國 立 交 通 大 學

電子工程學系 電子研究所碩士班

碩 士 論 文

應用於超寬頻3.1-10.6 GHz之無線接收端之低雜訊放大器之設計

An ultra-wideband CMOS LNA for 3.1 to 10.6 GHz wireless receivers

研究生: 陳懿範

指導教授: 荊鳳德 博士

中華民國九十六年五月

應用於超寬頻3.1-10.6 GHz之無線接收端低雜訊放大器之設計

An ultra-wideband CMOS LNA for 3.1 to 10.6 GHz wireless receivers

研 究 生：陳懿範　　Student: Yi-Fan Chen

指導教授：荊鳳德　博士　　Advisor: Dr.Albert Chin

國立交通大學

電子工程學系　電子研究所碩士班

碩士論文

A Thesis

Submitted to Department of Electronics Engineering & Institute of Electronics

College of Electrical and Computer Engineering

National Chiao Tung University

in Partial Fulfillment of the Requirements

for the Degree of Master

in

Electronics Engineering

May 2007

HsinChu, Taiwan, Republic of China

中華民國九十六年五月

應用於超寬頻3.1-10.6 GHz之無線接收端之低雜訊放大器之設計

學生：陳懿範　　指導教授：荊鳳德 博士

國立交通大學

電子工程學系電子研究所

摘要

本論文研製一個應用於超寬頻3.1-10.6 GHz的低雜訊放大器是採用電阻-電容回授串接電感做輸入匹配，而在輸出端是用current buffer做匹配。本研究是以0.18微米互補式金氧半製程實現。此低雜訊放大器是以三級放大為主架構，第一級為RC-feedback with series inductive peaking結構，是為了增加頻寬，第二級為cascode結構，可以增加平均順向增益(S_{21})，第三級則是current buffer，主要是在輸出端做匹配。為了能在所應用的頻段內達到相對的平坦增益，在前兩級中利用shunt peaking 的方法去實現。供應電壓V_{DD}為1.8伏特時，整個電路功率消耗約為17.02mW，及包含pad的情況下整個電路大小約為0.51 mm^2。本研究的低雜訊放大器所量測的規格，平均順向增益(S_{21})在3.1-10.6GHz時為6.73dB-13.20dB，逆向隔離(S_{12})為-39dB以下，S11為-7dB以下，S22約為-9.6dB以下，而平均雜訊指數約為5.3dB。

An ultra-wideband CMOS LNA for 3.1 to 10.6 GHz wireless receivers

Student: Yi-Fan Chen Advisor: Dr. Albert Chin
Department of Electronics Engineering & Institute of Electronics
Nation Chiao Tung University

Abstract

A 3.1-10.6 GH_Z low noise amplifier is applied for ultra-wideband, it introduces RC feedback with series inductive peaking for input matching. And current buffer is used for output matching. This research is fabricated in 0.18-μ m CMOS process. Three amplified stages are formed for main topology in low noise amplifier. The first stage introduces RC-feedback with inductive peaking configuration, it can improve the bandwidth. The second stage introduces cascode configuration, it can improve the average forward S_{21}. The third stage introduces current buffer configuration, it is used for output matching. Relatively flat gain is essential over the entire desired band. The low noise amplifier introduces the shunt peaking to achieve the above purpose. The total power dissipation of the chip is about 17.02mW at power supply 1.8 volt. The chip size included pad is 0.51 mm^2. The measurement result of this study expect that the forward S_{21} is between 6.73dB and 13.20dB at 3.1-10.6GHz, the reverse isolation S_{12} is under -39dB, the magnitude of S_{11} is under -7 dB, the magnitude of S_{22} is under -9.6dB, and the noise figure is 5.3dB.

誌謝

本論文得以完成，首先要感謝我的指導老師 荊鳳德 教授，在兩年的碩士研究生涯裡，給予我豐富的指導與照顧，不論是研究上與生活裡都讓我在這兩年裡獲得許多的收穫。

我還要感謝張慈學長、張國慶學長、王鴻偉學長與陳科閔學長他們在研究上與學業上給我的幫助，讓我得以順利完成碩士研究。也要感謝建宏學長、彬舫學長、軍宏學長、佩諭、正彥以及實驗室大家，因為有你們的陪伴與支持，讓我度過愉快又充實的這兩年。

最後，我要對我的父母獻上最高的敬意與謝意，感謝家人們對我的栽培、支持與鼓勵，才讓我有機會能接觸這一切並且完成我的學業與研究。

Contents

Figure Captions

Chapter 1 Introduction

Chapter 2 Basic Concept in RF Design

Chapter 3 How to design Basic Low-Noise Amplifiers

Chapter 4 UWB CMOS LNA Design

Chapter 1

Introduction

1.1 UWB CMOS Receivers

Ultra-Wideband (UWB) is a technology for transmitting information spread over a large bandwidth (>500 MHz) that should, in theory and under the right circumstances, be able to share spectrum with other users.

