

## INTRODUCTION

This application note describes the use and handling of AMCOM's BH and BI series of High Voltage FET (HiFET) devices. The high power, high voltage GaAs MESFET devices are packaged in a low cost drop-in style ceramic package with a solder-mountable base. The option of a surface mount package with bent leads is also available if required (please contact AMCOM for more information).

## BASIC OPERATIONS / DC BIASING

### Drain Bias

These devices are designed to operate within specifications when biased with drain voltage/current as specified in the datasheet. The HiFET devices should be mounted on a ridged heat sink (see Figure 3A). In all cases the absolute maximum rating for voltage, current, or power dissipation must not be exceeded. The negative gate voltage controls the DC drain current. If the HiFET is biased with a current and/or voltage less than  $I_{dq}$  then the gain and P1dB will be reduced.

### Gate Bias

As with all GaAs MESFET 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 device; therefore, it is recommended to provide safeguards in the circuit design to ensure that the gate bias is applied first. The gate voltage should be adjusted such that the drain current is equal to  $I_{dq}$ . One very important characteristic of the HiFET devices is that the negative gate bias will draw more current than a typical GaAs MESFET device (as is specified on the datasheet). This is normal for the operation of the device due to the internal resistances in the High Voltage configuration. An appropriate negative supply therefore should be selected to bias the gate.

### Stability

GaAs FET devices require carefully designed stable configurations when used in amplifier applications. Stability can be predicted using S-parameters and their derived K factor results for a given input and output matching network. In addition, the use of elements such as RC bypass circuits on the gate and drain bias lines improve stability at lower frequencies and are required when implementing the HiFET devices in an amplifier circuit.

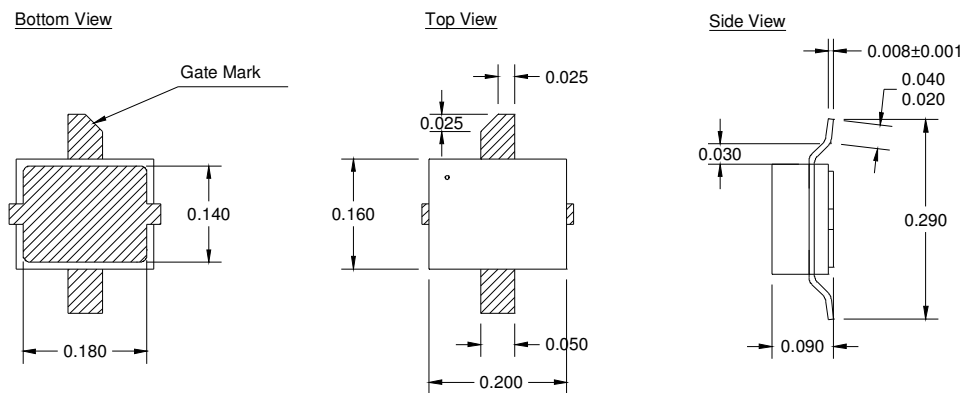
**HANDLING**

The HiFET is based on GaAs technology; therefore, it is sensitive to electrostatic discharge (ESD). AMCOM ships all power HiFETs 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 charge, and workbenches should have antistatic mats.

