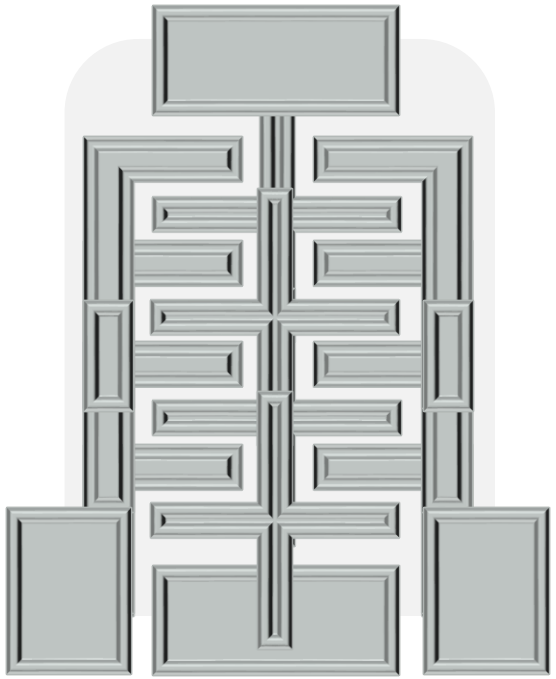
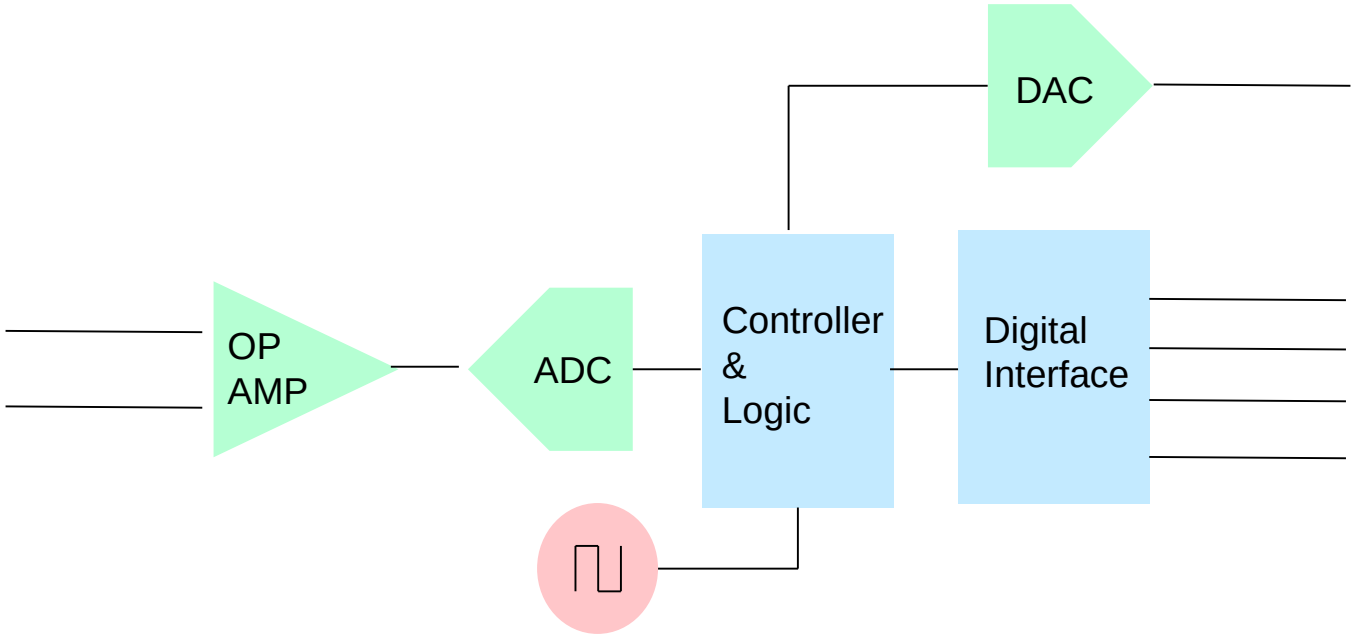
Table of content

