AN-2007
APPLICATION NOTE

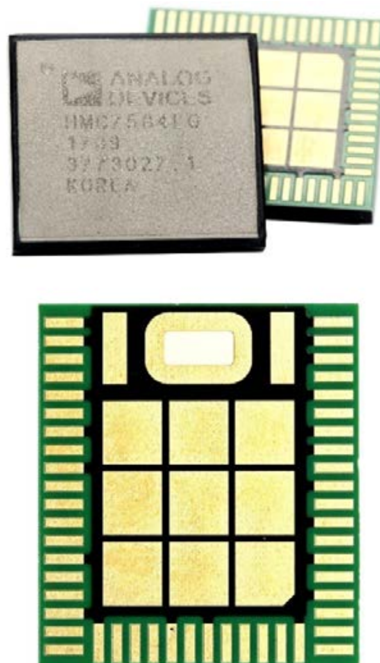
One Technology Way • P.O. Box 9106 • Norwood, MA 02062-9106, U.S.A. • Tel: 781.329.4700 • Fax: 781.461.3113 • www.analog.com

Considerations for Handling and Soldering E-Band SiP onto Printed Circuit Boards

by Zeyan Huang and John Poelker

INTRODUCTION

The HMC7584LG, HMC7585LG, HMC8326LG, and HMC8327LG are fully integrated system in package (SiP) devices, which are ideal for E-band communication systems, high capacity wireless backhauls, and test and measurement applications. All four devices are available in an air cavity land grid array (LGA) package that can be soldered onto a printed circuit board (PCB). It is important to handle the device with care during assembly to minimize stress and damage. This application note discusses the recommended practices for handling and soldering these devices onto PCBs.



20294-001

Figure 1. HMC7584LG Physical View

TABLE OF CONTENTS

Introduction 1
Revision History 2
Package Description 3
PCB Design 5
Assembly Considerations 7
Rework 8
Reliability Considerations 9
Mechanical Considerations 10

REVISION HISTORY

5/2019—Revision 0: Initial Version

PACKAGE DESCRIPTION

The HMC7584LG, HMC7585LG, HMC8326LG, and HMC8327LG are available in an air cavity LGA package. Inside the LGA package, a high frequency laminate circuit is combined with multiple millimeterwave ICs and a waveguide to microstrip transition to provide a complete E-band upconverter or downconveter solution. The package is nonhermetic and covered by a nickel plated lid (see Figure 2 and Figure 4). The package also has a substrate vent hole to allow moisture out gassing and to avoid high internal pressure during soldering and reflow (see Figure 3 and Figure 5). This substrate vent hole is solder sealed during the surface-mounted technology (SMT) reflow. The lid is attached to the laminate substrate with a conductive epoxy. A secondary nonconductive epoxy is applied around the package base to improve mechanical support.

