuss the solder mounting method.

[1] Printed Circuit Board
[2] Small Outline Transistor

Dies are connected to the leads by using bonded wires. It allows the PCB to be in contact with the component (Cf. Figure 1). Several problems can occur if several precautions are not taken into account. The soldering mounting method indeed requires knowing what temperature profile packages must be soldered onto the heatsink and what happens when different methods of soldering are used.

Other important problems are to take into account in mounting packages properly. Packages are unreliable if mounting conditions are not achieved. Damage, failure zero hour[1] or problems of reliability can be encountered.

1.2 Objective

To preserve the efficiency of Air Cavity Package it is necessary to carefully follow the prescribed mounting recommendations.
Recommendations are to be taken into account in order to have best thermal performance and good electrical contact. It will guarantee a good functioning device.

2 PCB/Heatsink design and tolerances

Because this document is applicable to packages with body dimensions the design rules of the mounting areas are general. The tolerance of each part of the device must be respected in order to ensure optimal assembly.
If the package is mounted to a heatsink in a cavity, the cavity dimensions must be at least:

- Cavity length= maximum package length+0.5 mm, allowed tolerance: ±0.1mm;
- Cavity width= maximum package width+0.5 mm, allowed tolerance: ±0.1 mm;
- Cavity depth: depending on the thickness of flange and glue layer the depth can vary but depth dimension must be such that the maximum lead deflection is 0.3 mm. The lead deflection must always be toward the back of the component because it is important in controlling stresses on the component and the solder joint. The tolerance of the recessed cavity depth is ±0.08 mm.

The maximum package length/width is defined as the nominal length/width plus the tolerance, as indicated in SOT outline drawing.
The achievable tolerance of the PCB thickness is to be kept to 0.18 mm.

3 Soldering the flange onto the heatsink

3.1 Description

The mounting method requires soldering the package to a heatsink through the flange with the leads soldered to a PCB. The most common ways to solder the flange to a heatsink are:

- Solder paste;
- Preform with pre-applied flux.

The flux is designed to activate the contact area.
Reflow soldering is used to surface mount the package to the PCB. A temperature profile is created to solder the package onto the heatsink (one reflow is when there is one pass of the device inside the oven. Two reflows are thus when the samples are twice inside the oven).

The following document will focus on the Air Cavity Plastic Packages because they are far more sensitive than Ceramic Packages. First, for plastic packages the ring-frame is glued to the flange (not brazed) and secondly intrinsically using plastic for the reflow step is more delicate (device are moisture sensitive): like popcorn, the moisture in some plastic packages can vaporize and expand rapidly due to high temperatures causing cracks.

The soldering areas flange/heatsink and Leads/PCB will be seen. The affected devices are the soldered air cavity plastic packages and soldered ceramic air cavity packages.

[1] Before the operational use

3.2 Mounting recommendations prior to reflow soldering

3.2.1 Requirements

- Due to the fact that many plastic packages and some ceramic packages are moisture sensitive the packages must be stored in an environmentally controlled area. For further instructions see JEDEC spec.doc. No 020-C.
- For electrostatic sensitive devices NXP advises to handle the devices according to the JEDEC spec.doc no. 625-A, "Requirements for Handling Electrostatic-Discharge-Sensitive (ESDS) Devices, (Revision of EIA-625) December 1999. At the very least, somebody handling a device should wear a wrist bracelet that is properly grounded to avoid damaging electrostatic sensitive devices.

3.2.2 Cleanliness of soldering areas

The soldering areas (leads/lands of the PCB and flange/heatsink) must be free of contaminations like grease, oils or other particles. It is advised to use gloves during handling.

3.2.3 Heatsink preparation

When mounting the package, observing the following will produce good thermal and electrical contact between the package and the heatsink.

For packages dissipating up to 80 W, heatsink thickness should be:

- At least 3 mm for copper heatsink (>99.9% ETP-Cu);
- At least 5 mm for Aluminium heatsink (>99.9% Al).