MEMS Sensors

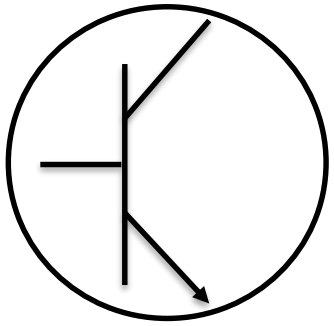




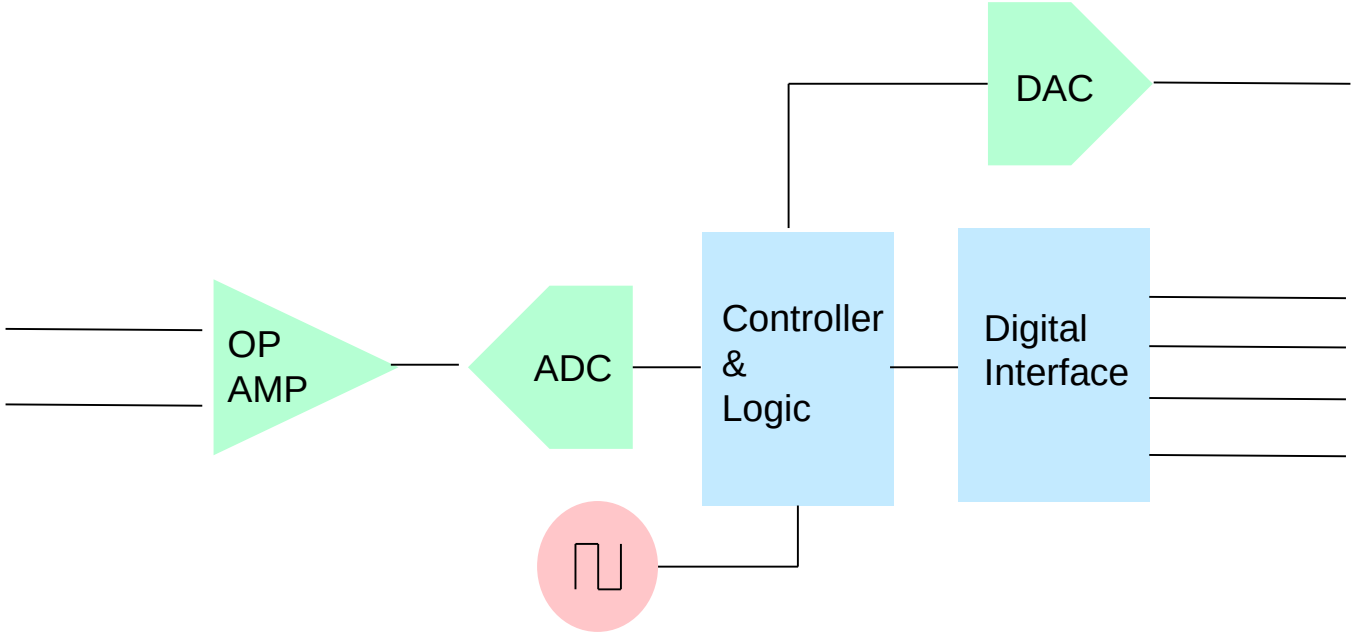
MEMS
Micro-Electro-Mechanical Systems

ASIC
Application-Specific Integrated Circuit

Silicon-based Sensors

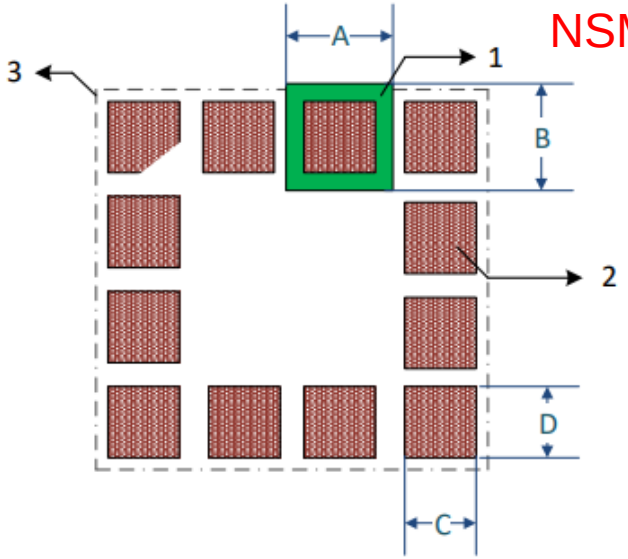


BJT
Bipolar Junction Transistor



ASIC
Application-Specific Integrated Circuit

Integrating sensors in PCB: Footprint



NSMD!

1. Solder mask opening
2. PCB land
3. Sensor package footprint

PCB land and solder mask recommendations for sensors with LGA package

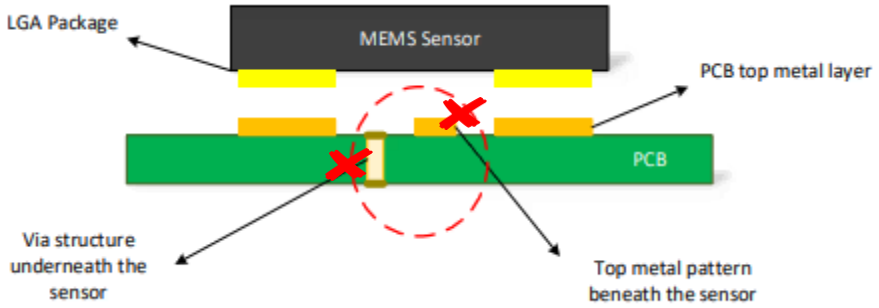
Dimension	LGA pad spacing > 200 µm	LGA pad spacing ≤ 200 µm
PCB land width: C	LGA solder pad width + 0.1 mm	LGA solder pad width
PCB land length: D	LGA solder pad length + 0.1 mm	LGA solder pad length

