RF3220

HIGH LINEARITY/DRIVER AMPLIFIER

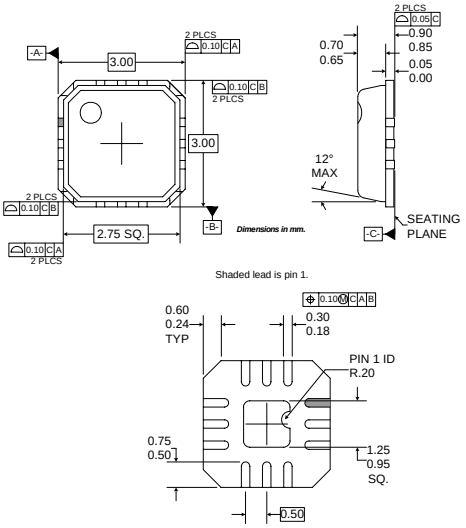
RoHS Compliant & Pb-Free Product

Typical Applications

- Basestation Applications
- Cellular and PCS Systems
- CDMA, W-CDMA Systems
- GSM/EDGE Systems
- Final PA for Low-Power Applications

Product Description

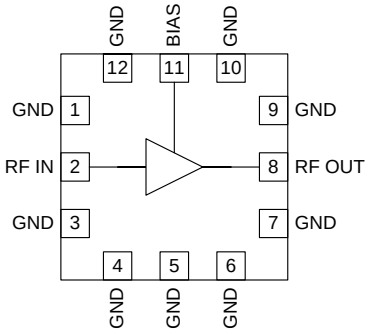
The RF3220 is a high-efficiency GaAs Heterojunction Bipolar Transistor (HBT) amplifier packaged in a low-cost surface-mount package. This amplifier is ideal for use in applications requiring high-linearity and low noise figure over the 500MHz to 3GHz frequency range. The RF3220 operates from a single 5V power supply, and is assembled in an economical 3mmx3mm QFN package.



Package Style: QFN, 12-Pin, 3x3

Optimum Technology Matching® Applied

- ☐ Si BJT
- ☑ GaAs HBT
- ☐ GaAs MESFET
- ☐ Si Bi-CMOS
- ☐ SiGe HBT
- ☐ Si CMOS
- ☐ InGaP/HBT
- ☐ GaN HEMT
- ☐ SiGe Bi-CMOS



Functional Block Diagram

Features

- 500MHz to 2GHz
- +40.8dBm Output IP3
- +14.2dB Gain at 1850MHz
- +12.4dBm Input P1dB at 1850MHz
- 2.8dB Noise Figure at 1850MHz
- Single 5V Power Supply

Ordering Information

RF3220	High Linearity/Driver Amplifier
RF3220PCBA-41X	Fully Assembled Evaluation Board

RF Micro Devices, Inc.
7628 Thorndike Road
Greensboro, NC 27409, USA
Tel (336) 664 1233
Fax (336) 664 0454
http://www.rfmd.com

Absolute Maximum Ratings

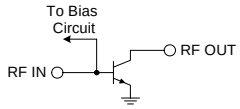
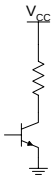
Parameter	Rating	Unit
RF Input Power	+20	dBm
Device Voltage	-0.5 to +6.0	V
Device Current	250	mA
Operating Temperature	-40 to +85	°C
Storage Temperature	-40 to +150	°C

Caution! ESD sensitive device.

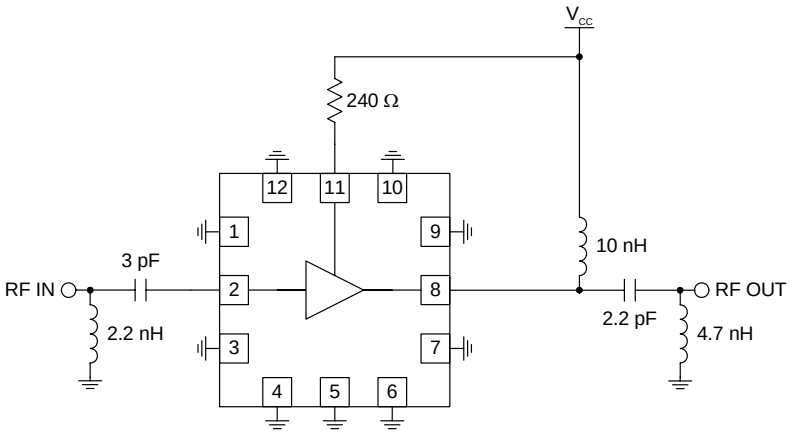
RF Micro Devices believes the furnished information is correct and accurate at the time of this printing. RoHS marking based on EU Directive 2002/95/EC (at time of this printing). However, RF Micro Devices reserves the right to make changes to its products without notice. RF Micro Devices does not assume responsibility for the use of the described product(s).