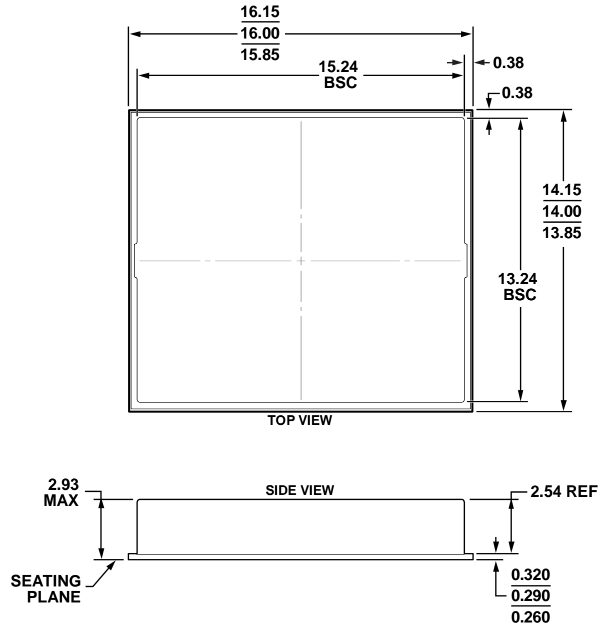


Figure 2. Top and Side View of the HMC7584LG and HMC7585LG

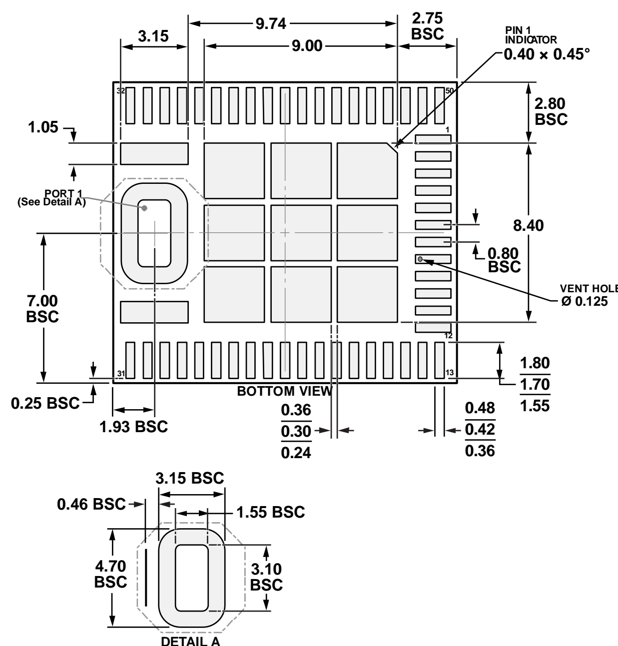


Figure 3. Outline Drawing of the HMC7584LG and HMC7585LG

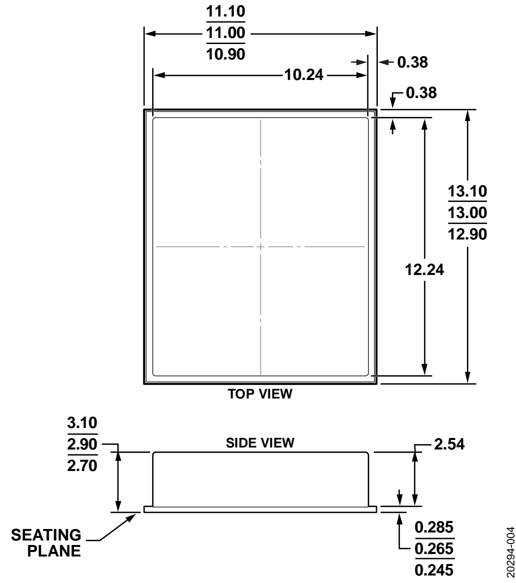


Figure 4. Top and Side View of the HMC8326LG and HMC8327LG

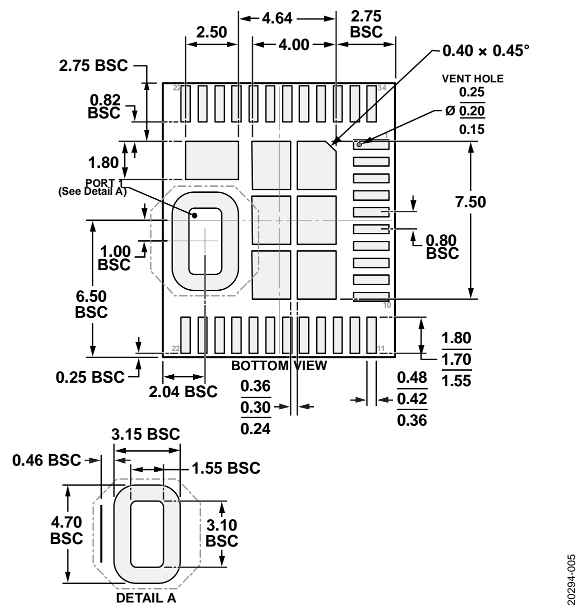


Figure 5. Outline Drawing of the HMC8326LG and HMC8327LG

PCB DESIGN

When designing a PCB with E-band SiP devices, follow the PCB footprints in Figure 6 and Figure 7. Use a high frequency, low loss dielectric material, such as Rogers 4350, under the metal layer to achieve optimal radio frequency performance. For an optimal ground connection, fill the ground pad with an array of vias as shown in Figure 6 and Figure 7. It is recommended