Table 1: PCB land design dimensions

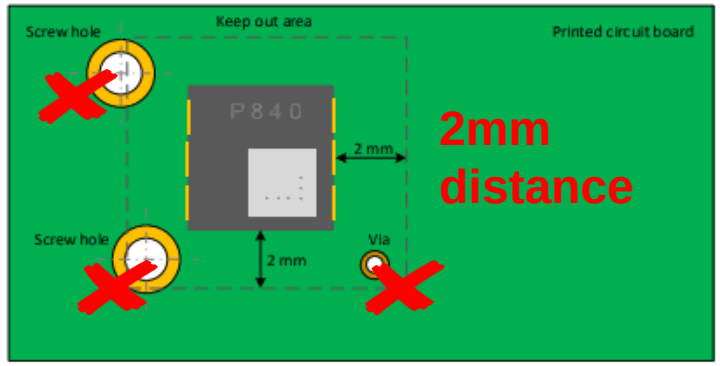
Dimension	Description
Solder mask opening width: A	PCB land length + 0.1 mm
Solder mask opening length: B (when applicable)	PCB land length + 0.1 mm

Table 2: Solder mask opening dimensions

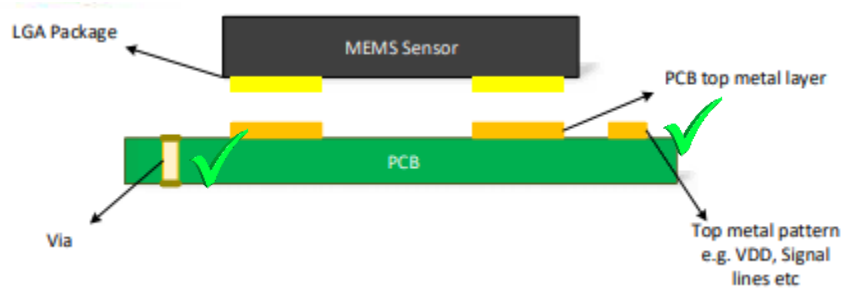
Integrating sensors in PCB: Keepout area