Parameter	Specification			Unit	Condition
	Min.	Typ.	Max.		
Overall					V_{CC}=5V, RF_{IN}=-10dBm, Freq=1850MHz, with Temp=25°C unless otherwise noted.
AC Specifications					
Frequency	500		2000	MHz	
Gain	12	14.2	15.5	dB	
Input VSWR		1.2	1.5	SWR	
Output VSWR		1.7	2.0	SWR	
Reverse Isolation	20	23			
Output IP3	36	40		dBm	F_1= 1850MHz, F_2=1851MHz
Output P1dB	23	+25.5		dBm	
Noise Figure		2.9	3.3	dB	
Thermal					I_{CC}=160mA, P_{DISS}=0.997W. (See Note.)
$Theta_{JC}$		76		°C/W	
Maximum Measured Junction Temperature at DC Bias Conditions		153		°C	T_{CASE}=+85°C
Mean Time To Failures		5800		years	T_{CASE}=+85°C
DC Specifications					
Device Voltage	4.5	5.0	5.5	V	I_{CC}=160mA
Operating Current Range	110	145	170	mA	V_{CC}=5V

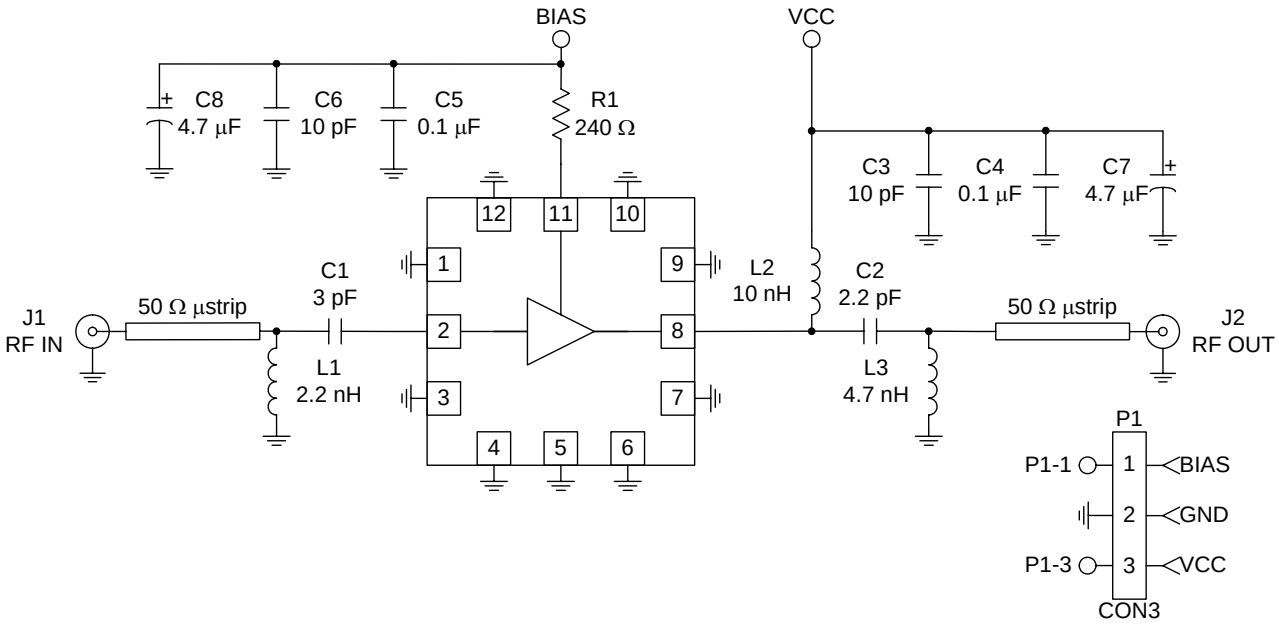
Note: The RF3220 must be operated at or below 175mA in order to achieve the thermal performance listed above. While the RF3220 may be operated at higher bias currents, 175mA is the recommended bias to ensure the highest possible reliability and electrical performance.