These thicknesses should be increased proportionally for packages dissipating more power.

Be advised that NXP recommends the following heatsink properties:

- Flatness of the mounting area: better than 0.02 mm
- Mounting area roughness: Ra<0.5µm
- Mounting area should be free of oxidation

3.3 Reflow soldering process

There are three basic process steps for single-sided PCB pads/Heatsink reflow soldering. These are:

- Applying solder paste or preform to the PCB pads/heatsink
- Component placement
- Reflow soldering

3.3.1 Applying solder paste/preform to the PCB pads or the heatsink

The flange is plated with typically 2.54 µm of gold. To avoid brittle solder joints due to too much gold in the joint, sufficient solder material should be applied (≈ 150 µm only based on NXP part). This ensures that the level of gold within the final solder joint does not exceed 4% by weight and therefore, it minimizes brittle AuSn[1] intermettalics that can introduce cracks during power cycling [1 − 2] (Cf. Appendix A). The leads are plated with a 0.8 µm to 1.27µm gold layer.

- **Preform**

  NXP used SAC 95.5Sn3.8Ag0.7%[2]Cu lead free solder performs. The required amount of solder is indicated in Appendix A. No special difficulty is encountered for the placement of the preform in the cavity (flange/heatsink attachment).

- **Solder paste**

  Using solder paste requires knowing, the amount of solder paste to spread and how to apply the solder paste onto the surface. As seen previously, the amount of solder paste depends on the 4% by weight threshold for gold embrittlement. (Furthermore, other important parameters that must be taken into account: the flux (necessary to activate the contact areas) could damage the glue layer situated in the interface flange/ringframe (Cf. Figure 2: interface A))

[1] AuSn= gold-tin
[2] By weight

Air Cavity Plastic Package

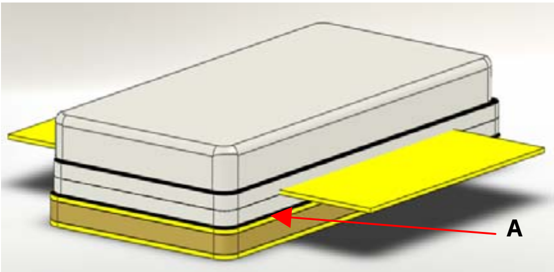
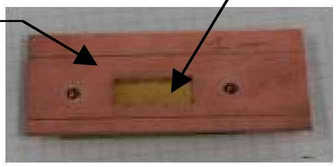


Figure 2 – Location of glue layers

Assessments with solder paste have been performed on the interface flange/heatsink.
The solder paste must be **evenly spread** on the soldering surface. This action is made using a stainless metal stencil and a squeegee (Cf. Figure 3). The squeegee is passed across the stencil and onto the solder lands on the PCB. The size of the stencil (length and width) is similar to the flange.

Stainless metal stencil



Surface where the solder paste is evenly spread

Figure 3 – Stainless metal stencil

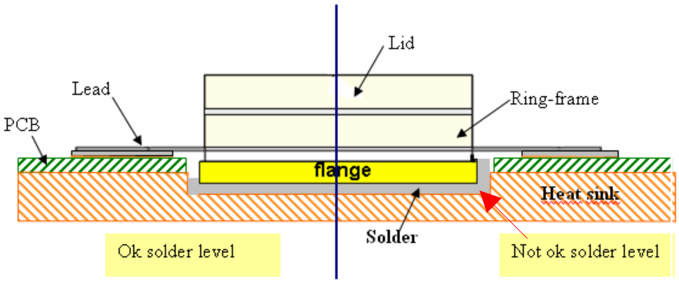


Figure 4 – Level of solder paste

According to tests performed using a lead-free solder SAC 95.5Sn3.8Ag0.7%Cu, the reduction of the weight of solder paste before and after soldering is about 12% (If a solder paste with 12% flux by weight is taken into account). Furthermore, the reduction of thickness (from solder paste to solder after the reflowing step) is about 50%.