Common Definitions

UWB: Total BW > 1.5 GHz , or Fractional bandwidth $= \frac{(f_H - f_L)}{f_c} > 25\%$.

Narrowband: $\frac{(f_H - f_L)}{f_c} < 1\%$.

FCC Definition of UWB

Modulation type not specified, but the transmitted signal must meet the BW requirements at all times during its transmission. So frequency hopping and frequency swept systems are not allowed to meet the large bandwidth requirements. Systems that achieve large bandwidth other than using pulses (such as very high rate DSSS) are allowed.

Total BW > 500 MHz., Fractional bandwidth（measured at the -10dB points）, or $\frac{(f_H - f_L)}{f_c} > 20\%$.

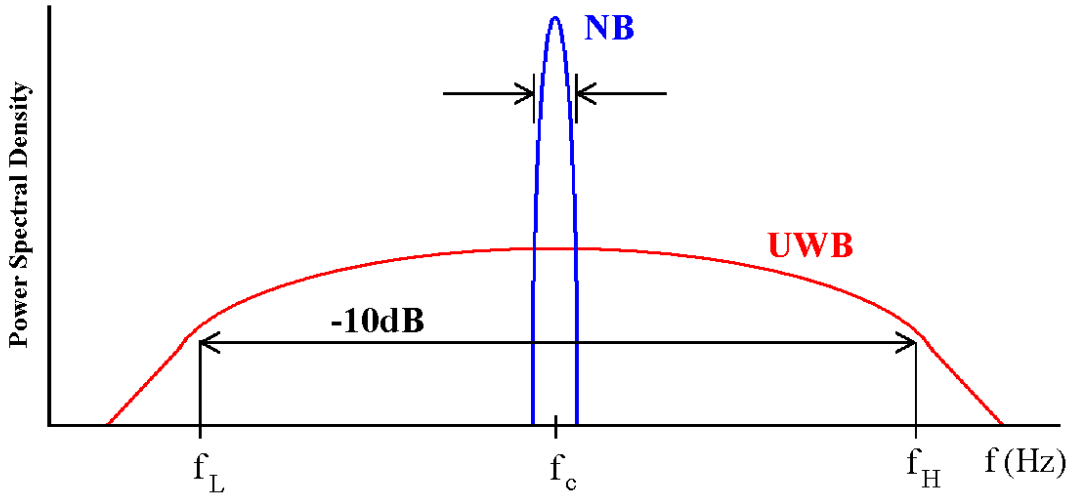


Fig. 1-1 The comparison with narrow band and UWB

From Fig.1-1, compare to traditional narrow band systems with communication, UWB technology has the promising ability to provide extremely high data rate performance in multi-user network applications, relativity immune to multipath cancellation effects as observed in mobile and in-building environments, and low interference to existing narrowband systems due to low power spectral density. In other words, UWB has the advantages of low power consumption, low cost (nearly “all digital”, with minimal RF electronics.), and a low probability of detection (LPD) signature. Since the FCC has allocated 7.5 GHz of spectrum for unlicensed use of UWB devices in the 3.1 to 10.6 GH_Z frequency band, the low noise amplifier needs to amplify the received UWB signal with sufficient gain and as little as possible. Due to the electricity, noise and other parameters have strict demands, RF integrated circuit use GaAs MESFET, Bi CMOS, hetero-junction bipolar transistor

(HBT), and PHEMT, etc to achieve these demands. In fact, in the past, most designer introduces the GaAs process to design their products for GaAs excellent high frequency parameter (like mobility), but with this work at [1] [2] [3], recently the sub-micro CMOS process like 0.25 micrometer, 0.18 micrometer or 0.13 micrometer CMOS process , reducing the minimum channel length and increasing the unity gain cut off frequency, also has acceptable high frequency parameters. Besides, CMOS process's cost is cheaper than other process's, we just focus on the CMOS technology. [4]

1.2 Motivation

As wireless communications become wide-spread, our research aims to deliver cost effective high-speed wideband applications with greater connectivity and robustness. No matter what we produce with wireless communication technology, such as cordless, cellular, global positioning satellite (GPS), wireless phones, wireless local area network (LAN), wireless modems, and RFID tags, etc, those products all need to use radio frequency communication. So how to develop the low power, low cost, low noise radio frequency integrated circuits (RFIC) is very important. [5] Under other chapters, I will discuss the basic concepts of RF design, some basic low-noise amplifiers design for UWB step by step, and finally present my implementation of a low-noise amplifier.

