

## GPS L2 Band Application

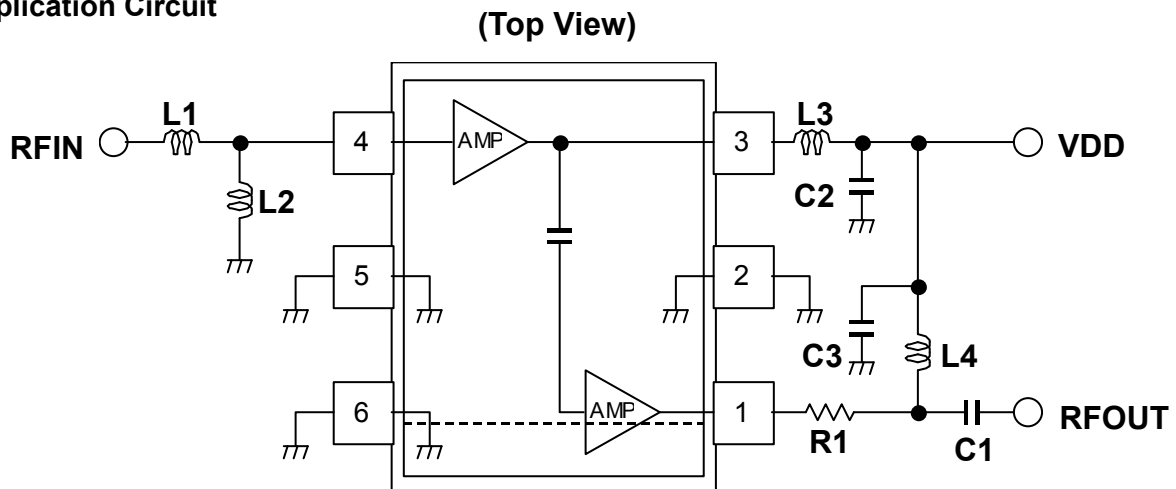
This is L2 Band (1.227GHz) application note of NJG1130KA1. This application note shows the typical electrical characteristics and application circuit.

### 1. Electrical Characteristics

General conditions:  $T_a=+25^{\circ}\text{C}$ ,  $V_{DD}=2.85\text{V}$ ,  $f_{RF}=1227\text{MHz}$ ,  $Z_s=Z_l=50\Omega$ , with application circuit

Parameters	Symbol	Conditions	Measurement Data	Units
Operation current	$I_{DD}$	RF OFF	5.03	mA
Small signal gain	Gain		31.4	dB
Noise figure	NF	Exclude PCB and connector losses (0.10dB)	1.02	dB
Output power at 1dB compression point	P-1dB		+9.6	dBm
3rd order output intercept point	OIP3	$f=1227+1227.1\text{MHz}$ , $P_{in}=-35\text{dBm}$	+14.8	dBm
Input VSWR	VSWR <sub>i</sub>		2.30	
Output VSWR	VSWR <sub>o</sub>		1.49	

### 2. Application Circuit

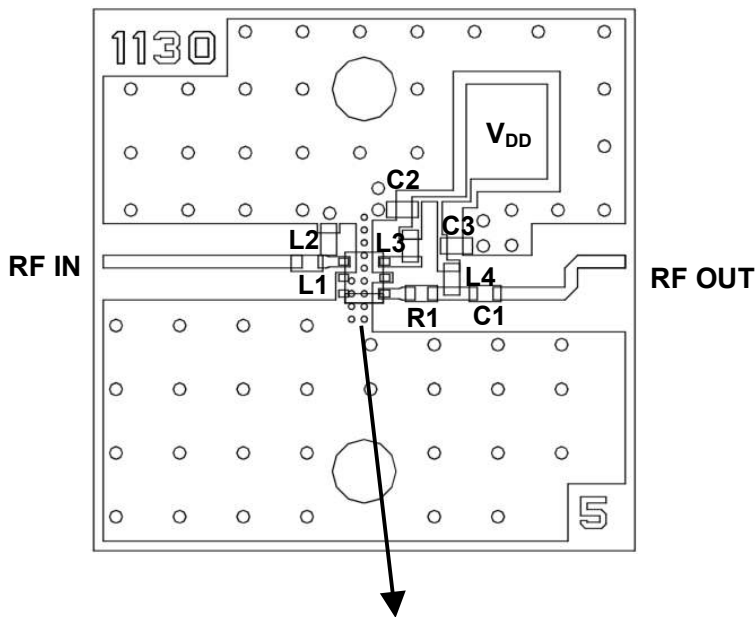


#### NOTES:

- L1 and L2 form the input matching circuit. The LNA has integrated coupling and DC-blocking capacitor at the input.
- L3 is a matching inductor of the integrated 1<sup>st</sup> amplifier and 2<sup>nd</sup> amplifier. It should be connected to the terminal3 as close as possible.
- L4 is an output matching inductor.
- C1 is a coupling and DC-blocking capacitor at the output.
- C2 and C3 are bypass capacitors. They should be connected between L3 and L4. C2 should be placed to the side of L3, and C3 should be placed to the side of L4, and should not be directly connected L3 and L4.
- R1 is a stability resistor at high frequency, and it should be connected to the terminal1.
- Ground terminal (No.2, 5, 6) should be connected to the ground plane as close as possible for good RF performance.
- For good performance, the terminal1,3 and 4 should not be coupled though floating-capacitance which exists between RF transmission lines.

