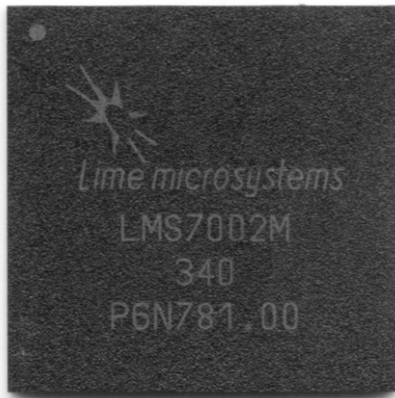


LMS7002M

Multi-band Multi-standard Transceiver – PCB layout

Purpose

This document provides PCB design guidelines for LMS7002M assembly, based on the industry standard aQFN261 type package. It outlines the package dimensions and the PCB design rules used on the LIME evaluation platforms and reference boards. It also provides for two footprint options for reliable and reproducible manufacturing for a given application. The instructions are meant as a guideline only and may need to be modified depending on specific PCB manufacturing processes.



Package description

The LMS7002M comes in a 261 pin advanced QFN (aQFN) 11.5 mm × 11.5 mm package. The pads are round and 0.27 mm in diameter with 0.666 mm pitch. There are 260 signal, power and ground pins with critical RF pins routed out to the outer rows. A centre ground pin is used for electrical grounding and thermal cooling. The package height is 0.85 mm.

Package dimensions

Please refer to *Figure 1* for package dimension and measurements.

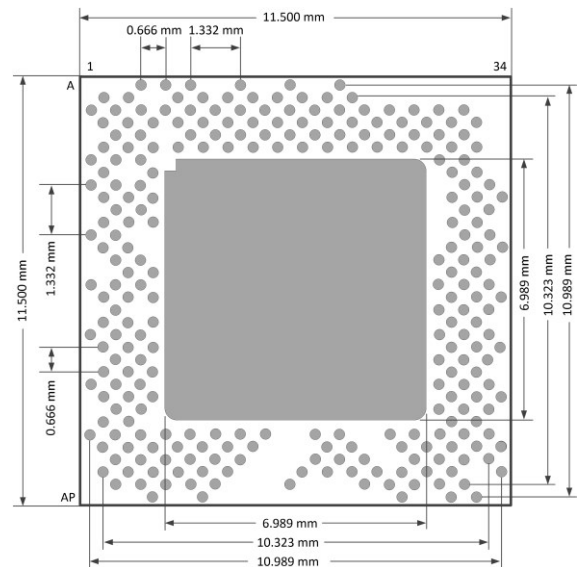


Figure 1 LMS7002M land pattern

PCB pattern

As shown in *Figure 2*, the PCB pattern contains 260 identical round 0.35 mm diameter pads and a thermal ground pad. Detailed dimensions on the land pattern are covered in *Figure 3*.

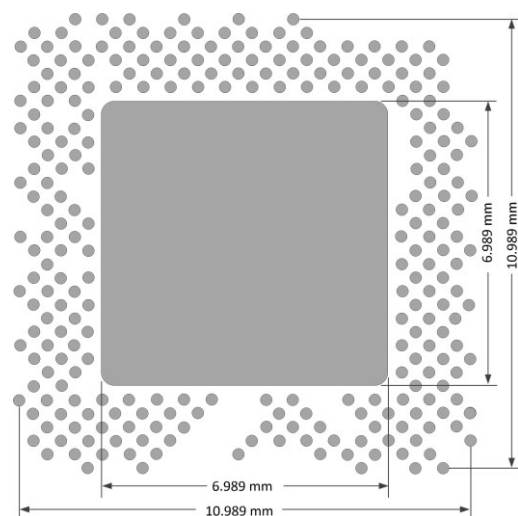


Figure 2 LMS7002M land pattern

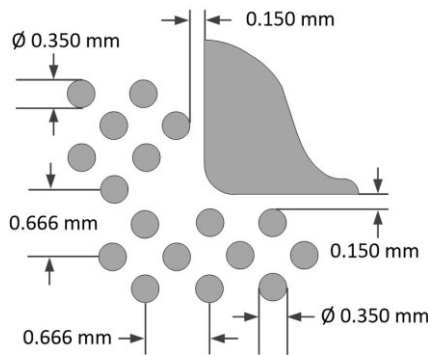


Figure 3 LMS7002M land pattern – detail

Note: In the case of LMS7002M Via in Pad land pattern it is advised to use resin filled vias with metal cap. LMS7002M thermal ground pad vias on the other hand have no such requirements and are simple through vias.

