

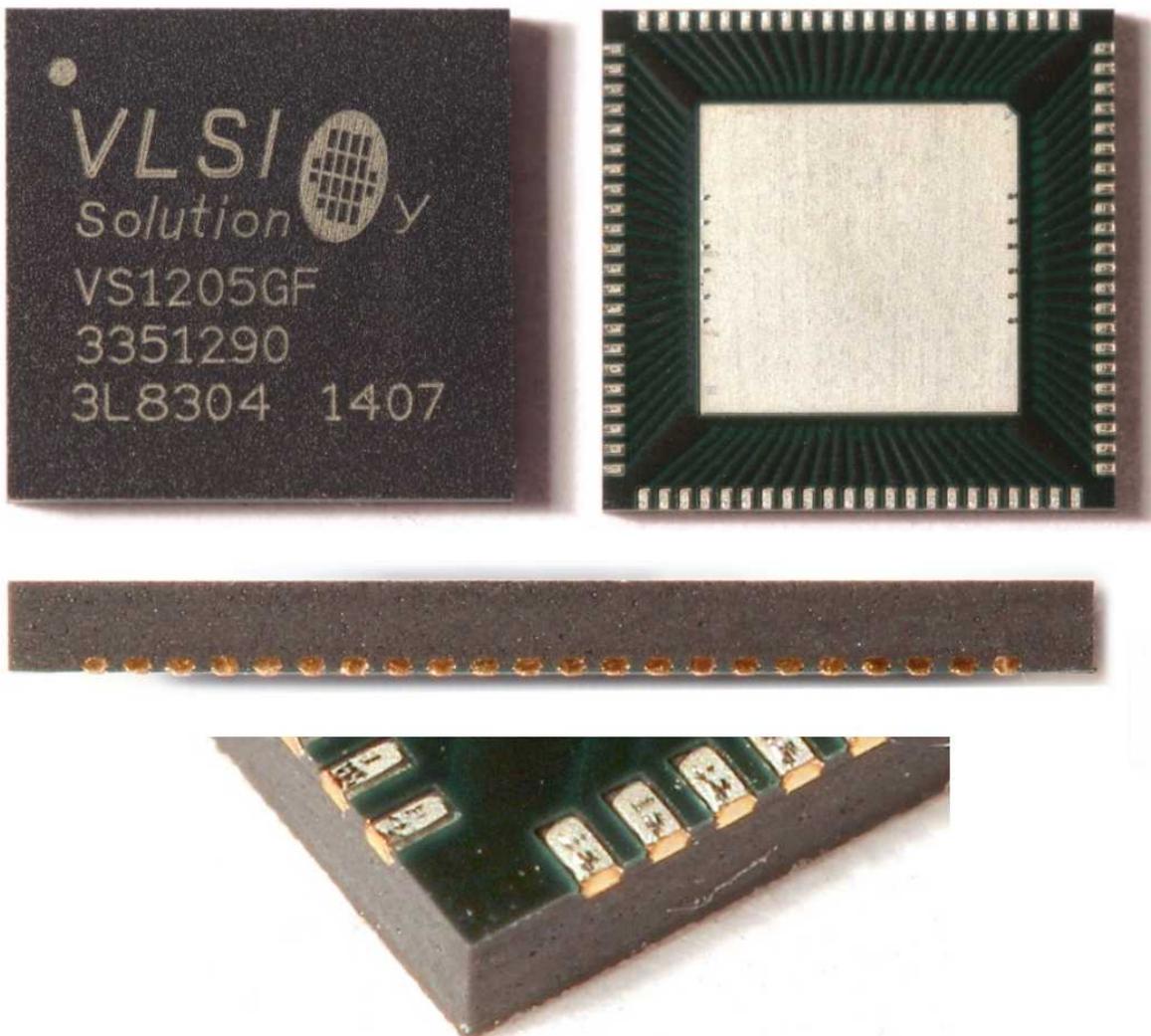
VS1005 Soldering Guide

1. Introduction

This document provides a short guide to the LFGA-88 PCB land pattern desing and soldering parameters. More information about fine pitch SMD soldering can be found from the references at the bottom of this guide.