## 3. TEST PCB LAYOUT

(Top View)



### Parts List

PART ID	Constants	備考
L1	15nH	TAIYO-YUDEN HK1005
L2	27nH	TAIYO-YUDEN HK1005
L3	10nH	TAIYO-YUDEN HK1005
L4	33nH	TAIYO-YUDEN HK1005
C1	100pF	MURATA GRM15
C2	1000pF	MURATA GRM15
C3	1000pF	MURATA GRM15
R1	27ohm	1005size

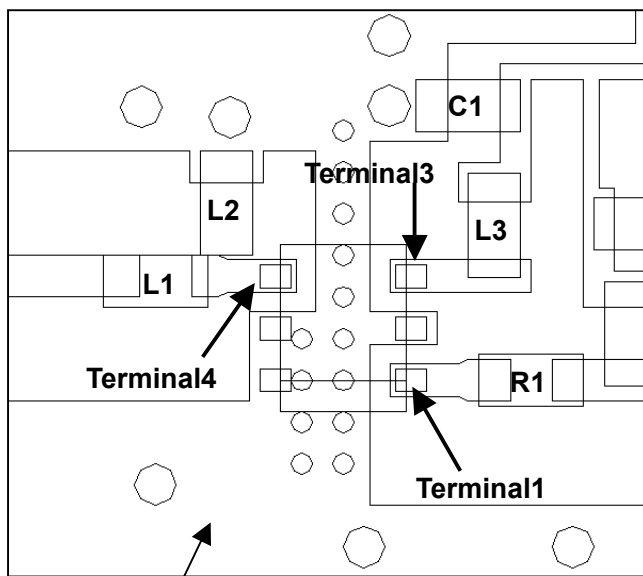
PCB (FR-4): t=0.2mm

MICROSTRIP LINE WIDTH

=0.4mm ( $Z_0=50\Omega$ )