to fill the vias with epoxy to avoid solder voids in the ground paddle. The suggested board finish is electrical nickel immersion gold (ENIG), and the recommended starting copper thickness is 18 µm (0.5 oz copper).

Follow both the soldermask and paste mask designs in Figure 6 and Figure 7. Separate the ground paddle into sections to help minimize the voids within the exposed pad interconnections.

The standard WR-12 waveguide opening must be fully edge plated. For optimum performance, the recommended radius of the waveguide slot corner is 254 µm. Do not use thru vias instead of edge plating for the waveguide opening.

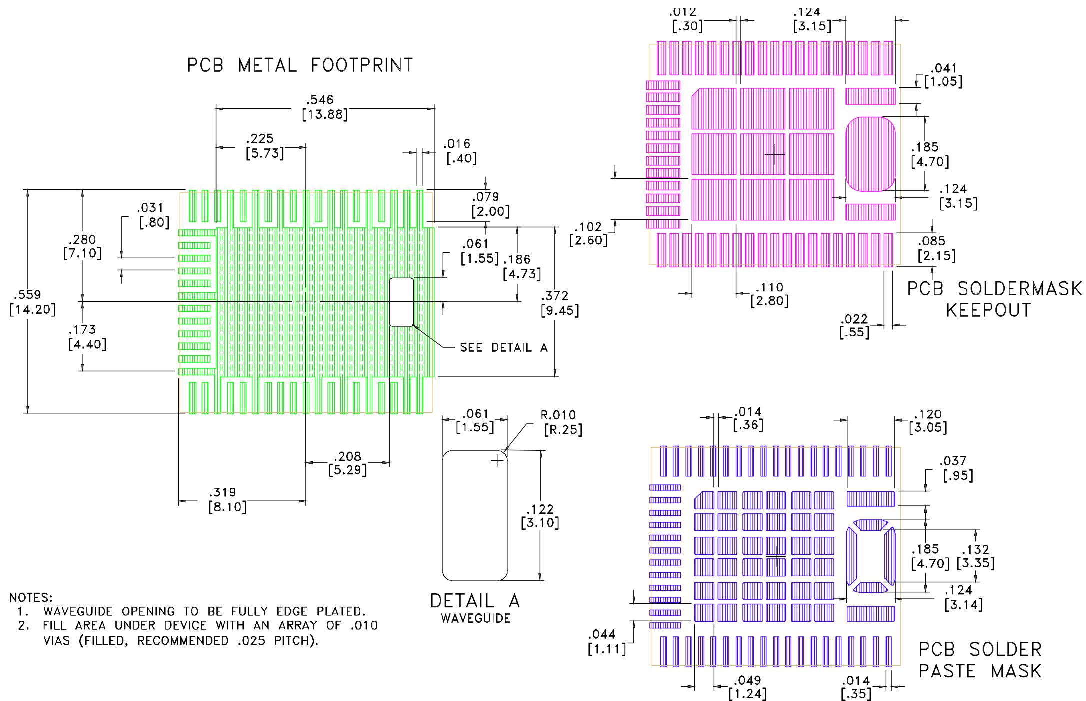


Figure 6. Recommended PCB Land Pattern, Soldermask, and Paste Mask Design Using the HMC7584LG and HMC7585LG