2. VS1005 mechanical dimensions:

The LFGA-88 is a 10x10x0.8mm near chip scale plastic encapsulated package whose contacts extend to the sides of the package (VS1005g datecode 1407 and newer). Detailed infromation about mechanical dimensions can be found at <http://www.vlsi.fi/en/support/download.html> .



3. Recommended land pattern:

Special attention should be paid in land pattern and PCB design for optimal thermal and electrical performance and solderability. Figure 2 shows recommended land pattern for VS1005g datecode 1407 and newer. An example footprint can be also obtained from VLSI's support page in Eagle library file.

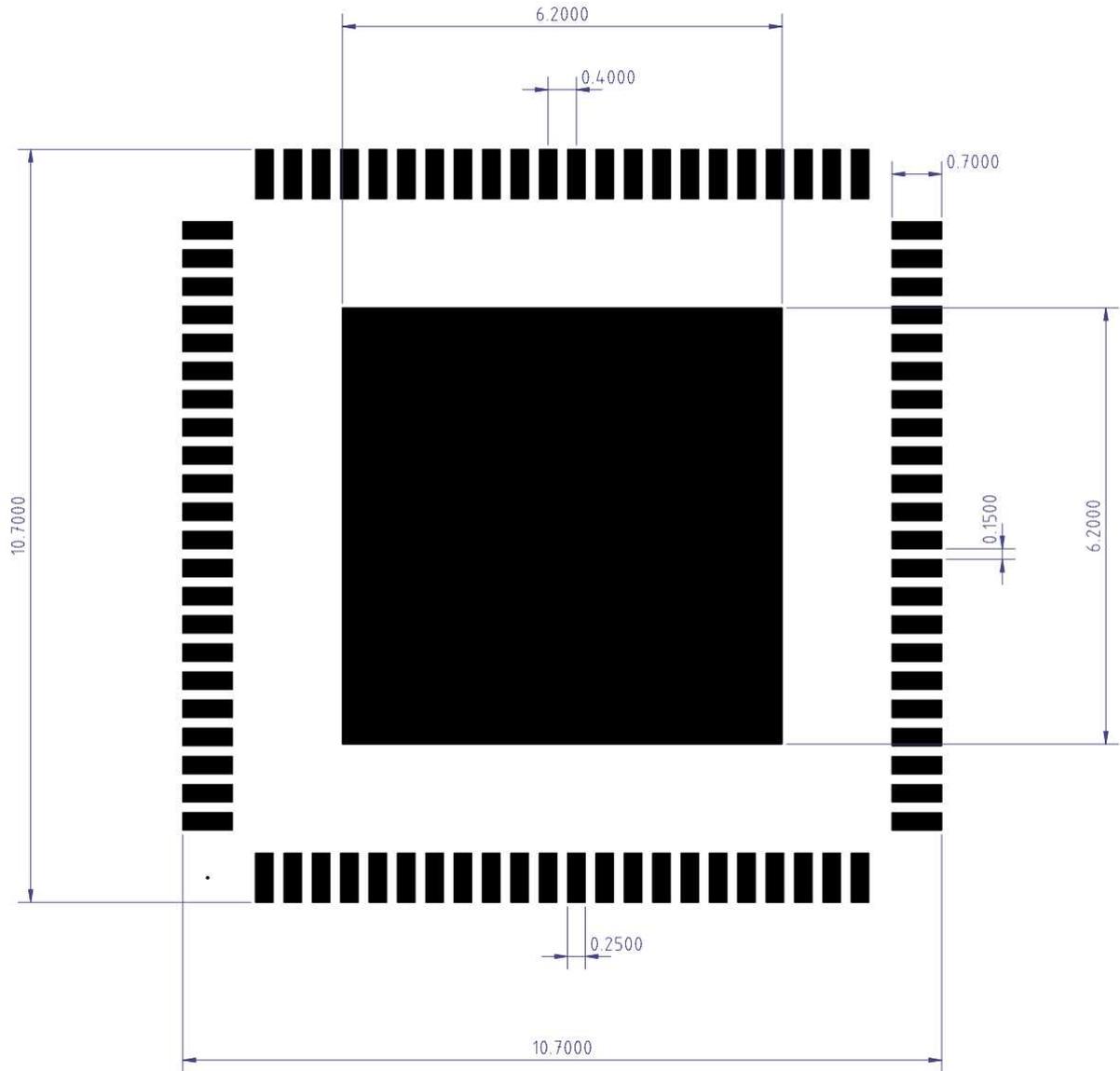