Pin	Function	Description	Interface Schematic
1	GND	Ground connection.	
2	RF IN	RF input pin. This pin is not internally DC-blocked. A DC blocking capacitor suitable for the frequency of operation should be used.	
3	GND	Ground connection.	
4	GND	Ground connection.	
5	GND	Ground connection.	
6	GND	Ground connection.	
7	GND	Ground connection.	
8	RF OUT	Amplifier output pin. This pin is an open-collector output. It must be biased to V_{CC} through a choke or matching inductor. This pin is typically matched to 50Ω with a shunt bias/matching inductor and series blocking/matching capacitor. Refer to application schematics.	See pin 2.
9	GND	Ground connection.	
10	GND	Ground connection.	
11	BIAS	This pin is used to control the bias current. An external resistor may be used to set the bias current for any V_{PD} voltage. Allows for trade-offs between IP3 versus noise figure and T_{MAX}.	
12	GND	Ground connection.	
Pkg Base	GND	Ground connection. Vias to ground required under the package base.	

Application Schematic - 1850MHz

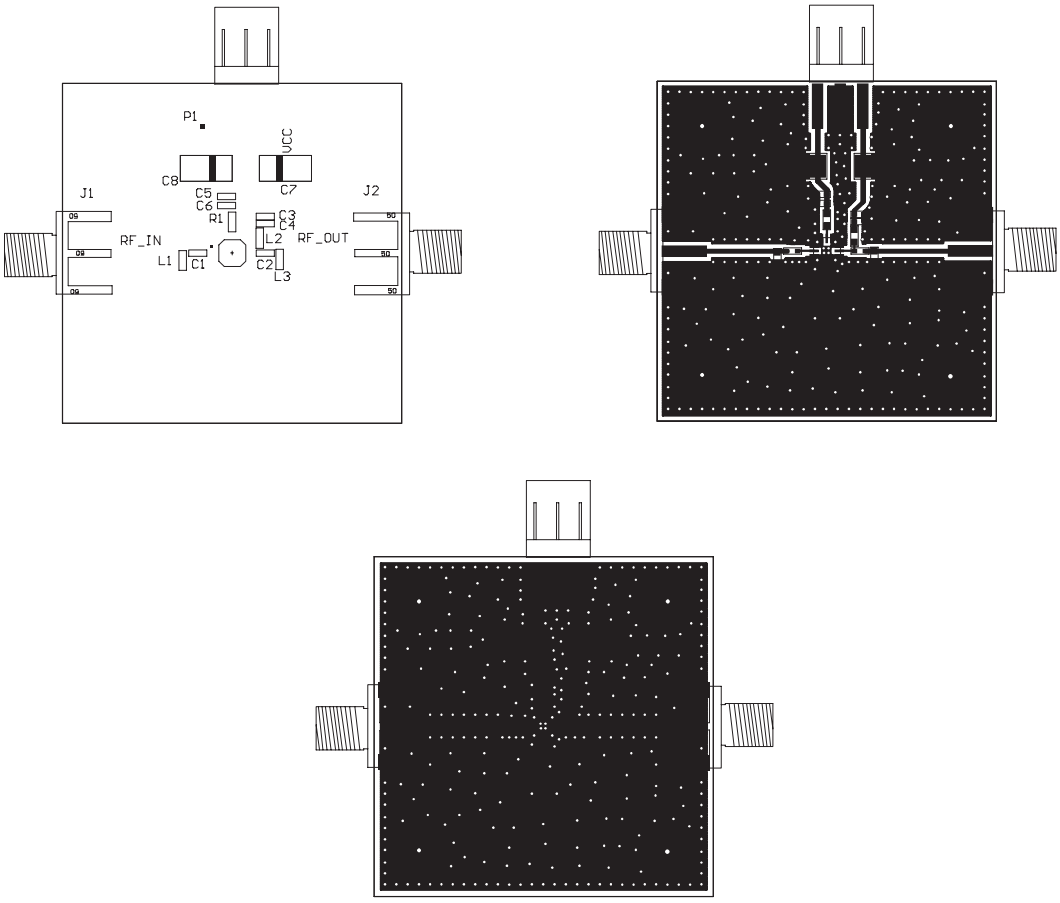


Evaluation Board Schematic

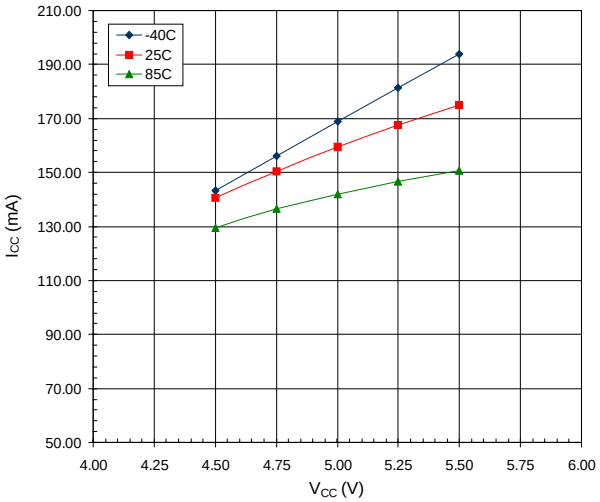


Evaluation Board Layout
Board Size 1.5" x 1.5"

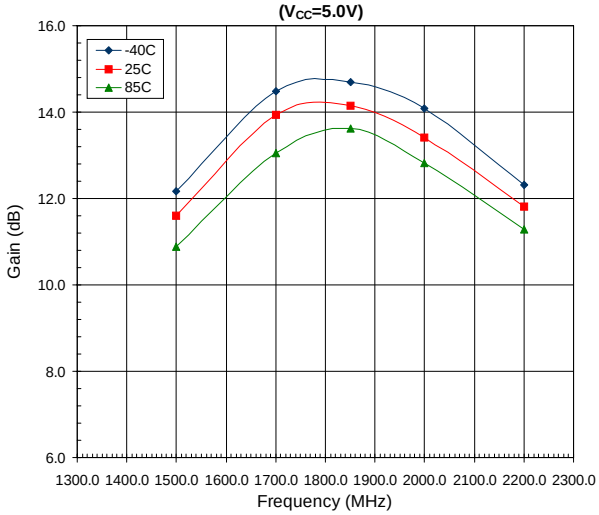
Board Thickness 0.032”, Board Material FR-4



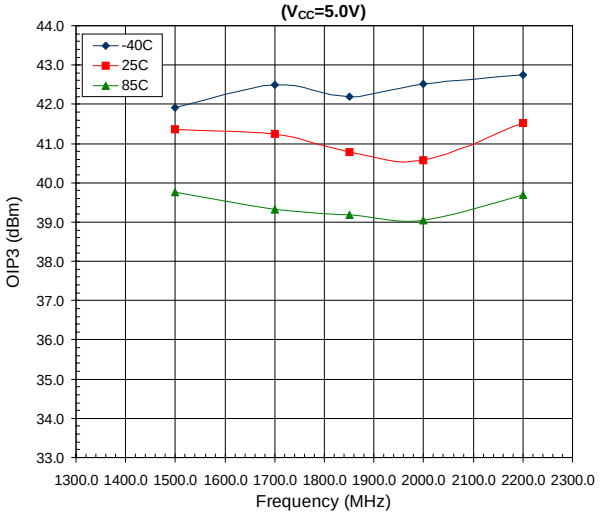
I_{CC} versus V_{CC} Across Temperature



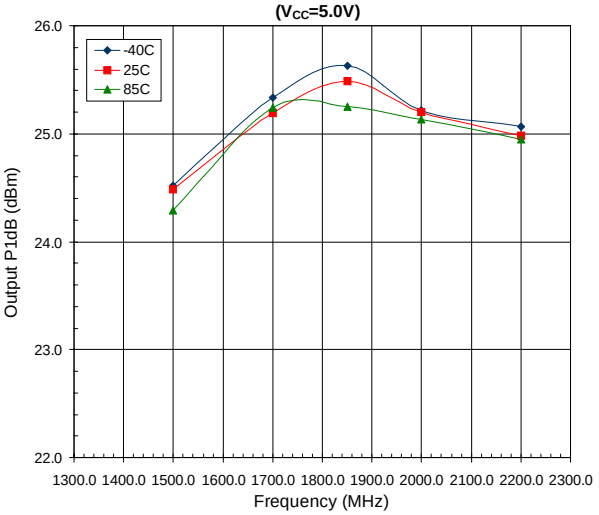
Gain versus Frequency Across Temperature
(V_{CC}=5.0V)