Chapter 2

Basic Concept in RF Design

2.1 Linearity in RF Circuits

Active RF devices are ultimately non-linear in operation. When driven with a large enough RF signal the device will generate undesirable spurious signals. How much spurious generated by the device is dependent on the linearity of the device.

Any nonlinear transfer function can be mathematically written as series expansion of power terms unless the system contains memory. For simplicity, we assume that:

$$Y(t) \approx \beta_0 + \beta_1 X(t) + \beta_2 X^2(t) + \beta_3 X^3(t) + \cdots \quad (2.1)$$

When a sinusoid uses in a nonlinear system, the integer multiples of the input frequency will be exhibited by the output frequency components. For example, if $X(t) = \alpha\cos\omega t$, then

$$Y(t) = \beta_0 + \beta_1 \alpha\cos\omega t + \beta_2 \alpha^2 \cos^2\omega t + \beta_3 \alpha^3 \cos^3\omega t + \cdots \quad (2.2)$$

It can be written as

$$Y(t) = \left(\beta_0 + \frac{\beta_2\alpha^2}{2}\right) + \left(\beta_1 + \frac{3\beta_3\alpha^3}{4}\right)\cos\omega t + \frac{\beta_2\alpha^2}{2}\cos 2\omega t + \frac{\beta_3\alpha^3}{4}\cos 3\omega t + \cdots \quad (2.3)$$

The term which has the input frequency is named as “fundamental” and the higher-order terms the “harmonics”. We can observe that the amplitude of the *n*th harmonic consists of a term proportional to α^n.

Two signals with different frequencies are applied to a nonlinear system, called the two-tone test, is one common way to characterize the linearity of a circuit.

We assume that:

$$X(t) = x_1(t) + x_2(t) = \alpha_1 \cos\omega_1 t + \alpha_2 \cos\omega_2 t \tag{2.4}$$

Apply this tone to (2.1), we can get

$$Y(t) = \beta_0 + \underbrace{\beta 1\ (x1\ (t) + x2\ (t))}_{\text{fundamental}} + \underbrace{\beta_2 (x_1(t) + x_2(t))^2}_{\text{second-order}} + \underbrace{\beta_3 (x_1(t) + x_2(t))^3}_{\text{third-order}} + \cdots \tag{2.5}$$

The output in general exhibits some components that are not harmonics of the input frequencies. Called intermodulation (IM), this phenomenon arises from multiplication of the two signals when their sum is raised to a power greater than unity.

For example, the second-order terms can be expands as follows:

$$\left(x_1(t) + x_2(t)\right)^2 = \underbrace{\left(x_1(t)\right)^2}_{dc + HD2} + \underbrace{2\left(x_1(t)x_2(t)\right)}_{IM2} + \underbrace{\left(x_2(t)\right)^2}_{dc + HD2} \tag{2.6}$$

$\left(x_1(t)\right)^2$ term has a zero frequency (dc) and another at the second harmonic of the input (HD2),

$$\left(x_1(t)\right)^2 = (\alpha_1 \cos\omega_1 t)^2 = \frac{\alpha_1^2}{2}(1 + \cos 2\omega_1 t) \tag{2.7}$$

Expand (2.5),

$$Y(t) = \beta_0 + \beta_1\left(x_1(t)+x_2(t)\right) + \beta_2\left(\left(x_1(t)\right)^2 + 2\left(x_1(t)x_2(t)\right) + \left(x_2(t)\right)^2\right) +$$

$$\beta_3\left(\left(x_1(t)\right)^3 + 3\left(x_1(t)\right)^2\left(x_2(t)\right) + 3\left(x_2(t)\right)^2\left(x_1(t)\right) + \left(x_2(t)\right)^3\right) + \cdots \quad (2.8)$$

For amplifier, we desired only the terms at the input frequency. But expand (2.8) by (2.4), discarding DC terms and harmonics, there still are intermodulation products.

$$\omega \to 2\omega_1 \pm \omega_2 : \frac{3\beta_3\alpha_1^2\alpha_2}{4}\cos(2\omega_1 + \omega_2)t + \frac{3\beta_3\alpha_1^2\alpha_2}{4}\cos(2\omega_1 - \omega_2)t \quad (2.9)$$

$$\omega \to 2\omega_2 \pm \omega_1 : \frac{3\beta_3\alpha_2^2\alpha_1}{4}\cos(2\omega_2 + \omega_1)t + \frac{3\beta_3\alpha_2^2\alpha_1}{4}\cos(2\omega_2 - \omega_1)t \quad (2.10)$$

