Printed Circuit Board Design for LSM1 VCO's

Rev. V4

Introduction

To achieve the best performance from M/A-COM's LSM1 VCO series, it is critical that good design practices are used in the layout of the printed circuit board. This application note describes the pad footprint recommended for solder attachment and some of the electrical and thermal considerations to incorporate into the pcb design.

Footprint Guidelines

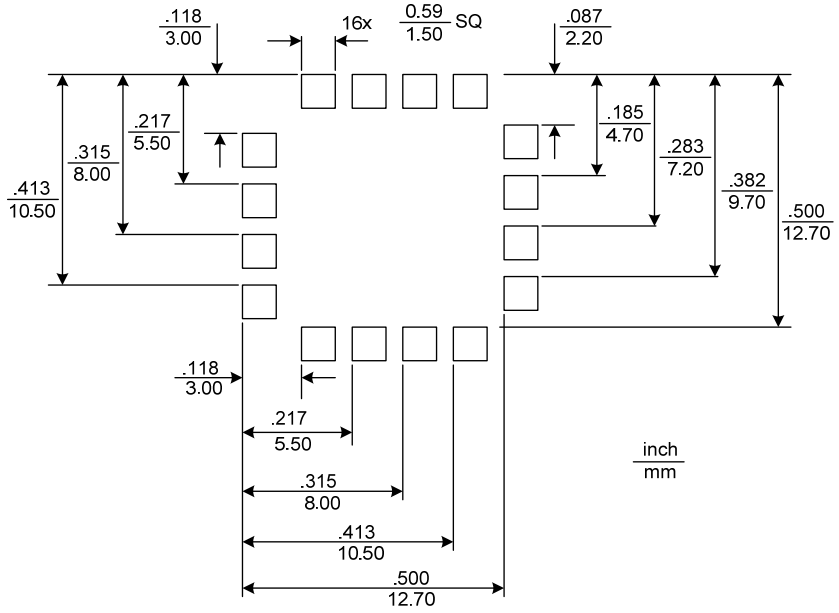
The pad footprint defined on the pcb must be the correct size to ensure a proper solder connection interface between the board and the package. With the correct pad geometry, the packages will self align when subjected to a solder reflow process and will also allow for just enough excess surface area for adequate solder filleting. The following pad footprint is a recommended guideline only and may require additional ground plane areas for electrical and/or thermal considerations. When an increase in pad or ground plane size such as this is implemented, the additional copper area should be covered by a solder resist mask so as to leave only the recommended pad footprint available for attachment. This will contain the solder reflow and eliminate any alignment problems during reflow soldering.

Electrical and Thermal Considerations

The circuit board used in the LSM1 VCO package is FR-4 material with a thickness of 0.8 mm (0.032 inch). The package interconnection pads are 1 oz. copper with a finish of 0.15 µm gold flash over 5 µm nickel plate.

The electrical performance of the VCO may be improved by extending the ground plate area beyond the pad footprint. For example, a continuous ground plane under the base of the VCO linking up all of the defined ground areas will improve grounding of the VCO and reduce phase noise. It is important to cover any additional pcb ground area with a solder resist mask so as to maintain the recommended attachment footprint. Depending on the output frequency of the VCO, additional via holes may also be necessary in the ground plane of the pcb to minimize unwanted ground reactance.

The RF output track on the circuit board should be designed as a 50 ohm impedance microstrip line where the width of the line is calculated in accordance with the dielectric constant and thickness of the pcb used. The RF output should operate into a 50 ohm load with VSWR less than 1.5:1 at a minimum distance from the VCO.



LSM1 Footprint

Printed Circuit Board Design for LSM1 VCO's

Rev. V4

To tolerate the impedance mismatch introduced by components such as mixers and prescalers some form of isolation is needed at the RF output. This can be achieved by using a 10 dB pad and buffer amplifier. Insufficient isolation of the VCO output may result in output power variations, degraded phase noise performance and increased output frequency pulling.

The LSM1 VCO design incorporates a bypass capacitor on the Vcc input to suppress supply generated noise. Additional filtering is recommended to reduce supply noise and its effect on VCO phase noise performance. Care should also be taken to minimize the introduction of noise from the tune voltage input by additional filtering as necessary.

Electrical and/or thermal considerations may also require the board to contain plated through holes located under the pad of a surface mount package. When this is necessary, the recommended plated through hole diameter is 0.25 to 0.38 mm (0.010 to 0.015 inch). This diameter range, along with the solder's viscosity, helps to prevent solder flow down these holes. Although a 0.8 mm (0.031 inch) diameter hole is typically used, holes this size will drain solder away from the base of the package and will make solder paste thickness more difficult to control.

Thermal resistance through the circuit board will be a function of the pcb material and thickness, the quantity and location via holes, the proximity of any additional thermal loads and whether forced air cooling is employed. Thorough testing of the prototype is the only way to prove the adequacy of a thermal design. In general these VCO's can be considered as low power dissipation components and no special precautions need to be taken.