Handling and Mounting of Packaged FETs
Application Note # AN300

V.2, December 2000

HANDLING

AMCOM's Plastic Packaged Power FETs are very sensitive to electrostatic discharge (ESD). AMCOM ships all Power FETs in electrostatic protection packaging. Users must be very careful when handling the FETs and should be properly grounded by a wrist strap or equivalent technique.

AMCOM's Plastic Packaged Power FETs have four pre-tinned copper leads and a pre-tinned base. Two of the copper leads are RF input and output leads. The other two are the grounding leads. The base of the package serves simultaneously as DC ground, RF ground, and thermal path. Personnel handling the FETs should be very careful to avoid the destruction of the coplanarity between the leads and the base. The FETs are shipped in a tray as shown in Figure 1. **Be sure to pay careful attention to the handling precautions during removal.**

- Quantity per tray: 40 pieces
- Each tray will be sealed with anti-ESD packing
- FETs can be removed from the packing with tweezers or vacuum pen.

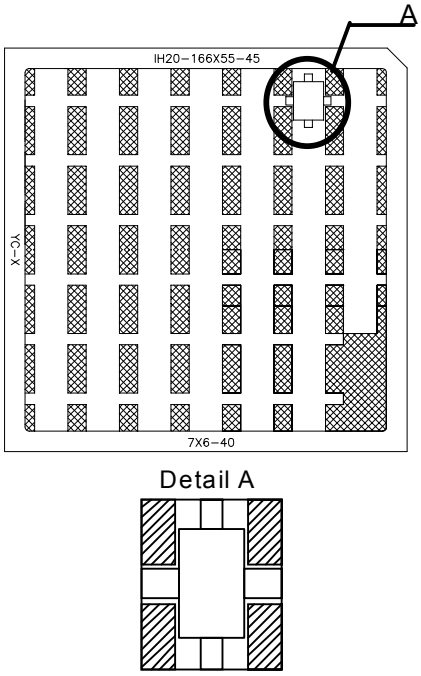

Figure 1

PACKAGE TYPE "TF" MOUNTING

AMCOM's "TF" type package (Figure 2) is specially designed for drop-in placement. The package has a source pad (base) at the bottom of the package and two source leads. Both the source leads and source pad should be soldered to the heat sink for better performance. The RF input and output leads should be soldered to the microwave circuit on the PC board.

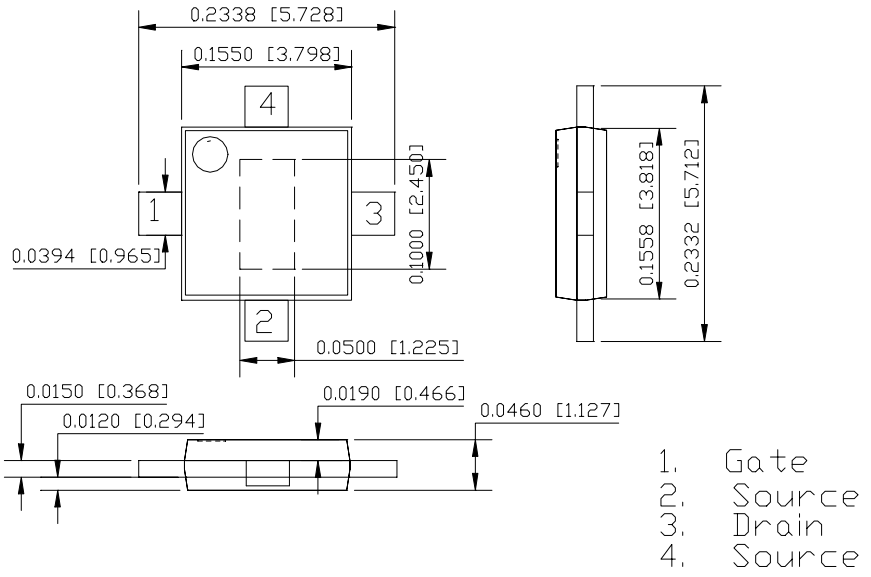

Figure 2. AMCOM FET in TF package (Unit: inches [mm])

e-mail: products@amcomusa.com
www.amcomusa.com

Tel. (301) 353-8400 Fax (301) 353-8401
P.O. Box 179, 22300 Comsat Drive, Clarksburg, MD 20871 USA

SOLDER SELECTION

The recommended solder is Sn63 (63% Sn, 37% Pb). Sn63 is an eutectic compound with a melting point of +183°C. This is high enough to exceed the standard operating temperature limit of most components (~+150°C) and low enough not to damage the internal component of the package during solder reflow.

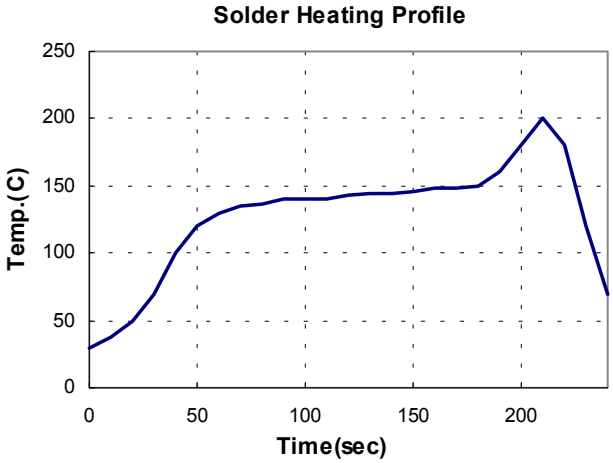

Figure 3. Solder Heating Profile

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.