**PACKAGE TYPE "BH" and "BI" MOUNTING**

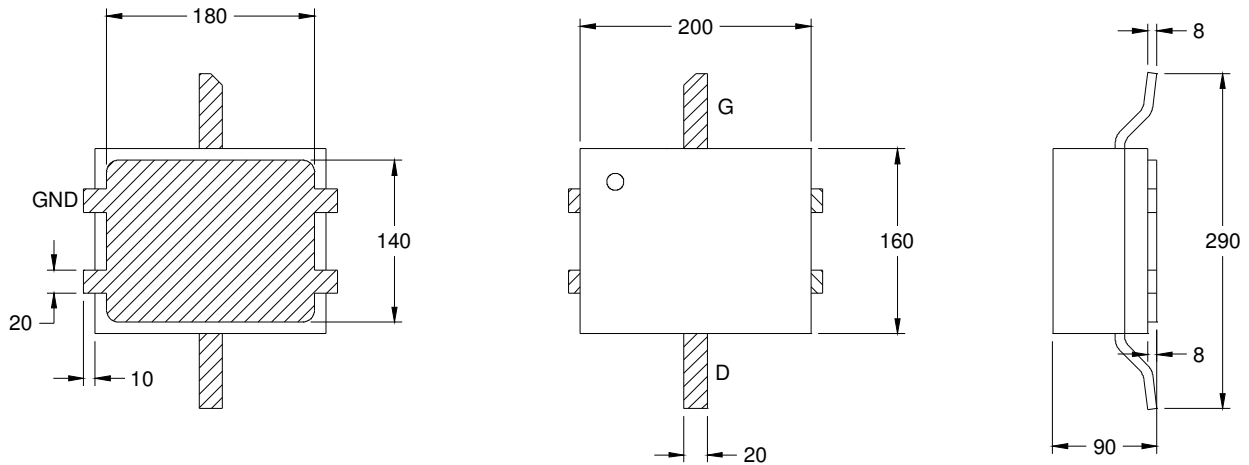
AMCOM's "BI" (Figure 1) and "BH" (Figure 2) type package is a low cost, straight-leaded, ceramic package with a ceramic lid. The package is designed to be mounted using standard reflow soldering processes. The package has a base pad at the bottom of the package. This base should be directly soldered to a heatsink for better performance. The RF input and output leads should be soldered to the microwave circuit on the PC board. An SMT version of this package with bent leads is available. When using an SMT package, the base should be soldered to a well-grounded pad with adequate thermal vias to a heatsink layer.

**PACKAGE OUTLINE (Figure 1)**



\* All Dimensions are in inches

**PACKAGE OUTLINE (Figure 2)**



\*All Dimensions are in inches

### SOLDER SELECTION

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

### 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.

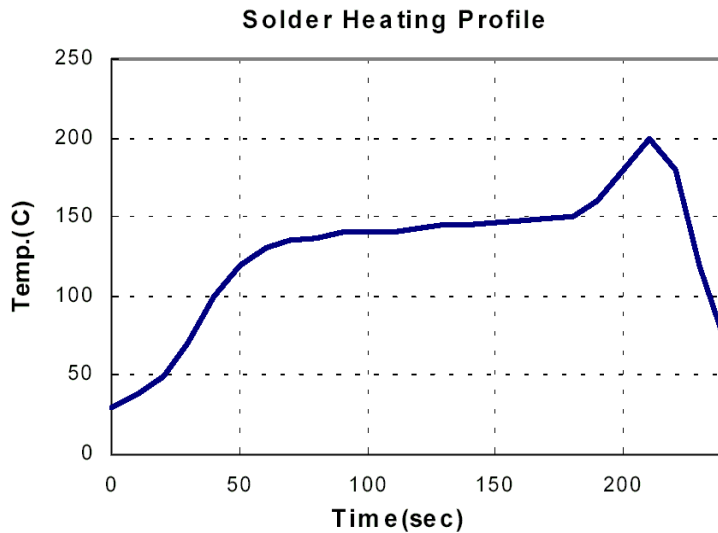


Figure 3

---

## RECOMMENDED PROFILE FOR REFLOW SOLDERING

The most common reflow method used is accomplished in a belt furnace using convection/IR heat transfer. A recommended heating profile that shows the temperature at package/circuit board interface versus time is shown in Figure 2. 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°C cooler than the adjacent solder joints due to the heat absorption.

## GENERAL PRECAUTIONS

- Always preheat the HiFET (150°C 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°C.
- Never exceed 230°C for more than 20 seconds, 220°C for 30 seconds, and 200°C 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

Circuits incorporating the HiFET can be implemented on a number of PC board materials such as FR4 ( $\epsilon_r = 4.2$ ), Rogers 4003 ( $\epsilon_r = 3.38$ ), or Rogers R/T Duroid 6010 ( $\epsilon_r = 10.2$ ). We recommend the Rogers 4003 or Rogers 6010, instead of FR4, for frequencies above 3 GHz to reduce circuit loss. Figure 3A shows the typical mounting arrangement for the HiFET. In this case, the package base is directly soldered to a metal heatsink providing good RF ground and good thermal transfer. The RF input and output pins are soldered to the PC board.