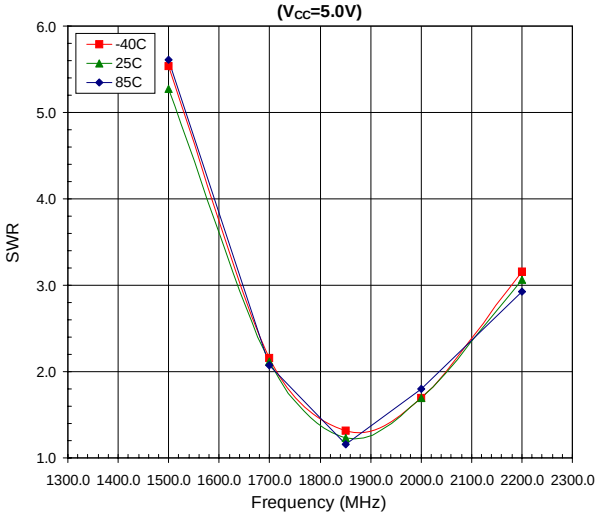
OIP3 versus Frequency Across Temperature
(V_{CC}=5.0V)



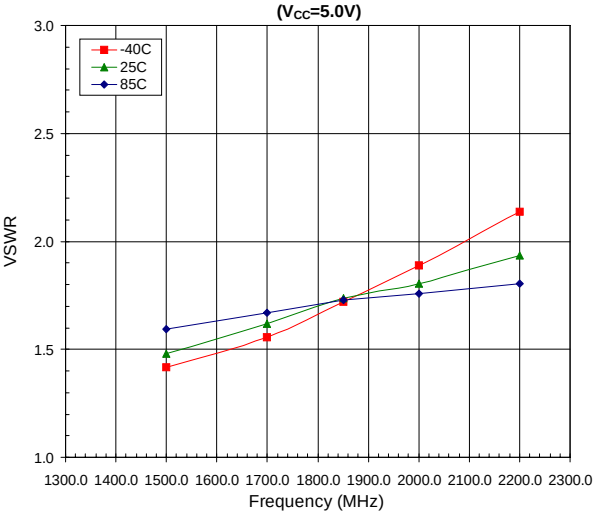
Output P1dB versus Frequency Across Temperature
(V_{CC}=5.0V)

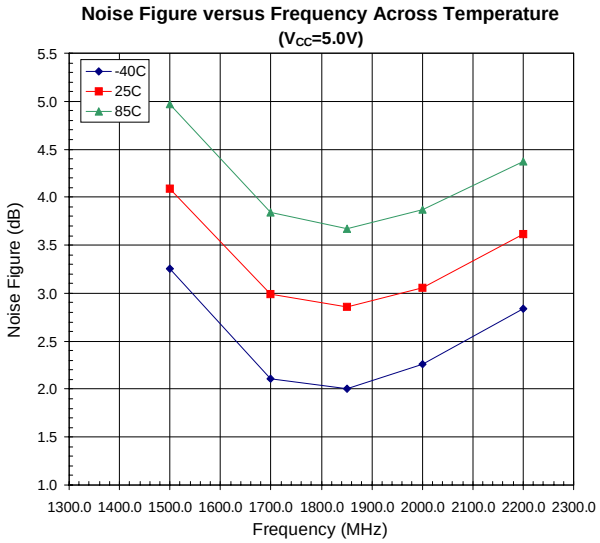


Input SWR versus Frequency Across Temperature
(V_{CC}=5.0V)



Output VSWR versus Frequency Across Temperature
(V_{CC}=5.0V)




Noise Figure versus Frequency Across Temperature
(VCC=5.0V)
-40C
25C
85C
Noise Figure (dB)
5.5
5.0
4.5
4.0
3.5
3.0
2.5
2.0
1.5
1.0
1300.0 1400.0 1500.0 1600.0 1700.0 1800.0 1900.0 2000.0 2100.0 2200.0 2300.0
Frequency (MHz)

PCB Design Requirements

PCB Surface Finish

The PCB surface finish used for RFMD's qualification process is electroless nickel, immersion gold. Typical thickness is 3μinch to 8μinch gold over 180μinch nickel.