The mixing components (IM) will appear at the sum and difference frequencies, in a typical two-tone test, $\alpha_1=\alpha_2=\alpha$, and the ratio of the amplitude of the output third-order products to $\beta_1\alpha$ defines the IM distortion. If a weak signal accompanied by two strong interferers experiences third-order nonlinearity, then one of the IM products falls in the band of desired output if ω_1 is close in frequency to ω_2 and therefore cannot be easily filtered out. The effect is that third-order nonlinearity can change the gain, which is seen as gain compression. And the two tone $(2\omega_1 - \omega_2, 2\omega_2 - \omega_1)$ are usually referred to as third-order intermodulation terms (IM3 products). [6]

2.1.1 The 1-dB Compression Point and third-Order Intercept point

If an amplifier is driven hard enough the output power will begin to roll off resulting in a drop of gain known as gain compression, and the phenomenon is

showed as Fig.2-1. The measurement of gain compression is given by the 1dB gain compression point.

1dB Compression Point:

Like Fig.2-2, this parameter in one measure of the linearity of a device and is defined as the input power that causes a 1dB drop in the linear gain due to device saturation.

When operating within the linear region of a component, gain through that component is constant for a given frequency. As the input signal is increased in power, a point is reached where the power of the signal at the output is not amplified by the same amount as the smaller signal. At the point where the input signal is amplified by an amount 1 dB less than the small signal gain, the 1 dB Compression Point has been reached. A rapid decrease in gain will be experienced after the 1 dB compression point is reached. If the input power is increased to an extreme value, the component will be destroyed. Passive, nonlinear components such as diodes also exhibit 1 dB compression points. Indeed, it is the nonlinear active transistors that cause the 1 dB compression point to exist in amplifiers. Of course, a power level can be reached in any device that will eventually destroy it.

$$P1dB_{output} = P1dB_{input} + (Gain - 1)\ dBm \qquad (2.11)$$

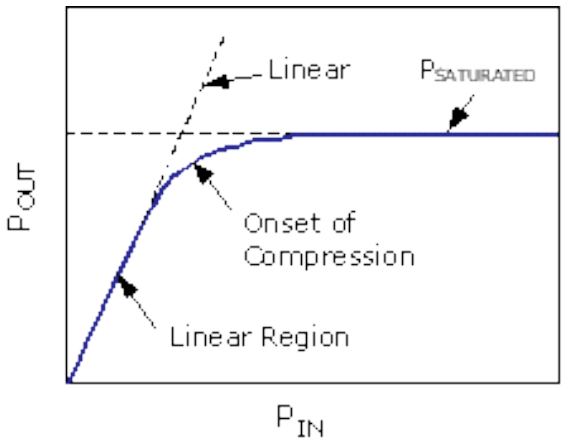


Fig. 2-1 The phenomenon of gain compression

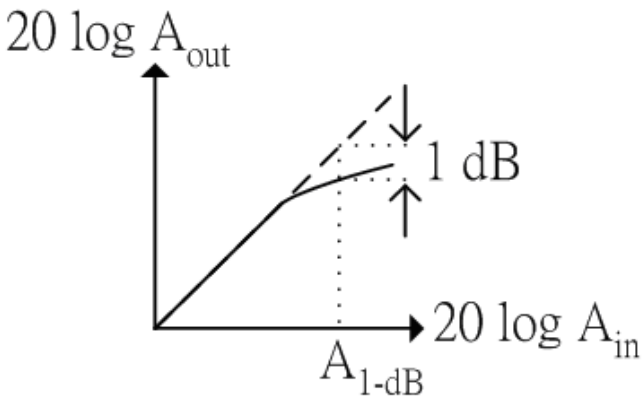


Fig. 2-2 Definition of the 1-dB compression point

A common rule of thumb for the relationship between the 3rd-order intercept point (IP3) and the 1 dB compression point (P1dB) is 10 to 12 dB. We will soon discuss what IP3 is.

Third-Order Intercept point:

A third-order intercept point (IP3 or TOI) is another measure for weakly nonlinear systems and devices, for example receivers, linear amplifiers and mixers. It is based on the idea that the device nonlinearity can be modeled using a low order

polynomial, derived by means of Taylor series expansion. The third order intercept point relates nonlinear products caused by the 3rd order term in the nonlinearity to the linearly amplified signal.

The third-order intercept point is a theoretical point where the amplitudes of the fundamental tones at $2\omega_1 - \omega_2$ and $2\omega_2 - \omega_1$ are equal to the amplitudes of the fundamental tones at ω_1 and ω_2.

