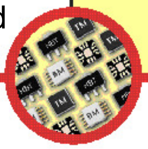
AMCOM COMMUNICATIONS
The RF Power House

BM Packaged MMIC Handling and Mounting

BM Package Assembly on PC Board

Application Note
AN700
Aug 2010
Rev 8



INTRODUCTION

This application note describes the use and handling of AMCOM's MMIC power amplifiers in BM package. The MMIC chips are housed in a 10-pin low cost surface mountable, ceramic package (BM). The packaged MMIC can be handled by standard pick and place equipment. Figure 1 at right is the photograph of the BM package.

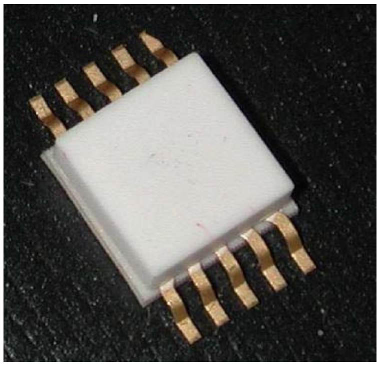


Figure 1: Photo of BM Package

ELECTRICAL SCHEMATIC

Please refer to each MMIC model's data sheet for detailed schematic and pin assignment. DC decoupling and filtering components need to be added externally to ensure stability of the amplifier and to prevent any possible spurious signal due to noisy supplies. Good RF and DC grounding are also required for maintaining stability.

AMPLIFIER BIAS CONSIDERATIONS

1) Gate Bias

As with all GaAs amplifiers, it is important to ensure that the gate bias is present before applying the drain voltage. Without gate control, the drain current will rise to a level that is potentially destructive to the MMIC; therefore, it is recommended to provide safeguards in the circuit design to ensure that the gate bias is applied first. A simple way to achieve this is to leave the gate voltage on all the time. The gate voltage should be adjusted such that the drain current is equal to I_{dq}.

2) Drain Bias

The amplifier is designed to operate within specifications when biased with drain voltage/current as specified in the data sheet. If the MMIC is mounted on a ridged heat sink (see Figure 6A) with good heat sink, it can be biased at a higher Vds (about 1-2V higher). In all cases the absolute maximum rating for voltage, current, or power dissipation must not be exceeded. The DC drain current can be controlled by the negative gate voltage. If the MMIC is biased with a current less than I_{dq} (Class AB) then the gain and P_{1dB} will be reduced.

HANDLING

The amplifier is based on GaAs FET (MESFET or PHEMT) technology; therefore, it is sensitive to electrostatic discharge (ESD). AMCOM ships all power amplifier MMICs in electrostatic protection packages. Users must be careful when handling the amplifier and should follow the standard ESD prevention techniques. A grounded wrist strap will give adequate protection against electrostatic discharge, and workbenches should have antistatic mats.

PACKAGE OUTLINE

AMCOM's "BM" type package is a low cost, leaded surface mountable, ceramic package. Figure 2 shows the package dimension. Refer to the specific data sheet for the exact pin out. The package is designed to be comply with the standard automated SMT reflow soldering process. The package has a metal base pad at the bottom of the package, which serves as DC and RF ground, and as a thermal path to remove the heat generated by the MMIC. The ground base should be soldered to the heat sink. The RF input and output leads should be soldered to the microwave circuit on the PC board.