PCB SIZE

=17.0mm × 17.0mm

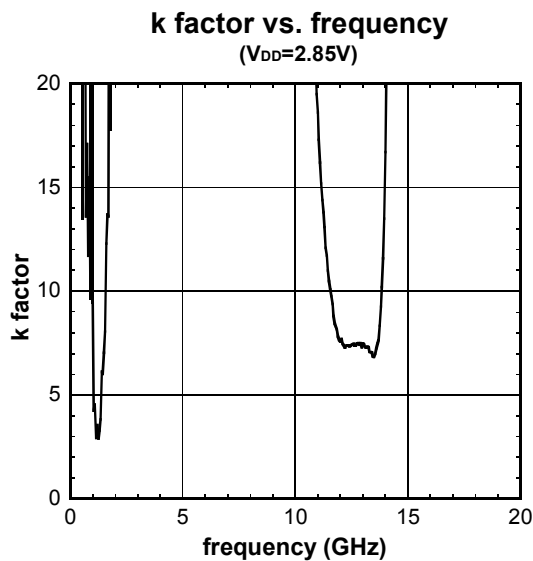
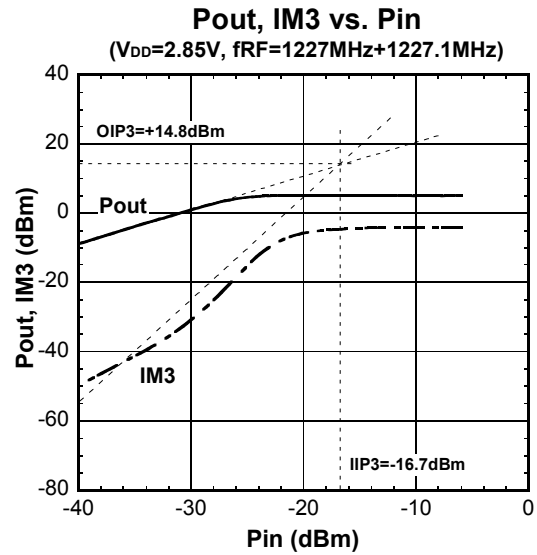
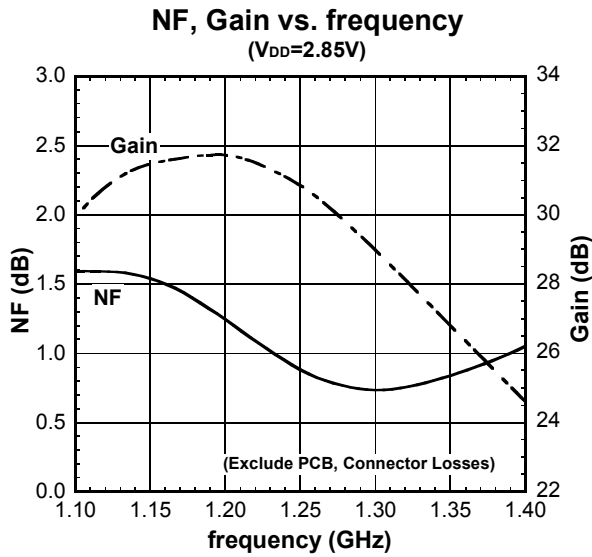
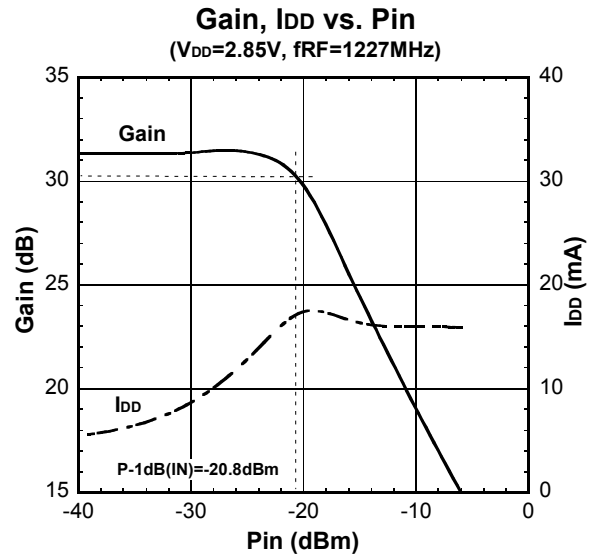
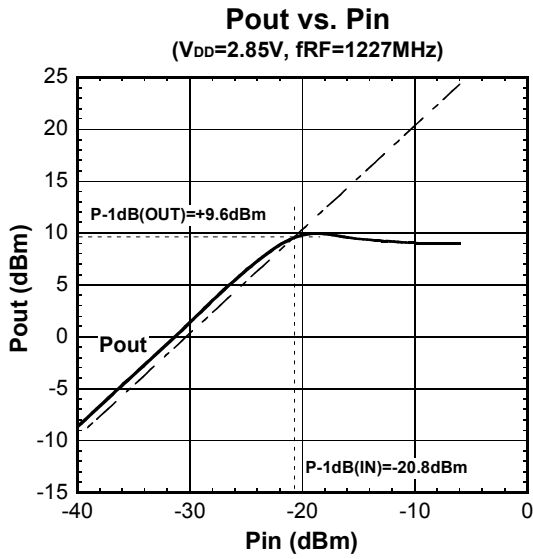


Ground plane

### PRECAUTION:

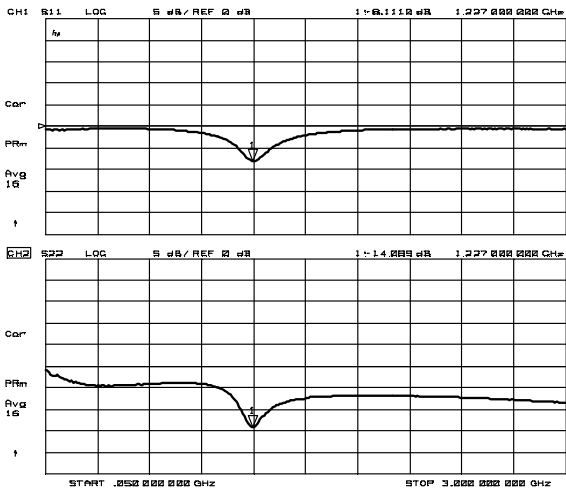
- [1] For good performance, the terminal1, 3 and 4 should not be coupled though floating-capacitance which exists between RF transmission lines.
- [2] In order not to couple with terminal 1, 3 and 4, please layout ground pattern under the IC.
- [3] C2 should be placed to the side of L3, and C3 should be placed the side of L4. They should be connect between L3 and L4, should not be directly connected L3 and L4.