## Thermal Considerations

The HiFET 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 device. The most effective way is to mount the amplifier to a metal ridge that is mounted directly to a heat sink (as described above and shown in Figure 3A). If an SMT package is used, this option requires adding a sufficient number of plated-through, filled, via holes to the PCB and soldering the base of the device to the PCB as shown in Figure 3B. The edge of the plated through via holes 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. Please note that the added thermal resistance and via inductance of this configuration will reduce the HiFET's power and gain performance.

## Important factors for thermal resistance of the PCB in SMT mounted applications

- Thickness of the PCB material
- Thickness of the metalization of the PCB
- Number of plated through vias
- Diameter of plated through vias
- Via-hole plating thickness and the amount of solder-fill in the plated-through vias.

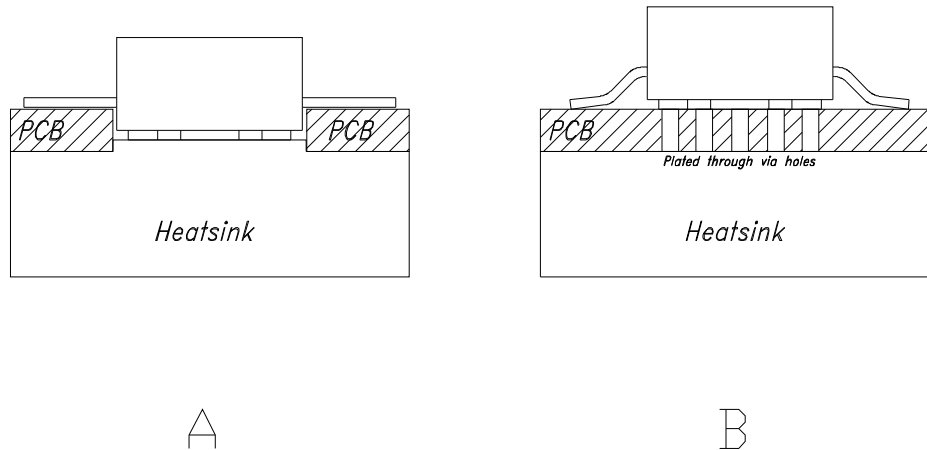


Figure 4

## 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  $\mu\text{m}$ .
- Mounting area flatness should be less than 30  $\mu\text{m}$ .

### 2) PC Board

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

## Mounting Sequence

Add solder pre-form or solder paste on the pads of the PCB and on the heatsink below the device. Add the required components for filtering and stability (capacitors and resistors). Preheat the HiFET (150°C for 2 minutes) to minimize the thermal shock. Position the device on the heatsink and PCB with the right orientation (the dot indicates the Gate). Use a hot plate or any of the common re-flow methods to solder the components to the PCB.

### 2.1 GHz Test Circuit using the AM030MH4-BI-R HiFET

Figure 6 shows a 2.1 GHz circuit design utilizing the AM030MH4-B-RI. The PCB board used is 32 mil FR4 material with the HiFET device solder-mounted directly to a Tin plated copper heatsink. The PCB Board is epoxied to the heatsink block using a silver epoxy preform. SMA connectors were used for input and output ports. Detailed CAD files of the PCB board and test block can be found by contacting AMCOM at [info@amcomusa.com](mailto:info@amcomusa.com). The measured performance of the 2.1 GHz circuit is shown in Figure 4 (Small Signal Data) and Figure 5 (Power Data).