From (2.5), when $\omega_1 = \omega_2 \rightarrow x_1(t) = x_2(t) = x_{in}$, the fundamental (F) of the third-order terms can be written as:

$$F = \beta_1 x_{in} + \frac{9}{4}\beta_3 x_{in}^3 \tag{2.12}$$

The linear component can be written as:

$$F = \beta_1 x_{in} \tag{2.13}$$

Compared to the third-order intermodulation term ($IM3 = \frac{3}{4}\beta_3 x_{in}^3$), since the IM3 terms rises three times as the fundamental (60dB/decade to 20dB/decade) if x_{in} is small, we can define a theoretical voltage ($x_{in} = v_{IP3}$) when these two tones will be equal:

$$\frac{\frac{3}{4}\beta_3 v_{IP3}^3}{\beta_1 v_{IP3}} = 1 \tag{2.14}$$

Therefore

$$v_{IP3} = 2\sqrt{\frac{\beta_1}{3\beta_3}} \tag{2.15}$$

Like Fig.2-3, the intercept point is obtained by plotting the output power versus the input power on dB scale. The input power at this point is called the *input third-order intercept point* (IIP3). If IP3 is specified at the output, it is called the *output third-order intercept point* (OIP3).

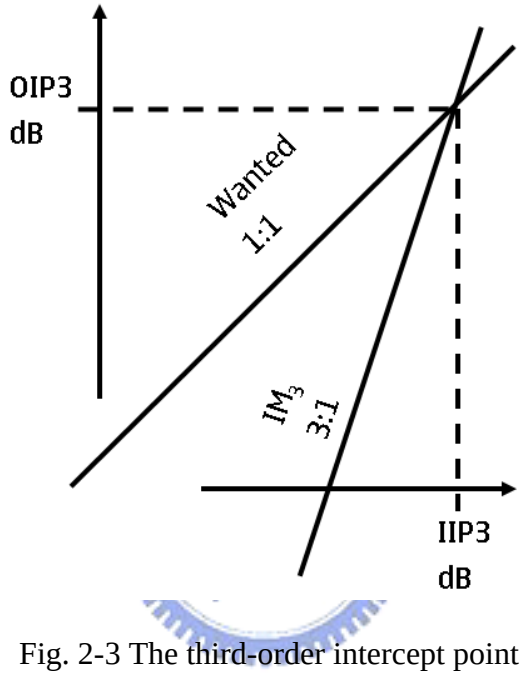


Fig. 2-3 The third-order intercept point

Two curves are drawn, one for the linearly amplified signal at an input tone frequency, one for a nonlinear product. On a logarithmic scale, "x to the power of n" translates into a straight line with slope of n. Therefore, the linearly amplified signal will exhibit a slope of 1. A 3rd order nonlinear product will increase by 3 dB in power, when the input power is raised by 1 dB. [7]

For instance, we have an output power called P1 at the fundamental frequency and an input power Pi called P3 at the IM3 frequency, and we know the IM3 terms

have a slope 3 times as the fundamental terms (60dB/decade to 20dB/decade). Thus, when The units of X-axis and Y-axis are dBm,

$$\frac{\mathrm{OIP3}-\mathrm{P_1}}{\mathrm{IIP3}-\mathrm{P_i}}=1 \text{ and } \frac{\mathrm{OIP3}-\mathrm{P_3}}{\mathrm{IIP3}-\mathrm{P_i}}=3 \tag{2.16}$$

Assume a device has power gain G, and G can be measured as:

$$\mathrm{G}=\mathrm{OIP3}-\mathrm{IIP3}=\mathrm{P_1}-\mathrm{P_i} \tag{2.17}$$

So we can solve IIP3:

$$\mathrm{IIP3}=\mathrm{P_1}+\frac{1}{2}[\mathrm{P_1}-\mathrm{P_3}]-\mathrm{G}=\mathrm{P_i}+\frac{1}{2}[\mathrm{P_1}-\mathrm{P_3}] \tag{2.18}$$

2.1.2 Cascaded Nonlinear Stages

Since in RF systems, signals are processed by cascaded stages, it is important to know how the nonlinearity of each stage is referred to the input of the cascade. Consider two nonlinear stages in cascade. As shown in Fig.2-4. Assuming that the input-output relationship is

$$y_1(t)=\alpha_1 x(t)+\alpha_2 x^2(t)+\alpha_3 x^3(t) \tag{2.19}$$

$$y_2(t)=\beta_1 y_1(t)+\beta_2 {y_1}^2(t)+\beta_3 {y_1}^3(t) \tag{2.20}$$

Substitute (2.19) into (2.20) results in the relation

$$y_2(t)=\alpha_1\beta_1 x(t)+(\alpha_3\beta_1+2\alpha_1\alpha_2\beta_2+{\alpha_1}^3\beta_3)x^3(t) \tag{2.21}$$