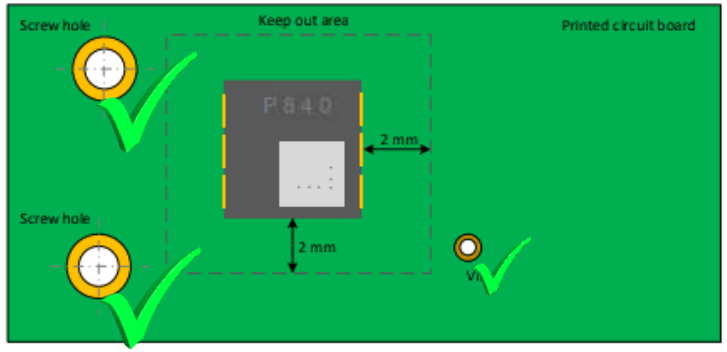
Incorrect PCB design



Components inside sensor keepout area

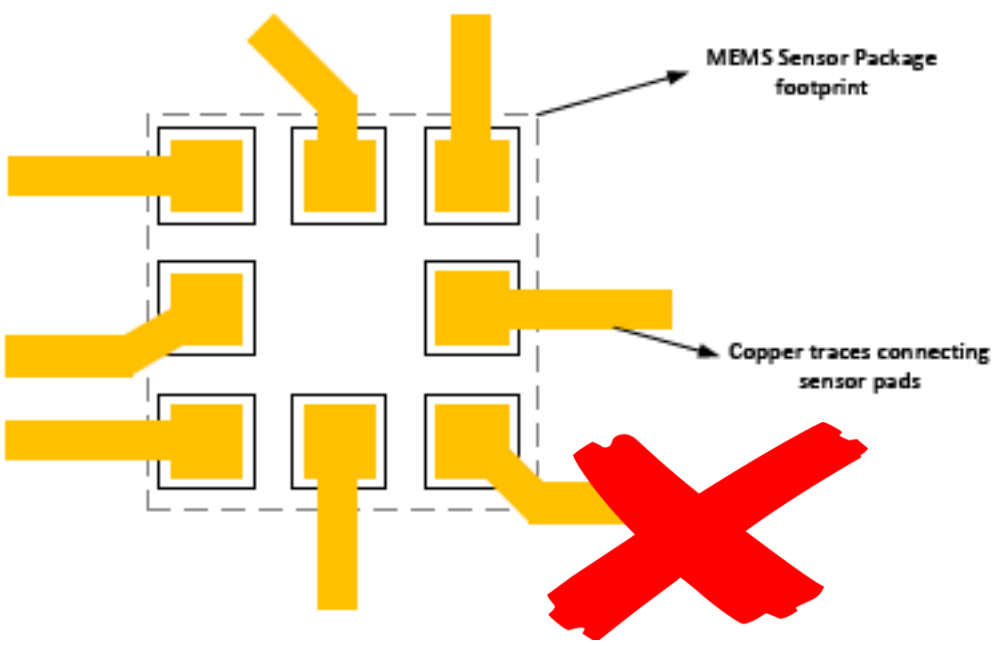


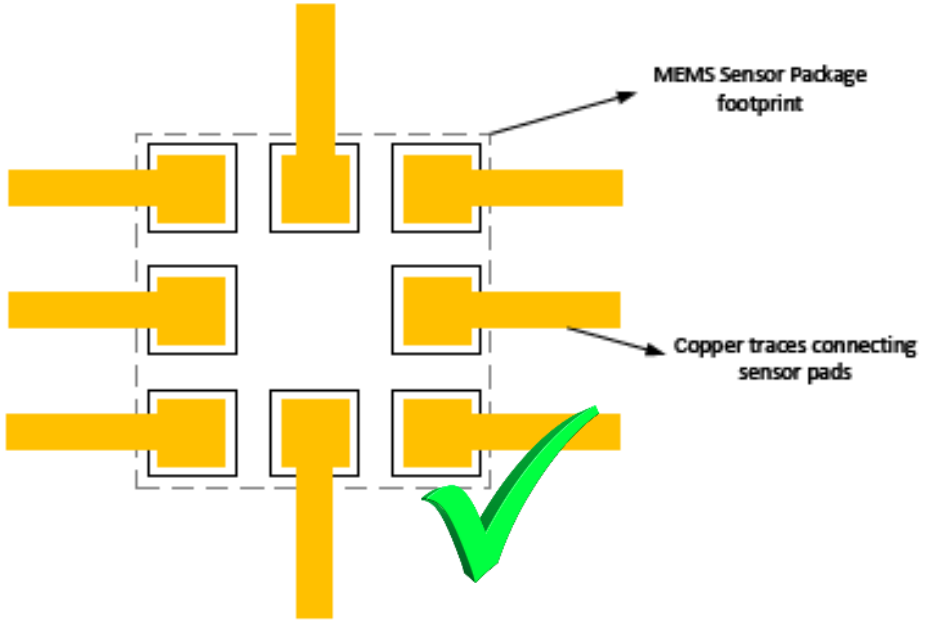
Components underneath sensor - keepout area



Components outside sensor keep out area

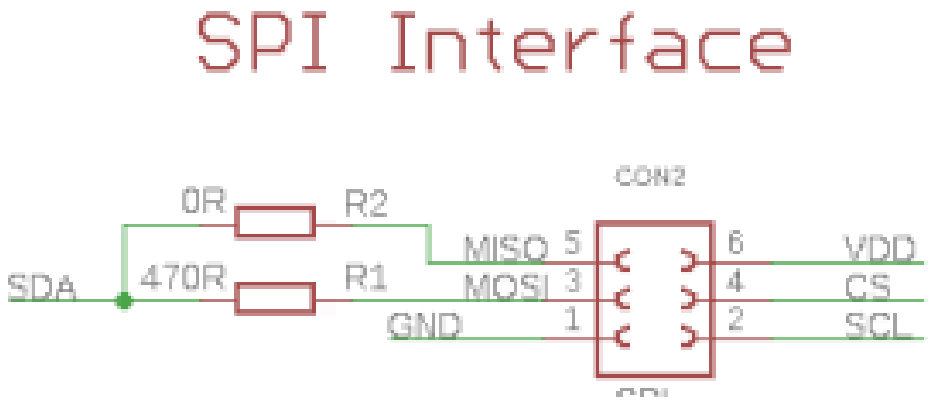
Integrating sensors in PCB: Layout hints



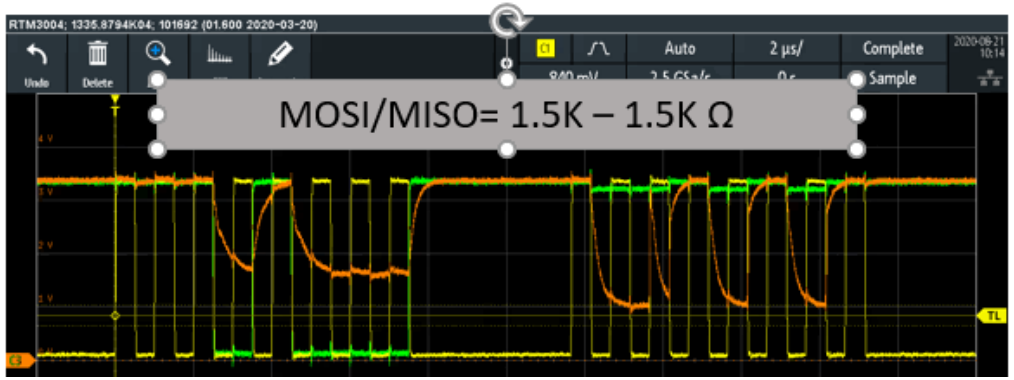


- The traces connected to the pads should be as symmetrical as possible
- Separate digital ground from analog ground in the PCB

Integrating sensors in PCB: Layout hints



- SPI/I²C trace shall be kept short and shall have similar trace length
- Propper GND plane beneath the signal traces
- Speed on the interface could be reduced to relax this issue
- Special case 3-Pin SPI humidity sensor 2525020210001

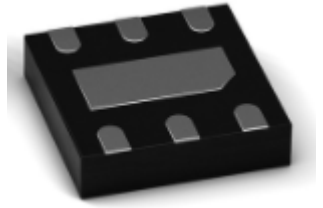


Soldering guidelines:

Stencil design and solder paste

For proper mounting process of the MEMS sensor, thickness and soldering paste pattern are very important.