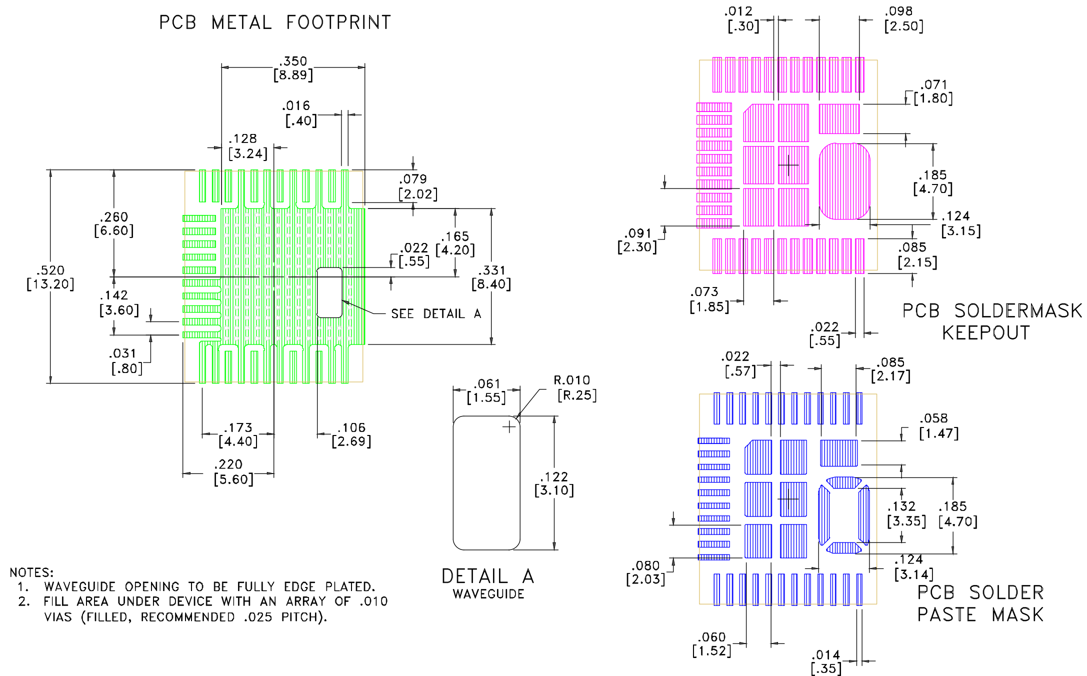


Figure 7. Recommended PCB and Land Pattern, Soldermask, and Paste Mask Design Using the HMC8326LG and HMC8327LG

ASSEMBLY CONSIDERATIONS

Place the device with equipment that minimize feeder vibration. Precision pick and place equipment is preferred.

Avoid chop or snap processes during PCB singulation. Clean debris and dirt from the PCB before assembly.

For the solder printing processes, the suggested stencil thickness is 102 µm based on the PCB footprint drawings in Figure 6 and Figure 7. The recommended solder paste is CVP-390.

The HMC7584LG, HMC7585LG, HMC8326LG, and HMC8327LG cannot self align during soldering due to the weight of the device. High precision alignment is necessary for attaching the device to the PCB. Vision systems in assembly machines can help precisely place the device.

A typical reflow profile for the device is shown in Figure 8. The peak temperature is 245°C, and the time above liquidus (TAL) is 65 seconds. Do not reflow the device more than three times in total.