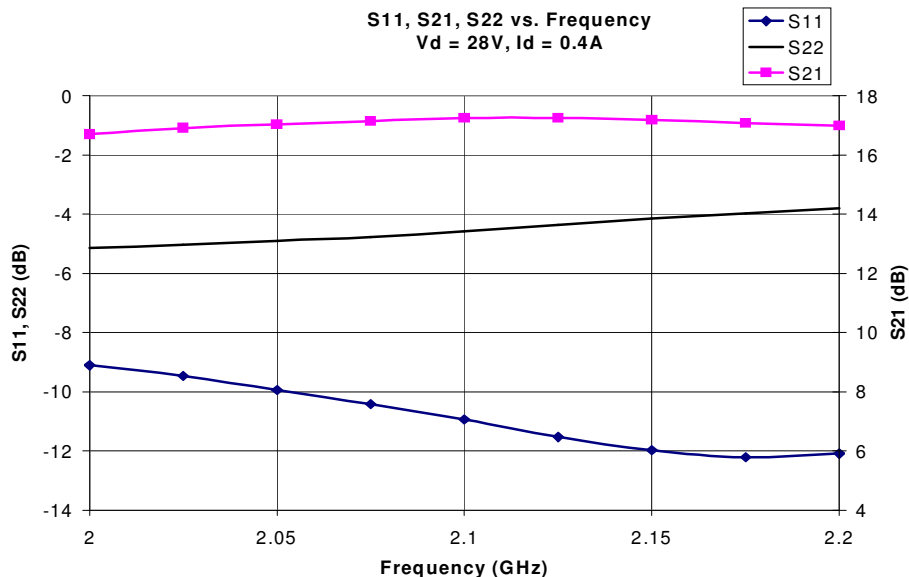
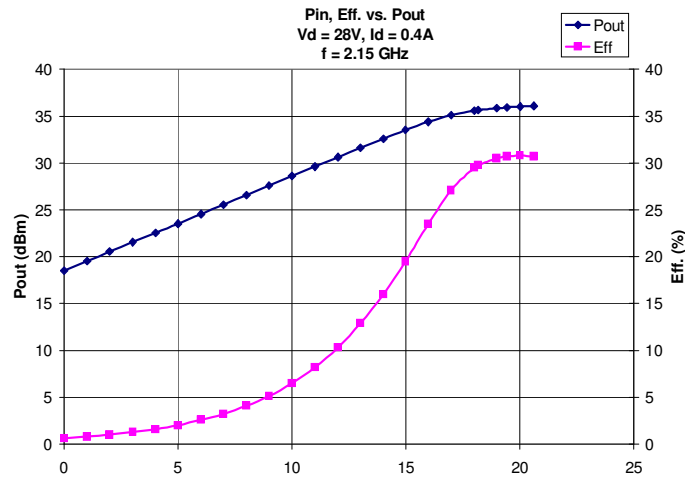


Figure 5. Small Signal Data for 2.1GHz Circuit (AM030MH4-BI-R)



P1dB, Efficiency vs. Frequency  
 Vd = 28V, Id = 0.4A

Frequency (GHz)	P1dB (dBm)	Efficiency (%)
2	35.97	31.3
2.05	35.89	31.1
2.1	35.81	30.7
2.15	35.66	29.8
2.2	35.51	28.9