- Stencil thickness of 90 - 150 µm (3.5 - 6 mils)
- Stainless steel stencils
- 70% to 90% stencil pad opening
- Trapezoidal and rounded corners aperture walls
- 25 µm (1 mil) alignment Stencil and PCB
- The final volume of the solder paste applied to a single PCB land should be less than 20% of the volume of the solder paste of all pads of one device



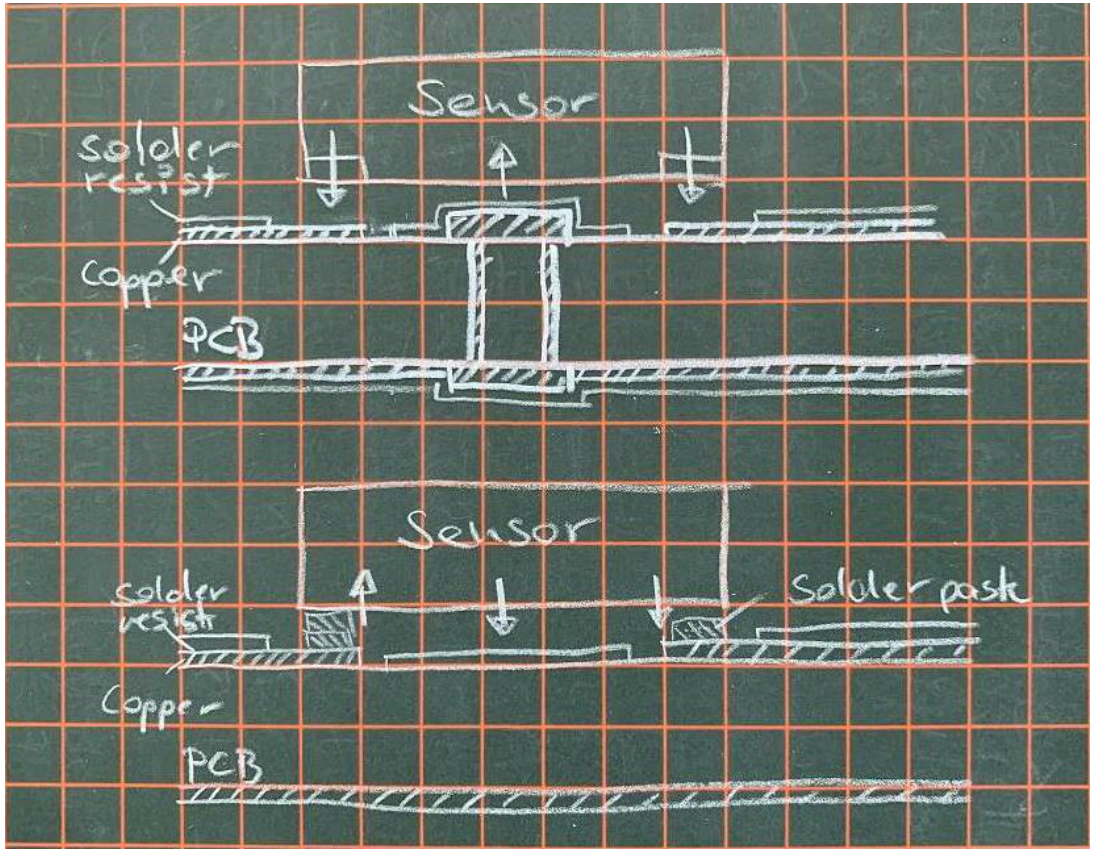
Soldering guidelines:
Stencil design and solder paste

- Sensor with opening surface on top should be handled carefully. Do not pick the component with vacuum tools which make direct contact with the opening of the sensor
- It is recommended to use a standard pick and place process and equipment./ Do not use the hand soldering process
- No-clean solder paste
- Rampdown < -3°C/s
- Avoid high-amplitude resonant vibrations