The amount of solder paste to spread is done in Appendix A.

- **Recommendation**

  As seen in the introduction a convection oven is used in order to solder the package onto the heatsink. When solder paste is used voids can appear on the soldering part. Therefore, the amount of flux must be reduced to the absolute minimum. According to these tests NXP **advises using preform** with pre-applied flux. It reduces voids and ensures the required amount of solder.

3.3.2 Component placement

The position of the component with respect to the solder lands is of crucial importance to the end result of the assembly process. Misalignment can lead to unreliable joints or bridges between leads and/or open circuits. Placement accuracy is defined as the maximum permissible deviation of the component outline or component leads, with respect to the actual position of the solder land pattern on the PCB.

3.3.3 Device moisture sensitivity

As seen previously plastic packages are moisture sensitive. The standard JEDEC moisture/reflow sensitivity classification procedure **(IPC/JEDEC J-STD-020-C)** for Pb-free solder is used. Since the products are rated MSL3 preconditioning, the material must be stored in a vacuum sealed bag with moisture dessicant inside, or when used in production, stored under nitrogen conditions for not more than 8 days at max 30°C and max 60% RH (Relative Humidity) to limit possible moisture intrusion.

3.3.4 Reflow step

Packages are submitted to reflow profile according JEDEC classification: Peak temperature 245 +0/-5°C. There are several methods of reflow soldering to provide the heat to reflow the solder paste like convection, conduction, and radiation. The conduction method is sensitive to tooling and the radiation method is influenced by color. The preferred method is convection reflow.

- **Description**

The device is reflow soldered and cooled. NXP advises to use a convection oven rather than a conduction or radiation oven. A convection oven provides a uniform heat and a very controlled temperature (±2°C). Moreover, it allows soldering a wide range of products due to the temperature uniformity. During the reflow soldering process all parts of the board are subjected to an accurate temperature/time profile.

NXP uses SAC lead free solders: e.g. 95.5Sn3.8Ag0.7Cu. Figure 5 shows a suitable profile framework for single sided reflow. The framework can be smaller due to other components that have other process requirements. It is important to know that this profile is for discrete packages only.

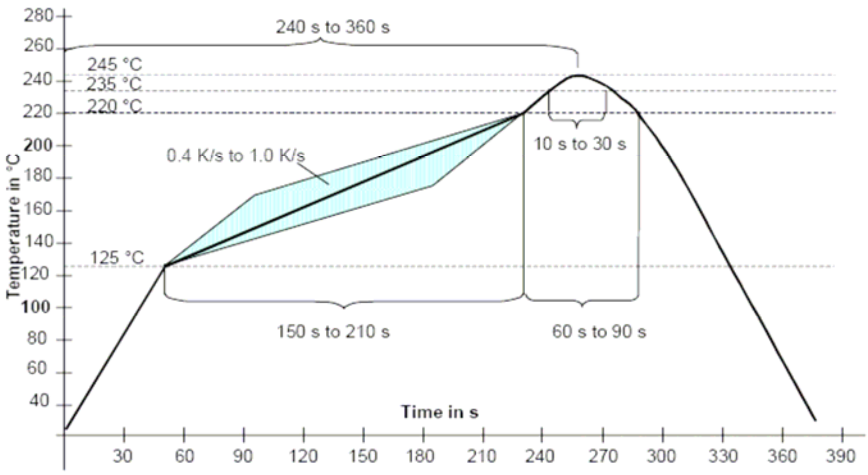


Figure 5 – Reflow profile

- **Calibration**
  - The reflow soldering profile should be calibrated with thermocouple glued down on the cap of the ACP device to prevent a temperature offset (Cf. Appendix B).
  - Create a reflow soldering profile with a good pre-heat period to allow the flux to react completely and allow all volatile and liquid content to evaporate and perform optimal cleaning before the used solder reaches liquidus. Furthermore, it allows the heating rate of the board and the components to be the same.