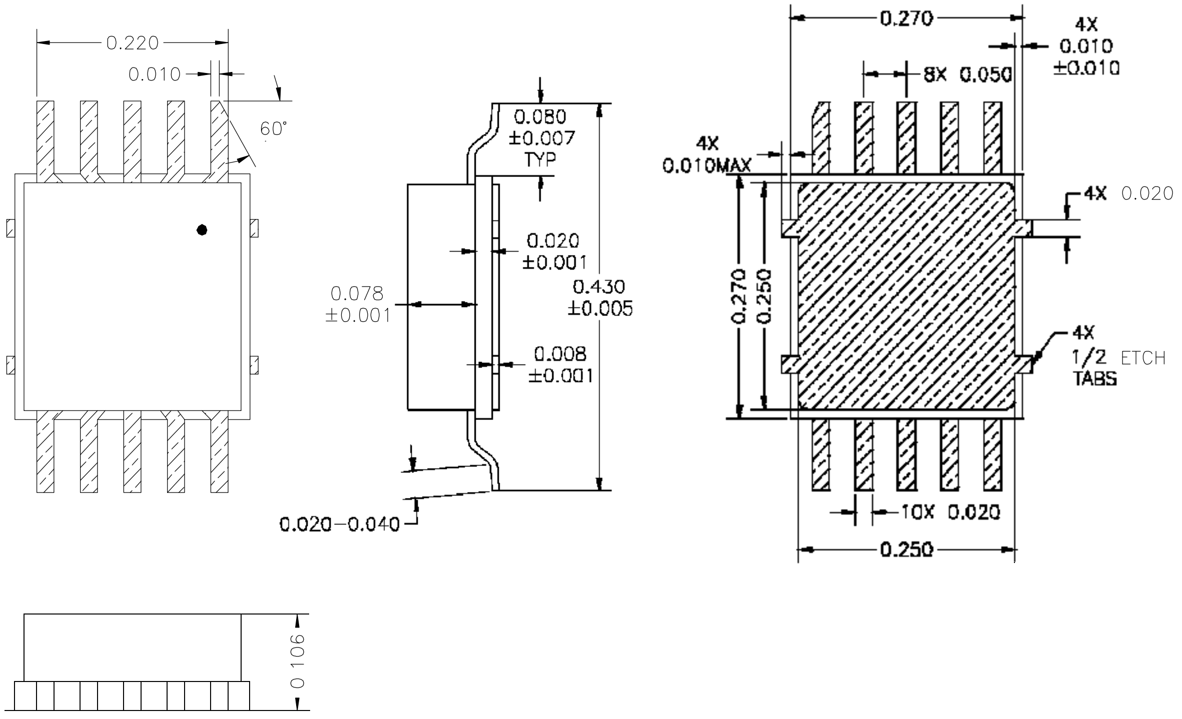


Figure 2: BM Package Outline Drawing

SOLDERING

1) Solder Selection

AMCOM's BM MMICs are RoHS compliant. We recommend R276 NC with a composition of Sn 96.5% / Ag 3% / Cu 0.5%. The melting point is 221℃.

2) Solder Stencil Guidelines

A solder stencil is required to screen solder paste onto the pads of the footprint. The thickness of the solder paste applied will directly affect the quality of the joint. The optimum thickness is 8-10 mils. The solder stencils are typically 8 mils thick and may be made of brass or stainless steel. The stencil opening should be the same size as the pads on the footprint for a 1:1 registration.

3) Suggested Profile for Reflow Soldering

The most common re-flow method used is accomplished in a belt furnace using convection/IR heat transfer. A typical heating profile for RoHS solder is shown in Figure 3. This profile may vary depending on the soldering system used, the density and types of the components on the boards, the type of solder used, and the type of board or substrate used. The temperature shown in the profile is the actual temperature on the board at / or near the central solder joint. It should be noted that the main body of the component may be up to 30℃ cooler than the adjacent solder joints due to the heat absorption.