ANM001 - MEMS Sensor PCB Design and Soldering Guidelines

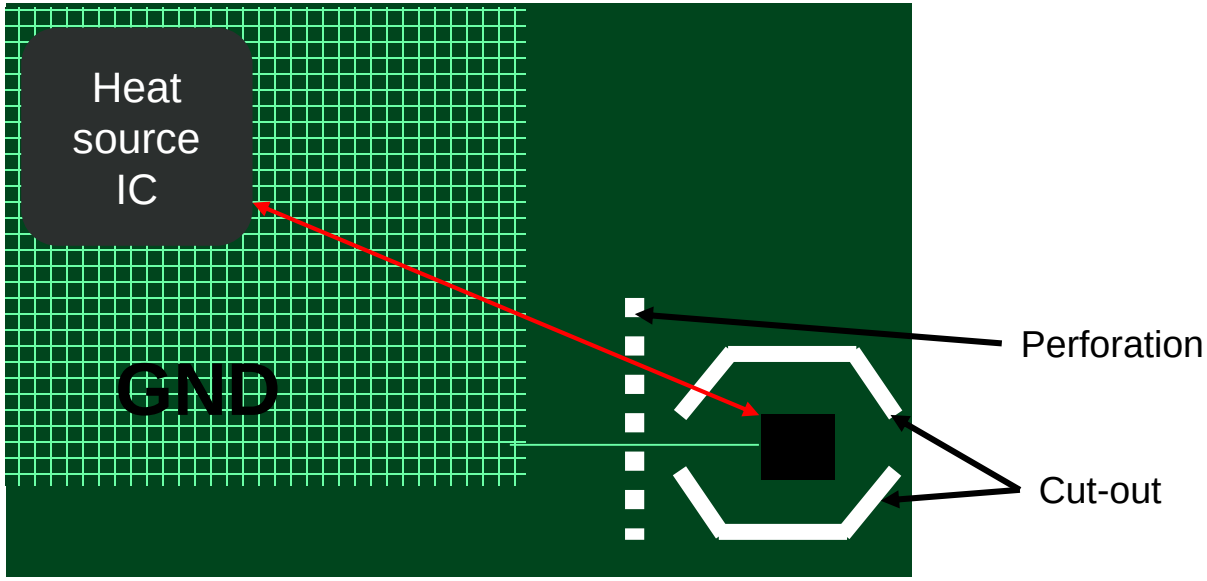
Explanation mechanical stress

- Different height between via capsule and „just“ copper
- Different thermal expansion of via/copper and PCB/FR4
- Different thermal expansion of solder paste and via/copper
- Solder paste thickness and amount must be the same at all Pads, else mechanical stress occurs

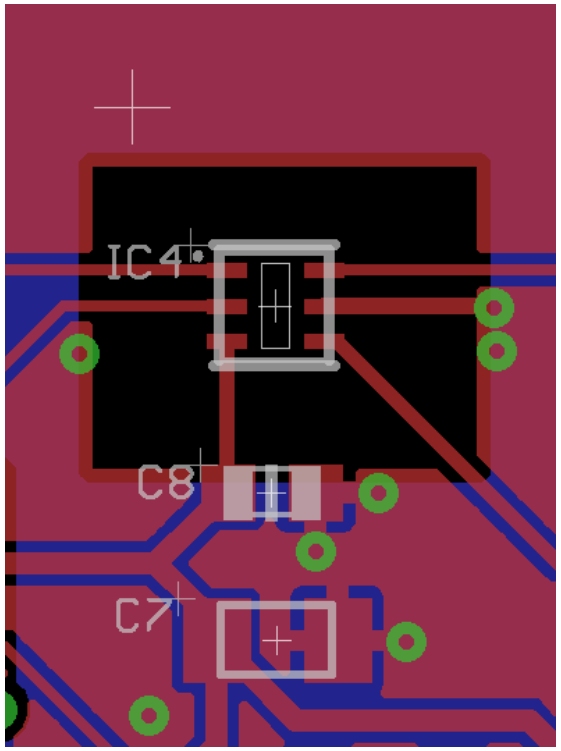


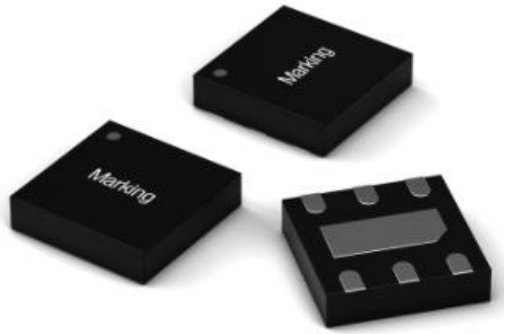
Avoid misuse: Temperature Sensor

Ambient temperature



- Sensing of environmental temperature
- Decoupling from heat sources on PCB



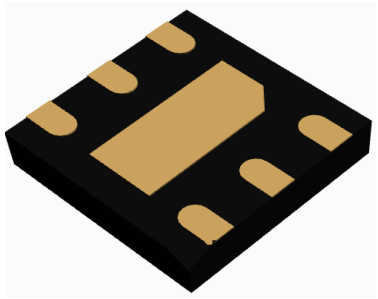


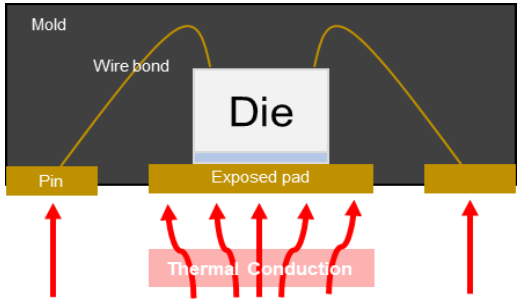
Avoid misuse:

Temperature Sensor

Component temperature

Leadless SMD package with exposed pad



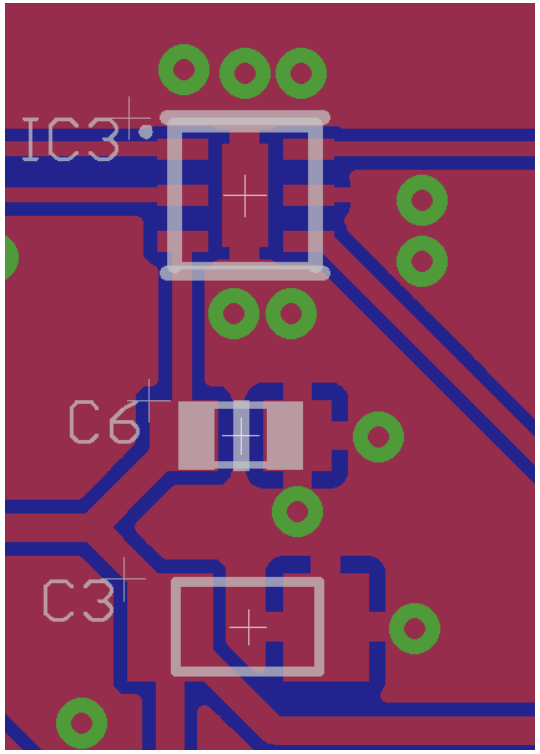


Heat Source IC ⟷ WSEN-TIDS

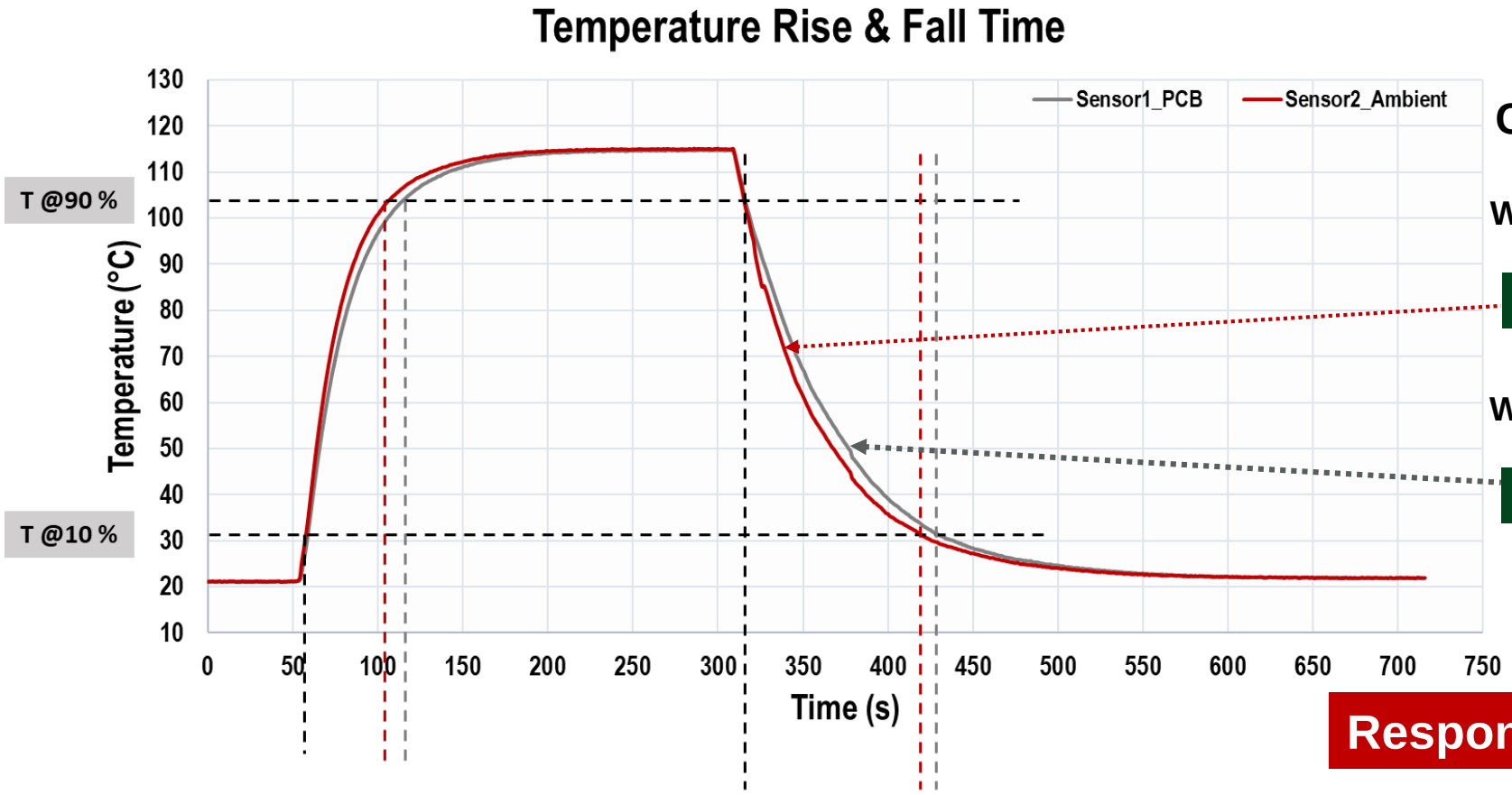
Heat conduction

Heat Source IC

- Sensing of temperature from chipsets on the PCB
- Coupling to that heat sources