If we consider only the first- and third-order terms, then

$$A_{IP3}=\sqrt{\frac{4}{3}\left|\frac{\alpha_1\beta_1}{\alpha_3\beta_1+2\alpha_1\alpha_2\beta_2+{\alpha_1}^3\beta_3}\right|} \tag{2.22}$$

From equation (2.22) can be simplified if the two sides are inverted and squared:

$$\frac{1}{A^2_{IP3}} = \frac{1}{A^2_{IP3,1}} + \frac{3\alpha_2\beta_2}{2\beta_1} + \frac{\alpha_1^{\,2}}{A^2_{IP3,2}} \tag{2.23}$$

where $A_{IP3,1}$ and $A_{IP3,2}$ represent the input IP_3 points of the first and second stages, respectively. From the above result, we note that as α_1 increases, the overall IP_3 decreases. This is because with higher gain in the first stage, the second stage senses larger input levels, thereby producing much greater IM_3 products. [7]

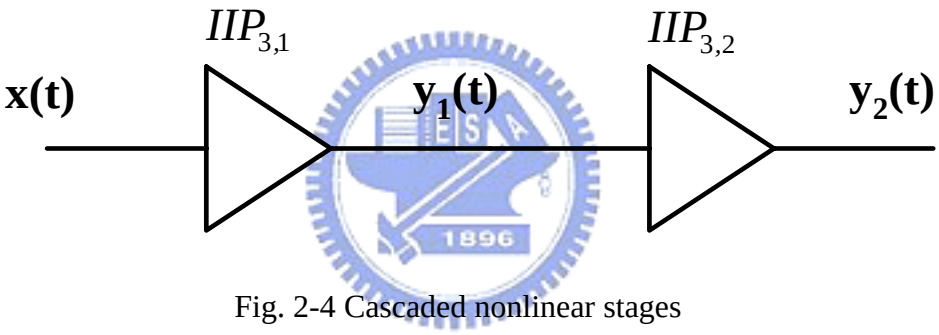


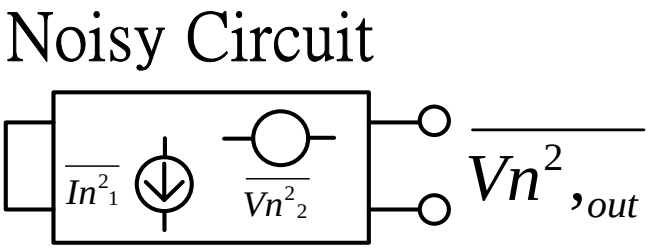
Fig. 2-4 Cascaded nonlinear stages

2.2 Noise

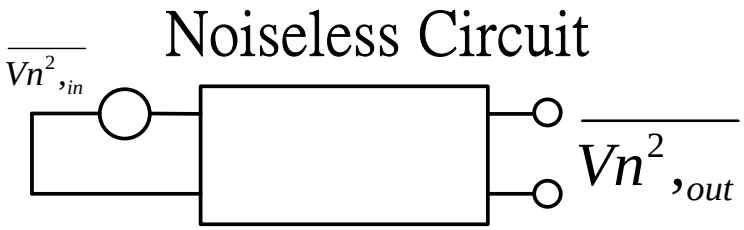
2.2.1 Noise Figure

Noise is usually generated by the random motions of charges or charge carriers in devices and materials. Because the noise process is random, one cannot identify a specific value of voltage at a particular time, and the only recourse is to characterize the noise with statistical measures, such as the mean-square or root-mean-square values. Because of having various noise sources in the circuit, we

need to simplify calculation of the total noise at the output. [8] Obviously, the output-referred noise does not allow a fair comparison of the performance of different circuits because it depends on the gain. According the circuit theory, we can use the input-referred noise of circuits to represent the noise of behavior in the circuits. To overcome the above confusion, we specify the “input-referred noise” of circuits. Illustrate conceptually in Fig. 2-5. To represent the effect of all noise sources in the circuit by a single noise source. The input-referred noise and the input signal are both multiplied by the gain as they are processed by the circuit. Thus, the input-referred noise indicates how much the input signal is corrupted by the circuit’s noise. The input-referred noise is a spurious quantity in that in cannot be measured at the input of the circuit. The two circuits of Figs. 2-5(a) and (b) are equivalent in mathematics but the real physical circuit is still that in Fig. 2-5(b). The noise of a two-port network can be modeled by two input noise sources: a series voltage source and a parallel current source. Generally, the correlation between the two sources must be taken into account. The situation is shown in Fig. 2-6, where a two-port network containing noise sources is represented by the same network with internal noise sources removed and with a noise voltage and current source connected at the input. It can be shown that this representation is valid for any source impedance, provided that correlation between the two noise sources is considered. [6]