4 Remark

NXP requires the use of a no-clean type of flux during any solder mounting process of the product, be it an intergrated part of pre-form solders, solder pastes or in a form that is applied separately from the soldering material when mounting LDMOS power amplifier devices made with Air Cavity Plastic (ACP) Package.

REFERENCES

[1] Ed Hare, "Gold Embrittlement of Solder Joints", 2004

[2] D.F.Susan et al, "Microscopy of Solder Joint Failures Due to Gold Intermetallic Embrittlement", Sandia National Laboratories, 2005

APPENDIX A: AMOUNT OF SOLDER REQUIRED

The flange is plated with typically 2.54 µm of gold. The leads are plated with Max 1.27µm of gold. To avoid brittle solder joints due to too much gold in the joint, sufficient solder material should be applied. It ensures that the level of gold within solder joint does not exceed 4% by weight and therefore, it minimizes brittle AuSn intermettalics which can introduce cracks during power cycling [1 − 2].

- **Solder paste**

NXP used Pb-free solder 95.5Sn3.8Ag0.7%Cu SAC for its evaluation. This solder paste contained 88% w/w[1] of powdered solder alloy (96SC) and 12% w/w flux. The density of the Sn3.8Ag0.7Cu Pb-free solder alloy is approximately 7.5 g/cc.

- **Flange to Heatsink attachment**

  - **Required amount of solder (after the reflowing step)**

$$\boxed{\text{Weight}_{Au} \leq 4\% \ \text{Weight}_{total}}$$

With: $\text{Weight}_{total} = \text{Weight}_{Au} + \text{Weight}_{solder}$

Therefore, the amount of solder required after reflow is calculated using,

$$\text{Weight}_{Au} = (\text{Volume}_{Au}) \times \rho_{Au} = (\text{Thickness}_{Au} \times \text{Width}_{Flange} \times \text{Length}_{Flange}) \times \rho_{Au}$$

$$\Rightarrow \quad \text{Weight}_{Au} = (\text{Thickness}_{Au} \times 1 \times 1) \times \rho_{Au} \quad \text{(for 1 unit of surface area)}$$

$$\Rightarrow \quad \text{Weight}_{Au} = (\text{Thickness}_{Au}) \times \rho_{Au}$$

$$\text{Weight}_{total} = (\text{Thickness}_{Au}) \times \rho_{Au} + (\text{Thickness}_{solder}) \times \rho_{solder} \quad \text{(for 1 unit of surface area)}$$

With:
- ρ is the density of the material. The density of gold is 19.3 g/cc
- $\text{Thickness}_{Au} = 2.54$ µm
- The density of solder is 7.5 g/cc
- $\text{Thickness}_{solder} = \mathbf{X}$ µm

To avoid brittle solder joint due to too much gold in the joint, **X**≈ 150 µm solder thickness (only based on NXP part) is required for flange/heatsink attachment.

[1] w/w : In relation to the weight and not to the volume

- **If solder paste is used**

The difference in thickness before soldering and after reflow is about 50%. Therefore, the thickness of solder paste to spread onto the soldering surface is Min 300 µm to ensure the Min 150 µm of solder required after reflow. Therefore, the thickness of stencil that should be used to spread the solder paste must be between 12-15 mils so that after reflowing the solder layer thickness is more than 150 µm. It is tricky to ensure a proper and reliable soldering step with this amount of solder paste. It is also the reason why NXP advises to use Preform for flange to heatsink attachment.