Avoid misuse: Temperature Sensor



Cross section of the mounted Sensor

WSEN-TIDS Pad open to Air

WSEN-TIDS Pad to PCB Ground

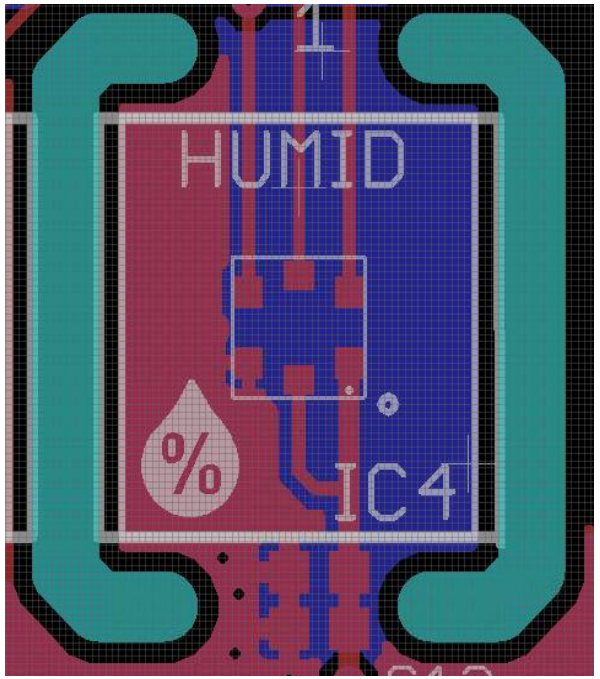
Response time ~7 seconds faster

- Sensor1_PCB = Thermal pad connected to **PCB Ground**
- Sensor2_Ambient = Thermal pad kept open to **air through PCB hole**

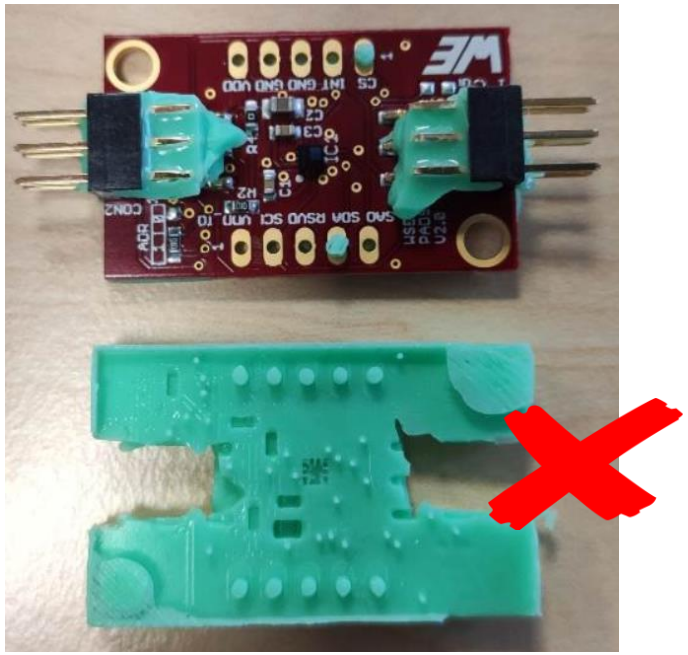
Avoid misuse: Humidity Sensor

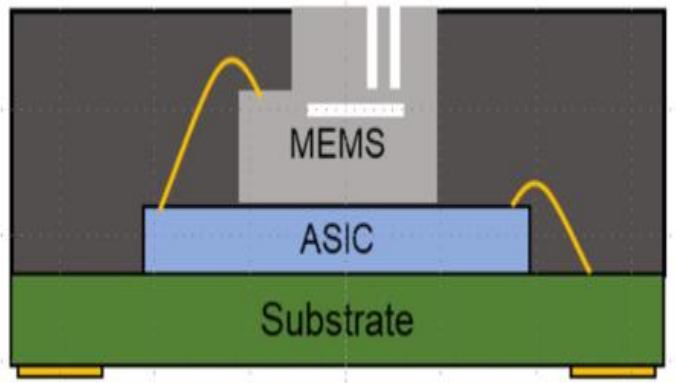
- Storage temperature: 10 °C to 40 °C & 20 rH% to 60 rH%
- Cut outs for air exchange, flow behavior and thermal decoupling is recommended

- **Recondition Process:**
  1. Baking: 100°C to 110°C at 5 rH% for 12h
  2. Re-hydration: 20°C to 30°C at 75 rH% for 12h



Avoid misuse: Mechanically





Summary

- In order to obtain the best performance of a sensor, best practice in the design phase need to be considered. For instance: proper footprint design, no routing and no mechanical holes underneath the sensor, ground separation and short traces
- High speed digital interface may pose a challenge due to stray capacitance. Good signal quality can be obtained by reducing the length of the traces and decreasing pull-up resistor
- When measuring ambient temperature it is recommended to connect the exposed pad to the bottom side of the PCB by means of vias. Additionally, cut-out areas or/and perforation can be added to increase thermal isolation.
- In case of measuring the heat produced by a component in PCB, the exposed pad of sensor and the exposed pad of the target unit should be connected with minimum possible distance
- Silicon membrane on the top should not be touched or obstructed

Wireless Connectivity & Sensors - Product guide

Thank you!