Figure 2. Recommended footprint. All dimensions in millimeters.

Most common mistake is not to connect the center pad to ground plane with multiple vias. Vias under the center pad should be plugged to prevent solder escaping through the PCB by capillary effect. Generally accepted drill diameter for 1.6mm thick FR4 material is 0.3mm. The via tenting can be done from bottom or top side of the board.

The PCB traces should be routed only orthogonal to the IC (except the outermost pads). Figure 3 explains the correct and incorrect way to connect traces with same net to the footprint.

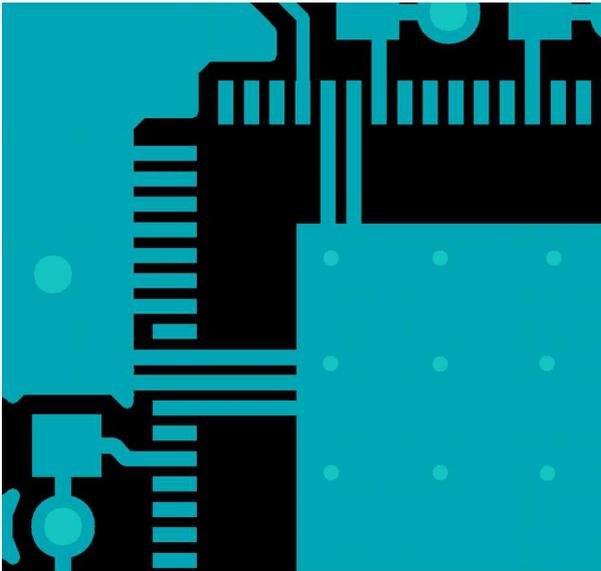


Figure 3a. Correct connections.

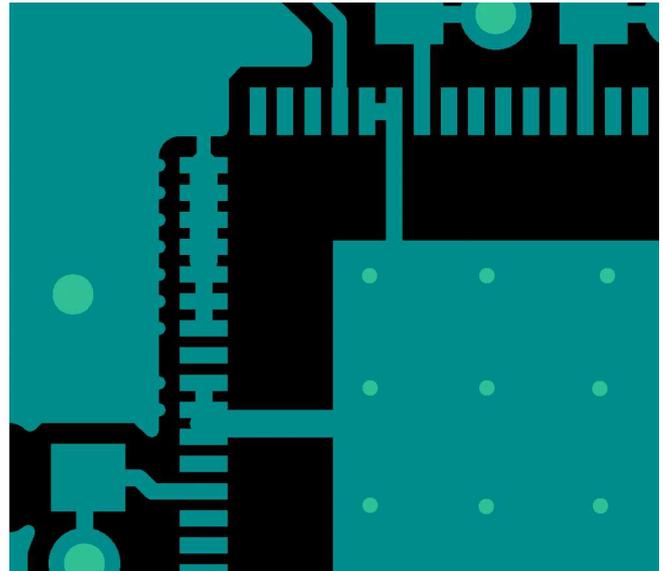


Figure 3b. Incorrect connections.