## 4.1 ELECTRICAL CHARACTERISTICS

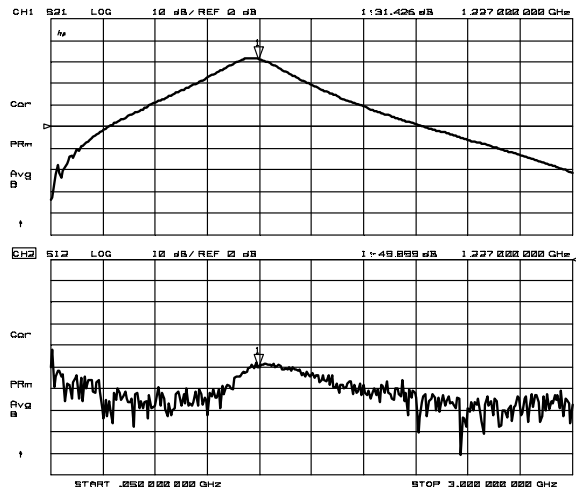


## 4.2 ELECTRICAL CHARACTERISTICS

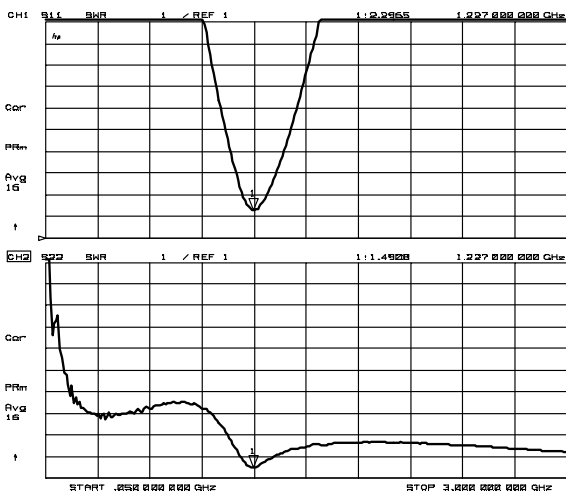
Condition : Ta=+25°C, V<sub>DD</sub>=2.85V, Z<sub>s</sub>=Z<sub>l</sub>=50ohm



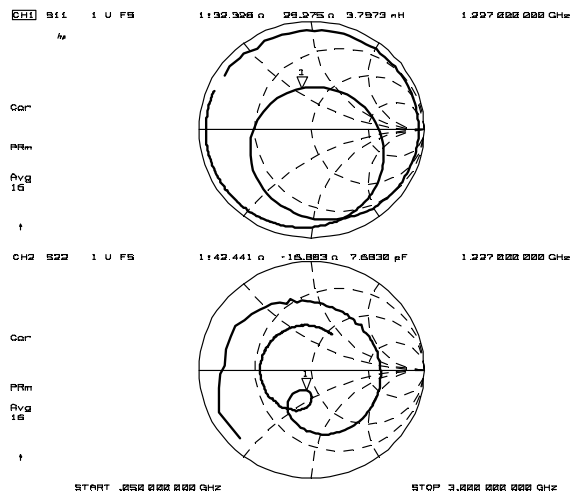
**S11. 22**



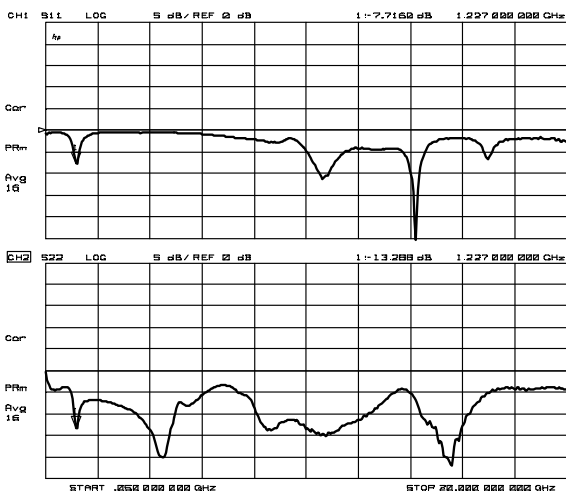
**S21. 12**



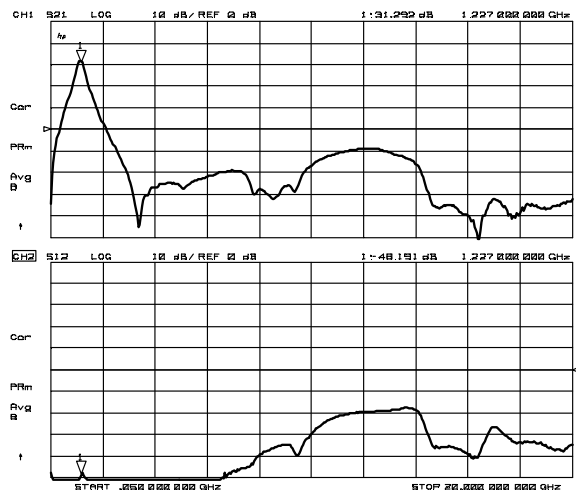
**VSWR**



**Zin. Zout**



**S11. S22 (f=50MHz~20GHz)**



**S21. S12 (f=50MHz~20GHz)**