PCB Land Pattern Recommendation

PCB land patterns are based on IPC-SM-782 standards when possible. The pad pattern shown has been developed and tested for optimized assembly at RFMD; however, it may require some modifications to address company specific assembly processes. The PCB land pattern has been developed to accommodate lead and package tolerances.

PCB Metal Land Pattern

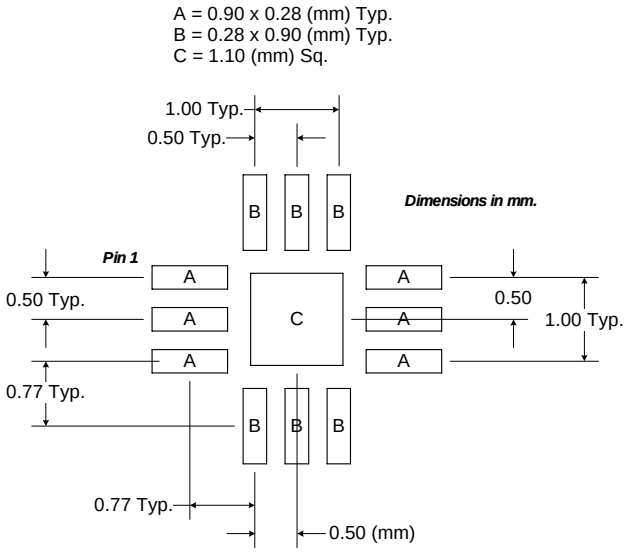


Figure 1. PCB Metal Land Pattern (Top View)

PCB Solder Mask Pattern

Liquid Photo-Imageable (LPI) solder mask is recommended. The solder mask footprint will match what is shown for the PCB metal land pattern with a 3mil expansion to accommodate solder mask registration clearance around all pads. The center-grounding pad shall also have a solder mask clearance. Expansion of the pads to create solder mask clearance can be provided in the master data or requested from the PCB fabrication supplier.

Thermal Pad and Via Design

The PCB metal land pattern has been designed with a thermal pad that matches the exposed die paddle size on the bottom of the device.

Thermal vias are required in the PCB layout to effectively conduct heat away from the package. The via pattern has been designed to address thermal, power dissipation and electrical requirements of the device as well as accommodating routing strategies.

The via pattern used for the RFMD qualification is based on thru-hole vias with 0.203mm to 0.330mm finished hole size on a 0.5mm to 1.2mm grid pattern with 0.025mm plating on via walls. If micro vias are used in a design, it is suggested that the quantity of vias be increased by a 4:1 ratio to achieve similar results.

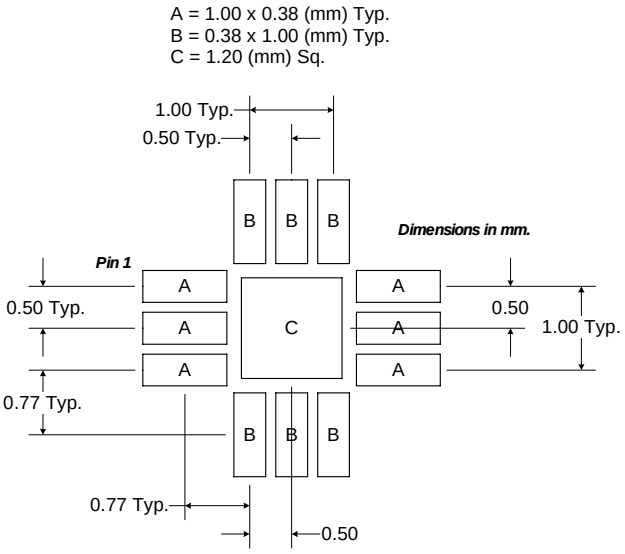


Figure 2. PCB Solder Mask Pattern (Top View)