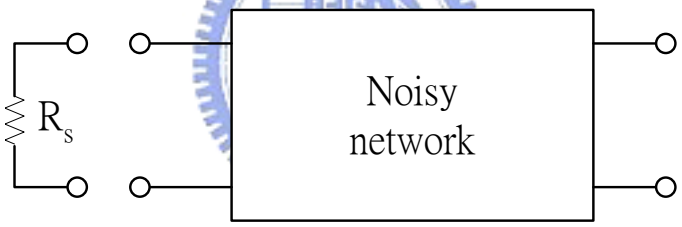


(a)

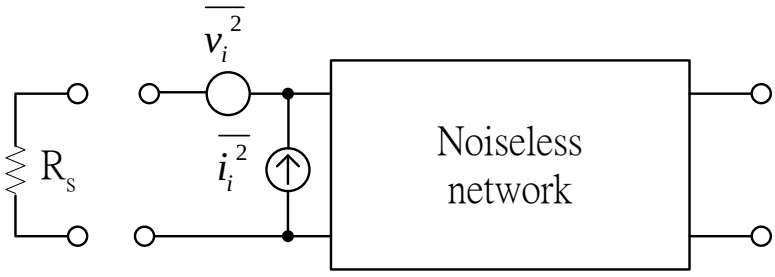


(b)

Fig. 2-5 Determination of input-referred noise voltage



(a)



(b)

Fig. 2-6 Representation of noise in a two-port network by equivalent input voltage and current sources

The noise figure (F) describes quantitatively the performance of a noisy microwave amplifier. The noise figure of a microwave amplifier is defined as the ration of the total available noise power at the output of the amplifier to the available noise power at the output due to thermal noise from the input termination R, where R is at the standard temperature$T = T_0 = 290°K$. The noise figure can be expressed in the form

$$F = \frac{P_{N_o}}{P_{N_i} G_A} \tag{2.24}$$

P_{N_o} is the total available noise power at the output of the amplifier, $P_{N_i} = kT_0B$ is the available noise power due to R at $T = T_0 = 290°K$ in a bandwidth B, and G_A is the available power gain.

G_A can be expressed in the form

$$G_A = \frac{P_{S_o}}{P_{S_i}} \tag{2.25}$$

P_{S_o} is the available signal power at the output, and P_{S_i} is the available signal power at the input, then noise figure can be written as

$$F = \frac{\frac{P_{S_i}}{P_{N_i}}}{\frac{P_{S_o}}{P_{N_o}}} \tag{2.26}$$

In other words, F can also be defined as the ratio of the available signal to noise (SNR) power ratio at the input to the available signal-to-noise power ratio at the output. A minimum noise figure is obtained by properly selecting the source

reflection coefficient of the amplifier. [6]

2.2.2 Noise Figure of LNA

For LNA the total noise figure is basically determined by the noise figure of the first stage of the LNA. The noise figure of a two-port amplifier is

$$F = F_{min} + \frac{r_n}{g_s}\left|y_s - y_{opt}\right|^2 \tag{2.27}$$

r_n is the equivalent normalized noise resistance of the two-port, $y_s = g_s + jb_s$ represents the normalized source admittance, and $y_{opt} = g_{opt} + jb_{opt}$ represents the normalized source admittance which results in the minimum noise figure, call F_{min}.

We can express y_s and y_{opt} in the terms of the reflection coefficients Γ_s and Γ_{opt}, thus

$$y_s = \frac{1 - \Gamma_s}{1 + \Gamma_s} \tag{2.28}$$

And

$$y_{opt} = \frac{1 - \Gamma_{opt}}{1 + \Gamma_{opt}} \tag{2.29}$$

So F can be written as

$$F = F_{min} + \frac{4r_n\left|\Gamma_s - \Gamma_{opt}\right|^2}{(1 - |\Gamma_s|^2)\left|1 + \Gamma_{opt}\right|^2} \tag{2.30}$$

The noise resistance r_n can be measured by reading the noise figure when $\Gamma_s = 0$, and the value F_{min}, which occurs when $\Gamma_s = \Gamma_{opt}$, and the source reflection coefficient that produces F_{min} can be determined accurately using network

analyzer. F_{min} is a function of the device operating current and frequency, and there is one value of Γ_{opt} associated with each F_{min}. [6]