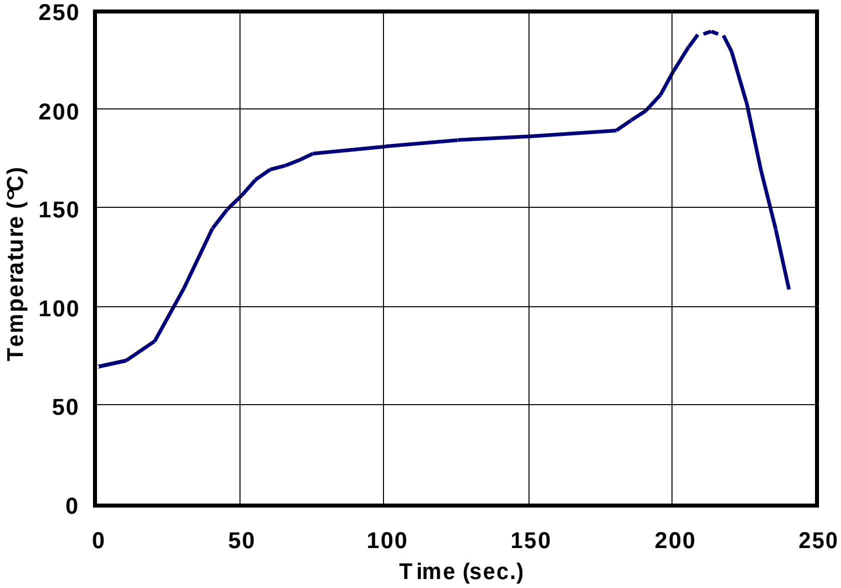


Figure 3: Solder Re-flow Temperature Profile

4) General Precautions

- Always preheat the MMIC amplifier (150℃ for 2 minutes) to minimize the thermal shock and mechanical stress.
- The temperature variation from the preheat stage to the maximum temperature should be less than 100℃.
- Never exceed 230℃ for 20 seconds, 220℃ for 30 seconds or 200℃ for 60 seconds.
- The device should be allowed to cool naturally for at least 3 minutes. Forced cooling may result in failure due to mechanical stress.
- Never apply mechanical stress or shock during cooling.

PC BOARD AND TEST FIXTURE

The packaged MMIC amplifier can be mounted on a number of PC board materials such as FR4 (ε_r = 4.2), Rogers 4003 (ε_r = 3.38), or Rogers R/T Duroid 6010 (ε_r = 10.2). We recommend the Rogers 4003 , Rogers 6010, or similar materials instead of FR4, for frequencies above 3 GHz to reduce circuit loss. For SMT applications the MMIC package is directly soldered to an array of grounded vias providing good RF ground and good thermal transfer. The RF input and output pins are to be soldered to 50 Ohm microstrip transmission line on the PC board. We recommend to use 10 mil thick PCB for good thermal resistance.

Figure 4 shows a test circuit (evaluation board) for a MMIC power amplifier, soldered to the PC board and mounted on a 1.6 x 1.1 x .4 inch Ni–plated aluminum block. Figure 5 is a photograph of the test circuit. SMA connectors are used for input and output ports. Rogers 4350 material with a dielectric constant $\varepsilon_r = 3.48$ and thickness of 10 mils was used for the PCB. Detailed AutoCad files of the PCB and test block can be found on our website at www.amcomusa.com.

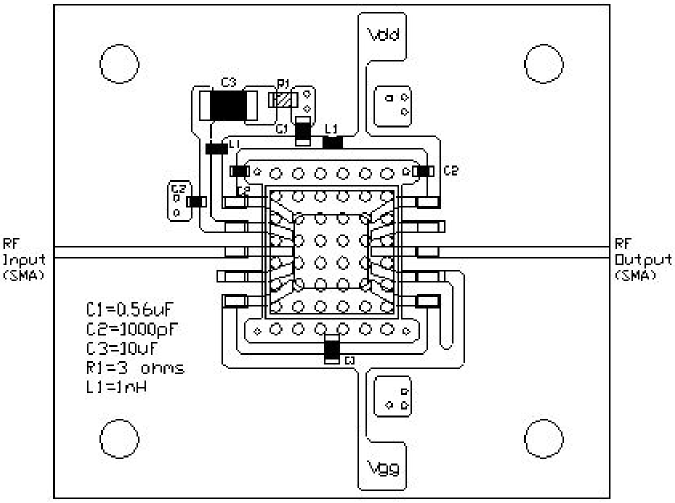


Figure 4: MMIC Mounted on PCB

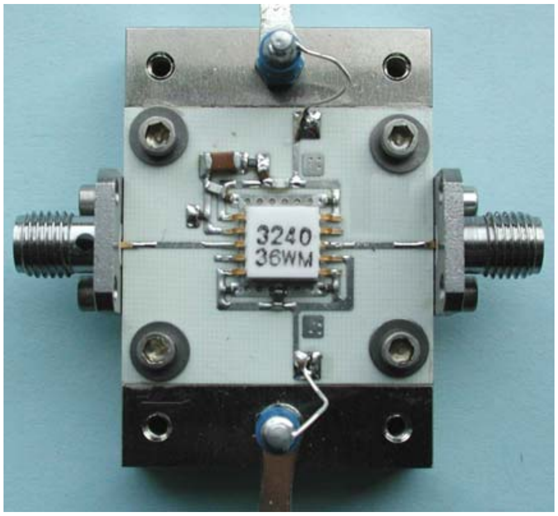


Figure 5: Photo of test board

(Photos are for reference only and may vary from MMIC to MMIC)