The soldermask opening should be 50-100 μm larger than the copper pads. In most cases trench type opening [1] has to be used for the side contacts. Figure 4 shows typical solder mask opening for VS1005 family chips.

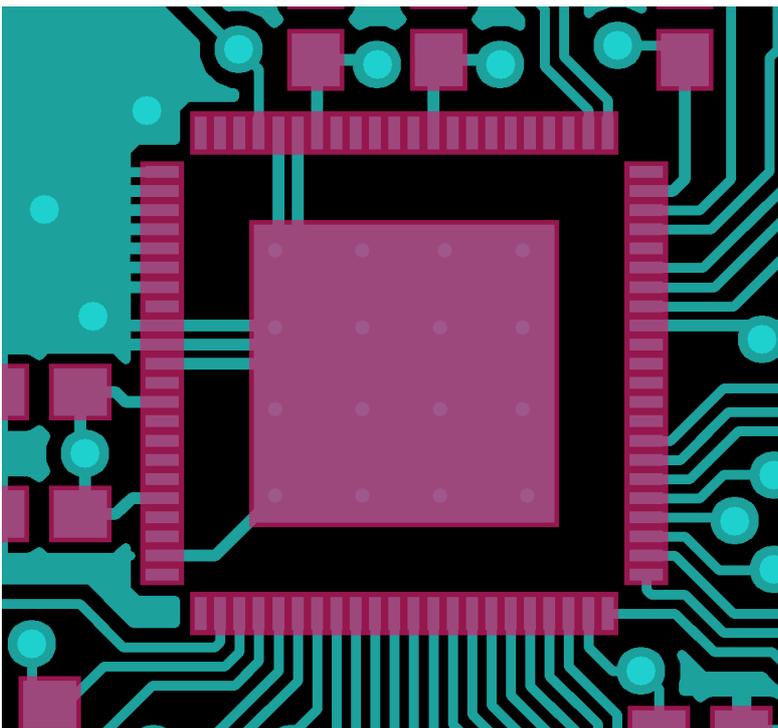


Figure 4. Solder mask opening.

3. Stencil and paste type:

Good starting point for stencil desing is 1:1 solder paste coverage for side contacts and over 50% coverage for the center pad [2]. There should be also clear paths for the gases to exit during reflow process. A 3x3 or 4x4 screen is recommended. Stencil thickness should be 100 μ m. Figure 5 illustrates example desing of the solder paste layer for VS1005.

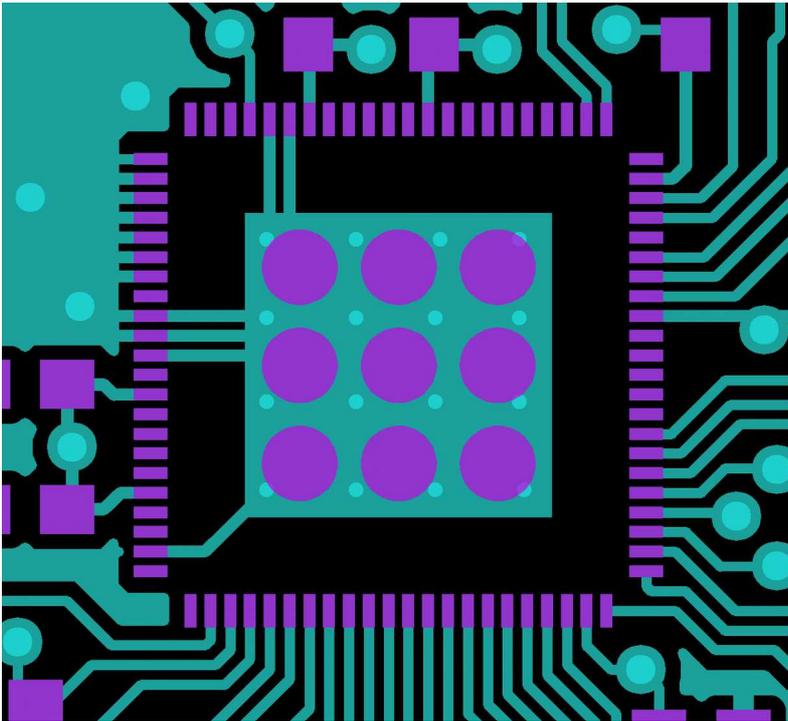


Figure 5. Example stencil opening.