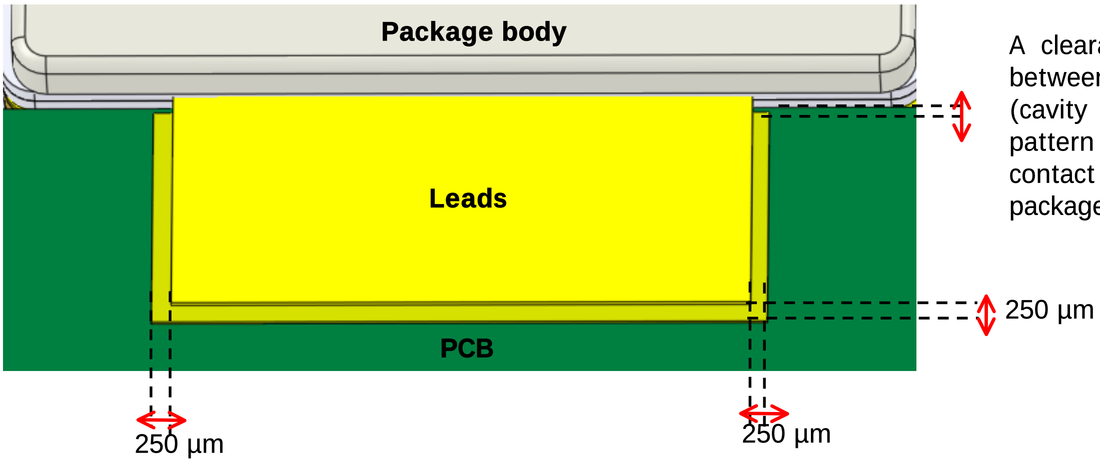
- **If Preform is used**

NXP advises using preform with pre-applied flux for flange to heatsink attachment. It reduces voids and ensures the required amount of solder.
NXP recommend to use a 150 µm solder perform in order to ensure the required amount of solder (to avoid Au embrittlement of solder joints).

- **Lead to PCB attachment**

The leads are plated with a thinner Au layer compared to the flange: [0.8µm – 1.27µm]. Therefore, the thickness of solder required after soldering is about 75 µm.
As seen previously, the difference in thickness before soldering and after the reflowing step is about 50%. Therefore, the thickness of solder to spread onto the soldering surface (PCB pad) is about 150 µm. It allows ensuring about 75 µm of solder after the reflowing step. A 6 mils stencil should then be used to evenly spread the solder paste.

The drawing below gives information about solder pads on PCB. The solder pads on PCB should be 0.250 mm larger than length and width of lead. This value is given as a Minimum.