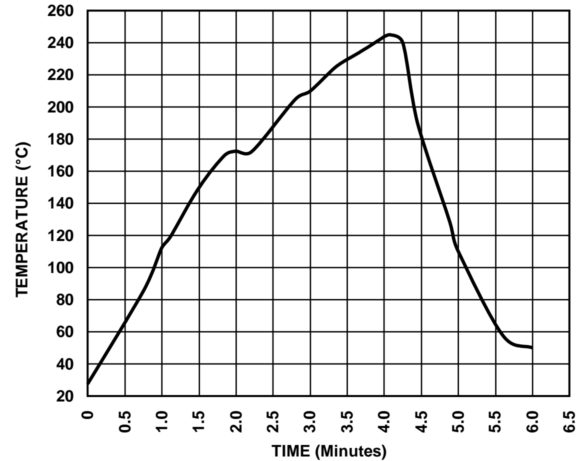


Figure 8. Typical Reflow Profile

If the peak reflow temperature settings must be changed, the settings must not exceed the maximum temperature, 260°C, for which the device is qualified. Lower peak temperatures can require longer TAL time.

REWORK

If defects are present, rework to remove the defective device and replace the device with a functioning one. The removed device must not be reused.

The conventional rework procedure is as follows:

1. Board preparation
2. Component removal
3. PCB clean up
4. Application of solder paste
5. Component alignment and placement
6. Component attachment
7. Inspection of rework

Careful removal of the device is critical, especially if the device is subjected to failure analysis. Avoid damage to the PCB, nearby components, and the device itself.

Reflow the solder joints before the device is lifted from the PCB. Follow the temperature profile used for the device attachment when performing the rework. To reduce the probability of damaging the device during reflow, apply heat to both the top and bottom of the PCB. Heating only the top side of the PCB or lifting the device before the solder joints are fully reflowed may damage the device.

RELIABILITY CONSIDERATIONS

Handle the device with proper moisture sensitivity level (MSL3) guidelines. Otherwise, trapped moisture can cause device failure.

The application board design must employ proper thermal management to achieve the data sheet specified mean time to failure (MTTF). Avoid thermal gap filler pads at the package cover. Thermal gap filler pads can weep silicone oil during temperature cycling, which can compromise the SiP lid epoxy seal.

MECHANICAL CONSIDERATIONS

Excessive mechanical stress can occur in different processes and handling steps. Practice the following procedures to help reduce device failure:

- The devices are stored and shipped in trays. Leave the devices in their original packaging until used. Do not store bulk devices in bins or expose devices to rough handling.
- The board mounting surface must be planar to avoid any mechanical stress.
- If fasteners are used on the PCB, attach the fasteners in a manner that reduces PCB flexing.
- Gentle operation is required when inserting the boards into the housing or enclosure.
- Place the PCB assemblies on a padded shelf or bench top. Do not stack up the PCB assemblies.
- When shipping the devices, assembled PCBs, or completed subassemblies, ensure there is adequate protection from shock during transport.

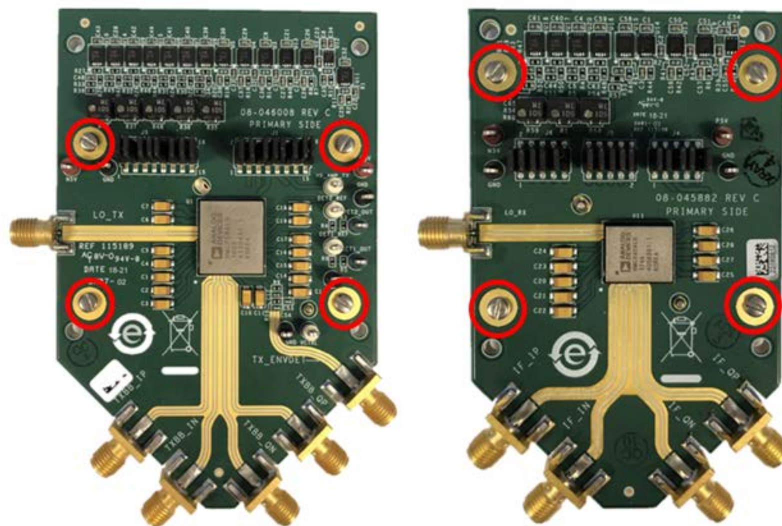


Figure 9. Typical Application Boards with Fasteners

©2019 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners.
AN20294-0-5/19(0)