Solder resist

The solder resist clearance is 0.05 mm as shown in Figure 4.

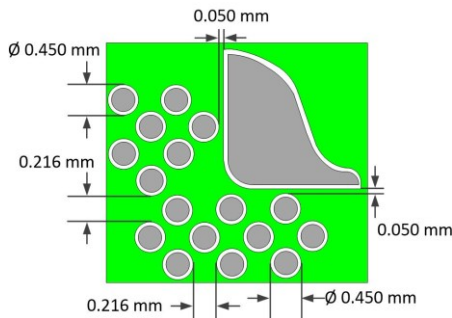


Figure 4 Solder resist clearance – detail

Solder paste

The solder paste mask is the same size as the IC pads with the exception of the large centre pad where a hatched pattern is used to reduce the amount of solder paste on the centre pin, as shown in Figure 5.

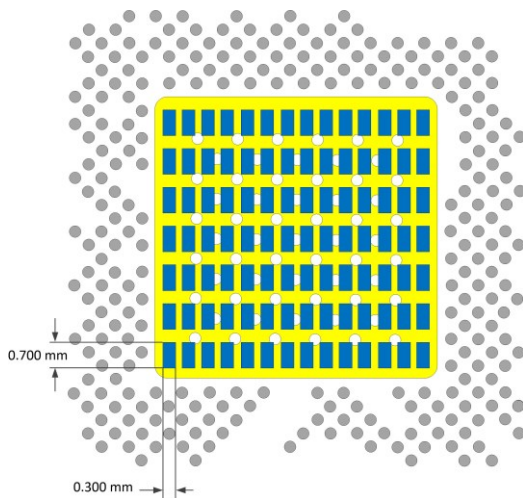


Figure 5 Centre pad solder paste

Solder paste grid used on the centre pad is a grid of 14x7 0.3 mmx0.7 mm rectangles (approx. 46 % coverage).

Thermal ground pad

The centre pad of the aQFN261 package is used for both electrical ground as well as providing the required thermal path. There should be both a low inductance path to the RF ground plane of the PCB as well as a low thermal resistance to copper planes which are used to dissipate the heat. It has been found that with proper design of PCB, no additional heat sinking is required to cool the LMS7002M IC.

A pattern of 61x0.2 mm through Vias (as shown in Figure 6) are used on the centre pad to provide for the mentioned above LMS7002M heat dissipation.

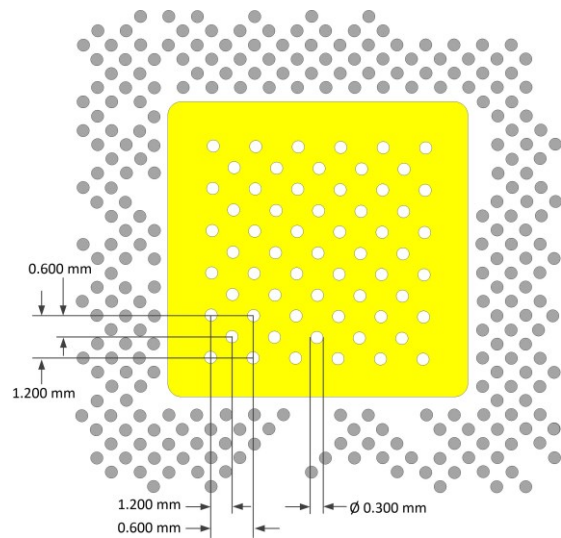


Figure 6 Centre pad thermal / ground Vias

It is important to note, that the Vias on the other/back side of the PCB are connected to a copper ground plane which in turn allows for the heat to radiate from the PCB surface. The copper area on the back should be larger than the thermal pad on the device; if possible the back of the PCB should be copper filled to dissipate as much heat as possible.

Inner pins and routing

The inner rows of pins of the LMS7002M carry no RF critical signals but do need to be routed away from the device. Due to the location of the centre ground pad this means that Via in pad are required.

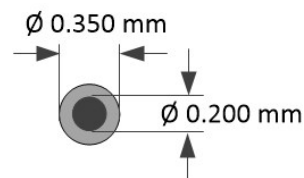


Figure 7 Via in pad

The LIME design uses a 0.2 mm drilled Via with a 0.2 mm annulus, as shown in Figure 7. It uses a 6 layer board, layer 1 (top) for RF tracks and routing, layer 2 for ground and layers 3, 4, 5 and 6 (bottom) are for routing. The Vias in the pads of the LMS7002M footprint are routed to the bottom layer. The Board has a 0.172 mm layer 1 to 2 height and a 0.076 mm layer 2 to 3 prepreg.