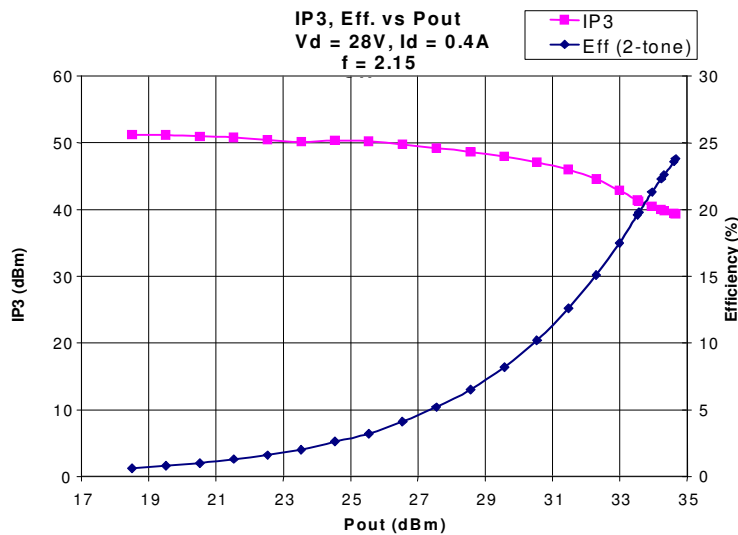
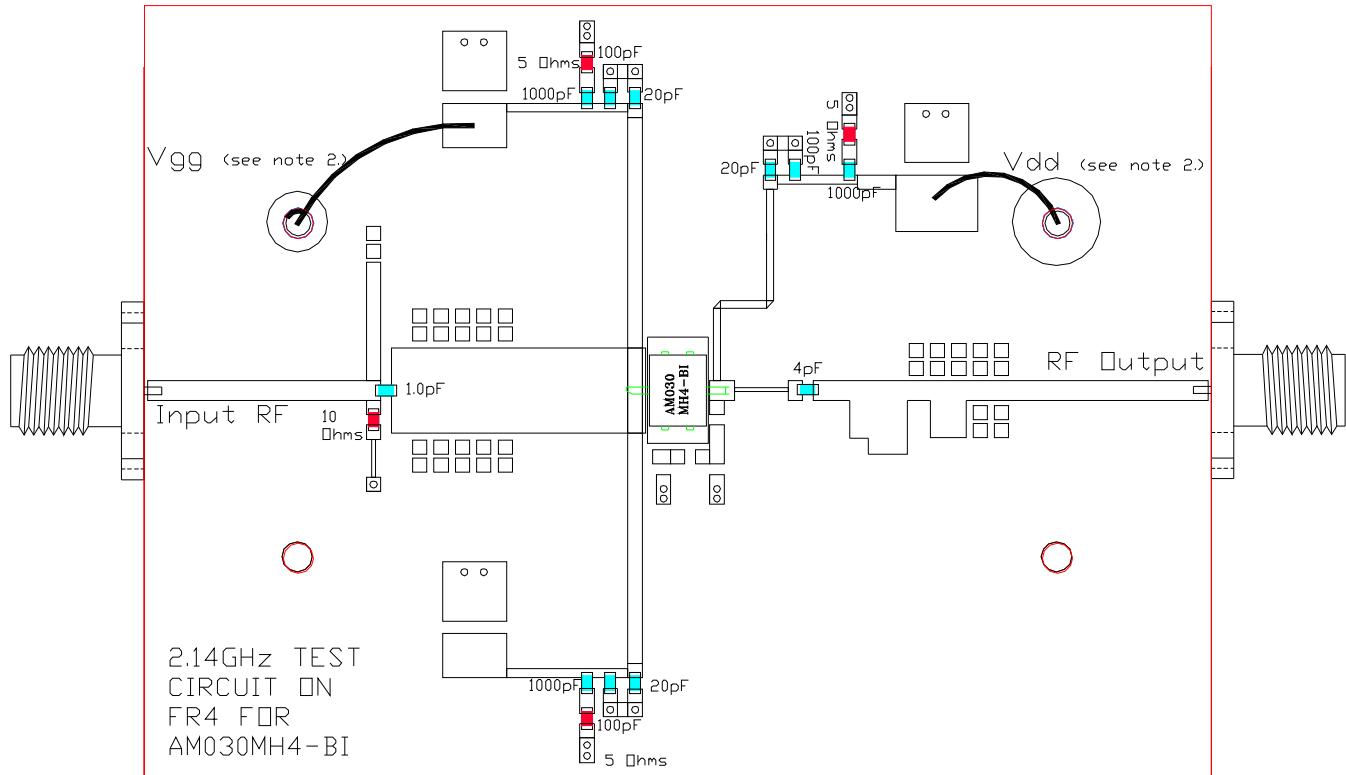


Figure 6. Power, Efficiency, and IP3 Data for 2.1 GHz Circuit (AM030MH4-BI-R)



Notes:

1. FR4 Material, 32 mil thickness
2. Vdd & Vgg should have 1uF bypass capacitors to GND
3. All dims. are in inches. All SMT components are 0603 size.
4. Package is solder mounted to heatsink directly

Figure 7. Board Layout of the 2.1GHz Circuit (AM030MH4-BI-R)