MMIC PA THERMAL CONSIDERATIONS

1) Mounting Method

A MMIC power amplifier dissipates several watts of power. It is important to provide a good heat sink to dissipate the heat. There are two options of mounting the amplifier. The most effective way is to mount the amplifier to a metal ridge heat sink as shown in figure 6A. We strongly recommend this way for high power MMIC such as AM142540MM-BM-R, which has 10W output power. The other option, which is more practical and less expensive, is to add sufficient number of plated through via holes to the PCB. The base of the device is soldered to the PCB as shown in figure 6B. The via hole wall should be plated by at least 1 oz thick (1.5 mil) of high thermal conductivity copper to conduct the heat from the top of PCB to the bottom of PCB. If you must use thicker than 10-mil PCB, follow the calculation in the next section to determine the PCB thermal resistance.

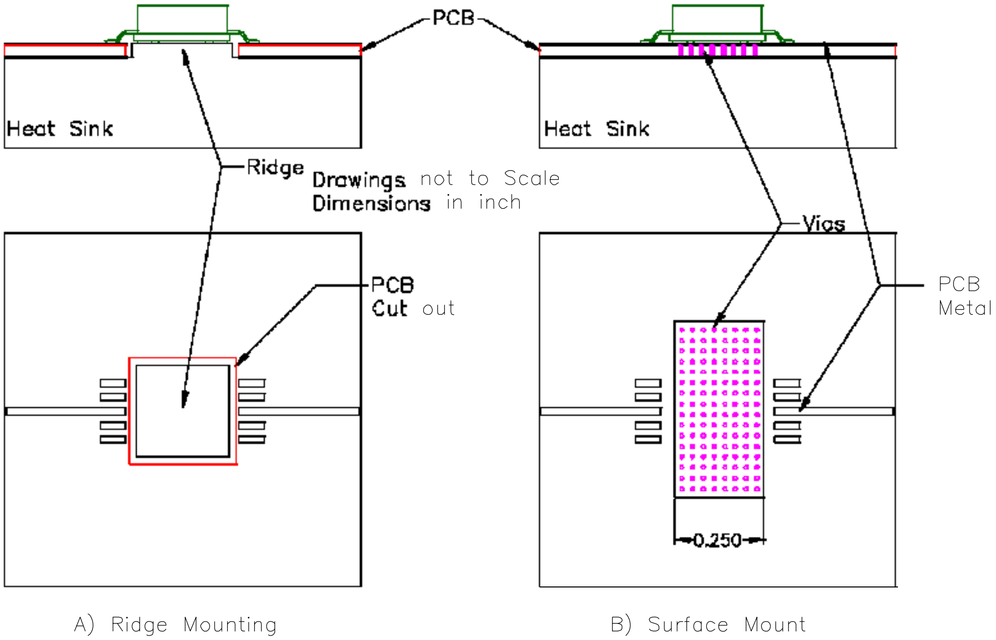


Figure 6: Package Mounting

2) PC Board Thermal Resistance

When BM package is surface mounted on the PC board, we recommend the PC board should have via holes under the BM bottom metal as follows:

1) Via hole diameter is 0.6mm (24mils).

2) Via hole center-to-center distance is 1.14mm (45.6mils)

3) The vertical wall of via hole is plated with a minimum thickness of 38µm (0.038mm, 1.5mils, or 1-Oz) of high thermal conductivity copper.

4) The bottom of the PC board is attached to a heat sink (See next section).

5) Fill via holes with solder.

There are 3 thermal paths in the PC board: (a). Through plated copper wall. b). Through the filled solder. (c). Through PC board epoxy fiberglass material. Because the thermal conductivity of epoxy fiberglass is much smaller than copper or solder, we will neglect the thermal conduction through epoxy fiberglass.