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 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°C cooler than the adjacent solder joints due to the heat absorption.

GENERAL PRECAUTIONS

- Always preheat the FET (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 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

The amplifier matching circuit to the TF series FETs can be fabricated on a number of PC board material such as FR4 ($\in$r = 4.2), Rogers 4003 ($\in$r = 3.38), or Rogers R/T Duroid 6010 ($\in$r = 10.2). For frequency above 3 GHz, we recommend the Rogers 4003 or 6010 for lower circuit loss than FR4.

Figure 4 shows the recommended test fixture arrangement for the TF series FETs. In this case, a PC board with a cutout is placed on a flat metal heat sink allowing the package to be dropped into the cutout region. The package base is directly soldered to the metal ground providing good RF ground and good thermal transfer. Furthermore, the two grounding 'gull wings' should also be soldered to the metal base to provide two additional thermal paths. This arrangement facilitates efficient thermal transfer to reduce the FET junction temperature. The RF input and output leads are to be soldered on to the microwave circuits on the PC board. The PC Board has a substrate thickness of 10 mils.

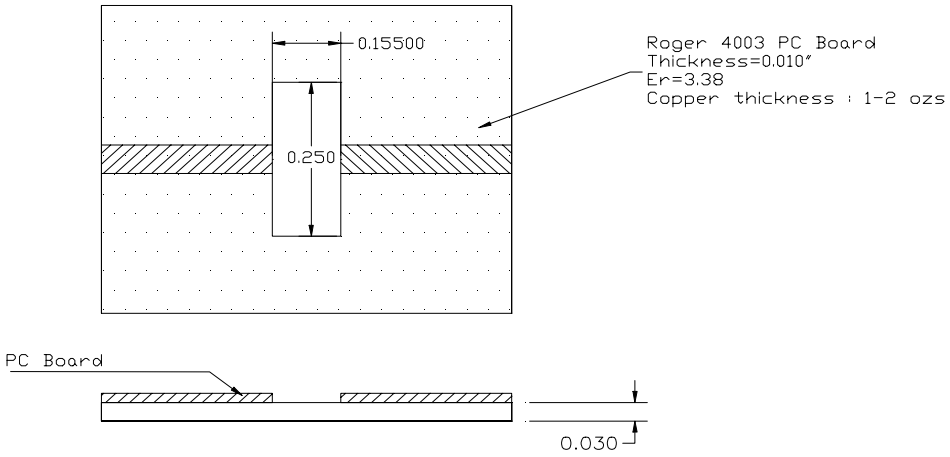

Figure 4. PC Board for testing AMCOM TF - series FETs (Unit: inch)

PC BOARD AND TEST FIXTURE - cont.

If a test fixture configuration with a different PC Board substrate thickness is to be used, it is required that the metal ground have a raised ledge as shown in Figure 5. It is important that the elevated distance between the top of the metal ledge and the top of the PC Board is not greater than 15 mils. The data shown in the data sheets are obtained using this test fixture.

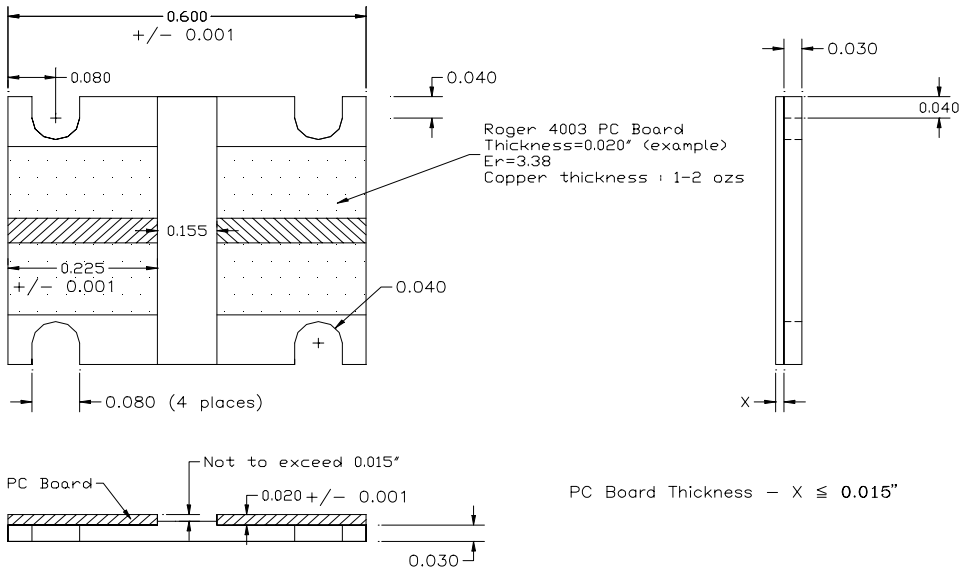

Figure 5. Carrier for testing AMCOM TF - series FETs (Unit: inch)