Size of the stencil openings should be reduced if thicker stencil has to be used. Too much solder paste can easily result in short circuits beneath the chip. Figure 6a shows the correct amount of solder paste and in 6b too much paste has been used.

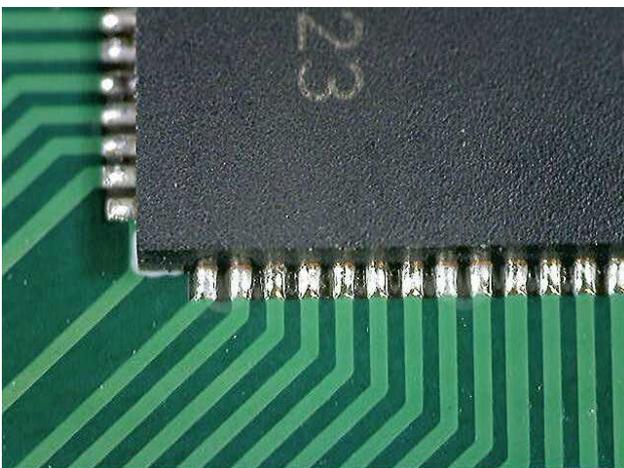
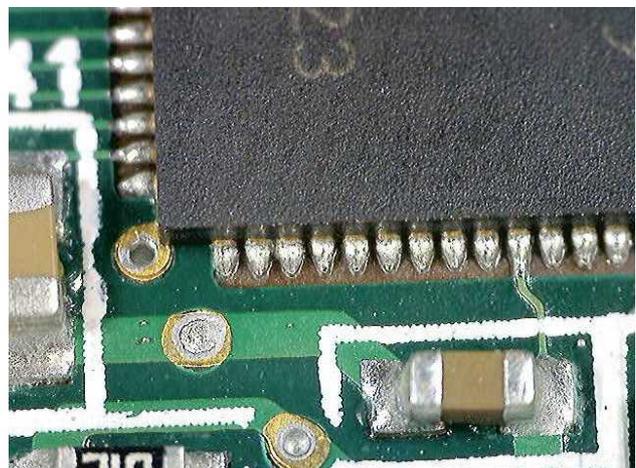


Figure 6a. Correct amount of solder paste



6b. Too much solder paste

Solder paste consistency should be as thick as possible to prevent short circuits from forming in reflow phase. Solder paste type SAC305 and grain size 4 has been proven to produce good solderability.

5. Reflow oven profile:

A typical reflow oven profile is shown in table 1. Actual parameters depends upon used solder paste, PCB type and other components used. In addition to carefully tuned temperature profile a five stage reflow oven and nitrogen atmosphere are required for optimal results.

	Ramp up	Preheat	Reflow	Peak	Peak temp.
Temperature	25-160 °C	160-190°C	>220°C	230°C	235-245°C
Time	90-130s	30-60s	20-50s	10-15s	

Table 1. Typical reflow temperature profile.

6. References:

- [1] QFN Package Mounting Guidelines, 8583A, Atmel Corporation,
URL: www.atmel.com/images/doc8583.pdf
- [2] QFN Mounting Manual, R50ZZ0005EJ0150, Renesas Electronics Corporation,
URL: https://www.renesas.com/ko-kr/doc/products/others/r50zz0005ej0150_pkg_qfn.pdf