Figure 7 shows the top view of a via hole:

Via hole diameter, D_1 = 0.6mm.
Via hole opening after copper plating, D_2 = 0.524mm.
Cu thickness = $D_1 - D_2$ = 38µm = 0.038mm.
Via Center-to-center = 1.14mm

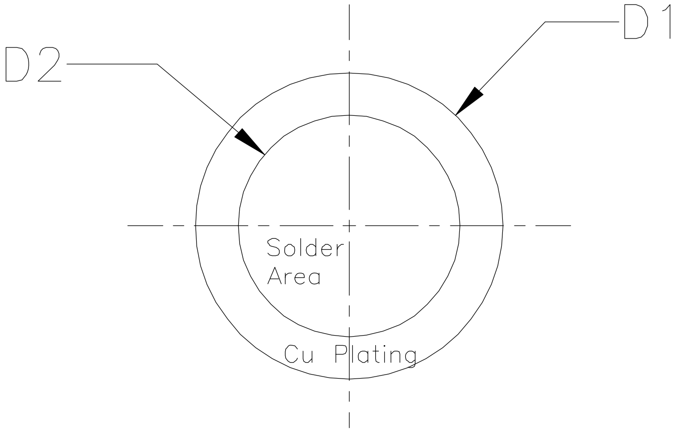


Figure 7: Top View of a Via Hole

Thermal conductivity of each via hole is given by:

Thermal conductivity (via) = $K_{cu}*A_{cu}/L + K_{sold}*A_{sold}/L$ …….Equation 1.

Where:

K_{cu} is thermal conductivity of copper = 0.397 W/(mm-°C)
A_{cu} is area of copper
K_{sold} is thermal conductivity of solder = 0.050 W/(mm-°C)
A_{sold} is area of solder
L is thickness of PC board in mm.

Now, D_1 = 0.6mm. Thickness of plated copper = 0.038mm.

$D_2 = 0.6-2*0.038 = 0.524$mm
$A_{cu} = (3.1416/4)*(0.6*0.6 – 0.524*0.524) = (3.1416/4) (0.36 – 0.275) = 0.067$ mm^2
$A_{sold} = (3.1416/4)*(0.524*0.524) = 0.216$ mm^2
$K_{cu}*A_{cu} = 0.397*0.067 = 0.0266$
$K_{sold}*A_{sold} = 0.050*0.216 = 0.011$
$K_{cu}*A_{cu} + K_{sold}*A_{sold} = 0.0266 + 0.011 = 0.0376$

The via hole thermal resistance is:

R_{th} (via hole): $L/(K_{cu}*A_{cu} + K_{sold}*A_{sold}) = L / 0.0376$…….. Equation 2 (L in mm)

For a PC board thickness of 0.25mm (10mils), the thermal resistance per via hole is:

R_{th} (via) = 0.25/0.0376 = 6.65 °C/W per via hole

The dimension of the BM package base metal is 6.25x6.25mm^2 (0.25"x0.25"). Using center-to-center of 1.14.mm, there are about 36 via hole under the metal base. The PC board thermal resistance is:

R_{th} (PCB) = 6.65/36 = 0.18 °C/W

3) Interface thermal resistance between PCB and metal heat sink

When the PCB is mounted on the metal heat sink, there is an interface thermal resistance depending on how good the thermal contact is. Both the metal heat sink surface and the PCB bottom surface should be flat. The contact between the metal heat sink and PCB should not have any air pocket. We recommend the following to minimize the interface thermal resistance:

1- The ground metal heat sink should be Sn or Ag plated

2- There should be at least four screws near the active device to hold down the PCB to the ground metal heat sink.

3- To achieve good electrical and thermal contacts between the PCB and the ground metal heat sink, we recommend to put a sheet of silver epoxy between the PCB and ground. We recommend Ablefilm 5025E by Emerson and Cuming. This silver epoxy needs to be cured at 150℃ for 30 minutes. There should be a pressure of about 60lb applied during curing (Either by a clump or by weight).

MOUNTING RECOMMENDATIONS

1) Heat sink

- Mounting area of the heat sink should be clean and free of oxidation.
- The dimensions of the heat sink should be adequate.
- Mounting area roughness should be less than 3 µm.
- Mounting area flatness should be less than 30 µm.

2) PC Board

- Tin and clean the PCB.
- Add solder perform or solder paste.

3) Mounting Sequence

Add solder preform or solder paste on the pads of the PCB. Add the required components for filtering and stability (capacitors and resistors). Preheat the MMIC amplifier (150℃ for 2 minutes) to minimize the thermal shock. Position the device on the PCB with the right orientation (the dot indicates pin 1). Use hot plate or any of the common reflow methods to solder the components to the PCB.