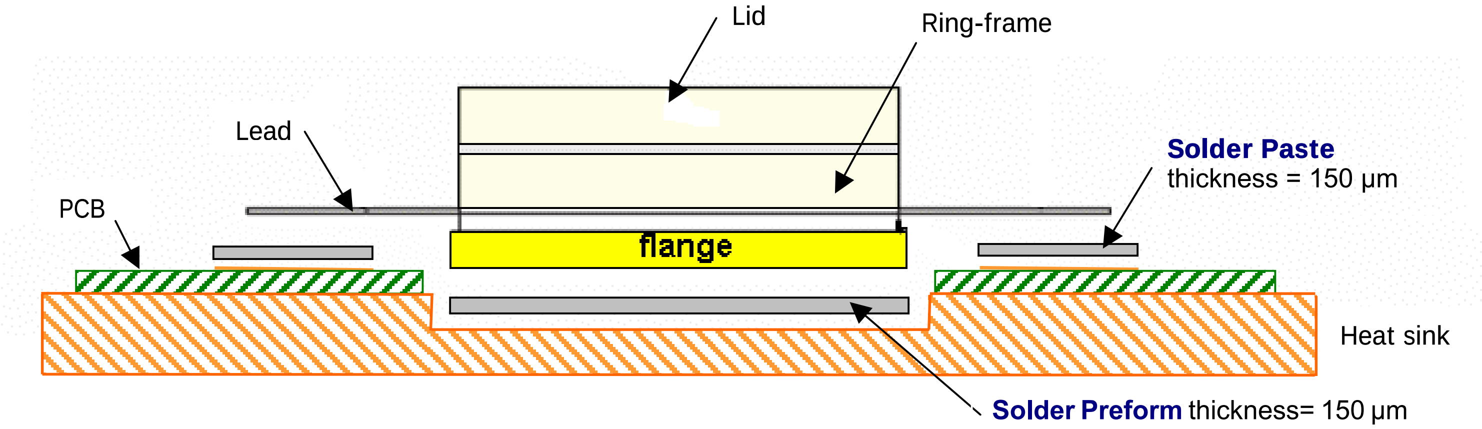


A clearance must be present between the package body (cavity edge) and the solder pattern to avoid any solder contact with the Plastic package: Min 250 µm

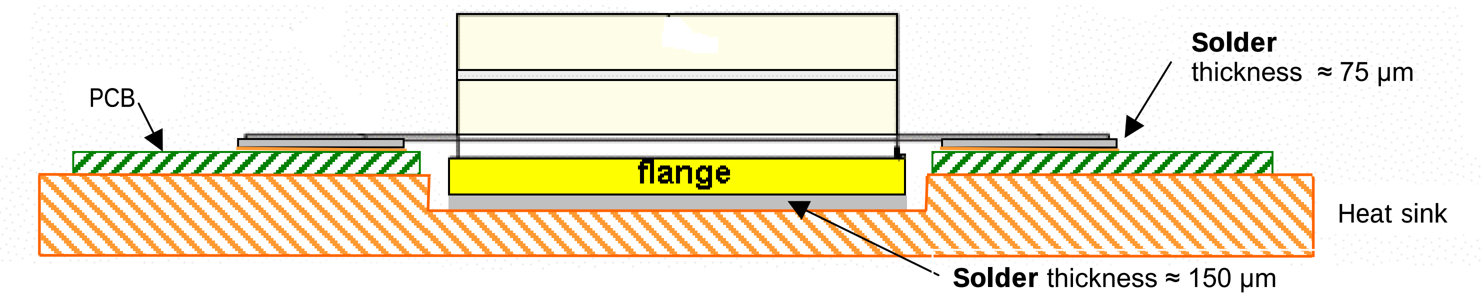
This ensures that the package leads will not come to rest on the solder resist ridges on the copper, which would give the package a vertical offset and perhaps even a tilt.

- **Conclusion/recommendations**

  - **Before the reflowing step**



  - **After the reflowing step**



APPENDIX B: CONVECTION OVEN

The convection oven is about 3 m long. It is composed of 6 zones with adjustable temperatures to produce the desired temperature profile.


Figure 6 – Entry of the convection oven (left) and exit of the convection oven (right)

The graph below shows three very slow temperature profiles. These temperature profiles have been obtained (built) by using three captors situated on three devices representing exactly the devices used for the reflow tests: it is thus important to know the total mass of the package/heatsink/PCB to be soldered in order to create the temperature profile.

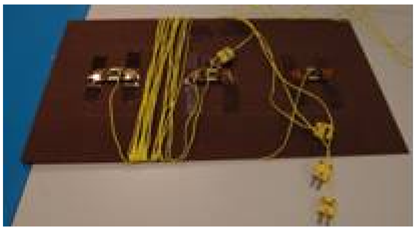
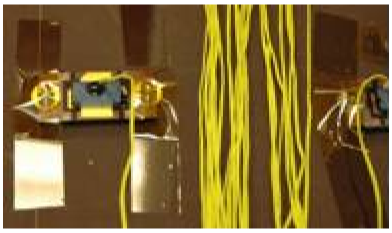



Figure 7 – The three devices with captors on the top of the packages to create the temperature profile

As example, the following graphs indicate the temperature profiles measured onto several packages (sensor glued onto the package cap). The belt speed and the temperature inside each part of the oven are indicated.