For a given noise figure $F = F_i$, define the noise figure parameter N_i as

$$N_i = \frac{F_i - F_{min}}{4r_n}\left|1 + \Gamma_{opt}\right|^2 \tag{2.31}$$

2.2.3 Noise Figure of cascade LNA

In the case of a two-stage (or n-stage) amplifier, the noise figure of the second stage is reduced by G_{A1}. Therefore, the noise contribution from the second stage is small if G_{A1} is large and can be significant if the gain G_{A1} is low. In a design, a trade-off between gain and noise figure is usually made. [6]

The quantity M is known as the noise measure, and be defined as

$$M = \frac{F - 1}{1 - \frac{1}{G_A}} \tag{2.32}$$

It expresses the fact the lowest total noise figure is not obtained with $\Gamma_s = \Gamma_{opt}$ in each stage but with a value of Γ_s that minimizes each stage noise measure. In most designs, the values of G_A are sufficiently large so that the value of $\Gamma_s = \Gamma_{opt}$ that minimizes a given stage noise figure also minimizes its noise measure.

When two cascaded amplifier to determine which on should be used first in order to achieve the lowest noise figure, we should use the lowest M to be the first. We conclude that when two amplifiers are cascaded, the lower total noise figure is

achieved when the amplifier with the lowest value of M is connected at the input. For the case of a chain of n amplifiers, the total noise figure is written as

$$F = F_1 + \frac{F_2 - 1}{G_{A1}} + \frac{F_3 - 1}{G_{A1}G_{A2}} + \frac{F_4 - 1}{G_{A1}G_{A2}G_{A3}} + \cdots \tag{2.33}$$

If the amplifiers are identical with $F = F_1 = F_2 = \cdots F_n$ and $G_{A1} = G_{A2} = \cdots = G_{An}$, reduces to

$$F = 1 + \frac{F_1 - 1}{1 - \frac{1}{G_{A1}}} = 1 + M_1 \tag{2.34}$$

Chapter 3

How to design Basic Low-Noise Amplifiers

3.1 Consideration to design a LNA

The most important design considerations in a microwave transistor amplifier are stability power gain, bandwidth, noise, and dc requirements. It is important to select the correct dc operating point and the proper dc network topology in order to obtain the desired ac performance.

3.1.1 Stability

An unconditionally stable transistor will not oscillate with any passive termination. On the other hand, a design using a potentially unstable transistor requires some analysis and careful considerations so that the passive terminations produce a stable amplifier.

The stability of an amplifier, or its resistance to oscillate, is a very important consideration in a design and can be determined from the S parameters, the matching networks, and the terminations. In a two-port network, oscillations are possible when either the input or output port presents a negative resistance. In order to make it possible, $|\Gamma_{IN}| > 1$ *or* $|\Gamma_{OUT}| > 1$, and Γ_{IN} and Γ_{OUT} are defined as follows:

$$\Gamma_{IN} = S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{22}\Gamma_L} \tag{3.1}$$

$$\Gamma_{OUT} = S_{22} + \frac{S_{12}S_{21}\Gamma_S}{1 - S_{11}\Gamma_S} \tag{3.2}$$

If $S_{12} = 0$ that $|\Gamma_{IN}| = |S_{11}|$ and $|\Gamma_{OUT}| = |S_{22}|$. Hence, if $|S_{11}| > 1$ the transistor presents a negative resistance at the input, and if $|S_{22}| > 1$ the transistor presents a negative resistance at the output.

The two-port network is shown in Figure 3-1. For unconditional stability any passive load or source in the network must produce a stable condition. In terms of reflection coefficients, the conditions for unconditional stability at a given frequency are

$$|\Gamma_s| < 1 \tag{3.3}$$

$$|\Gamma_L| < 1 \tag{3.4}$$

$$|\Gamma_{IN}| < 1 \tag{3.5}$$

$$|\Gamma_{OUT}| < 1 \tag{3.6}$$

We can get the required conditions for the two-port network to be unconditionally stable by solving (3.3) to (3.6). [9]

$$k = \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |\Delta|^2}{2|S_{12}S_{21}|} \tag{3.7}$$

$$\Delta = S_{11}S_{22} - S_{12}S_{21} \tag{3.8}$$

Combine (3.1) and (3.2) to (3.7) and (3.8), a convenient way of expressing the necessary and sufficient conditions for unconditional stability is

$$k > 1 \tag{3.9}$$

$$|\Delta| < 1 \tag{3.10}$$

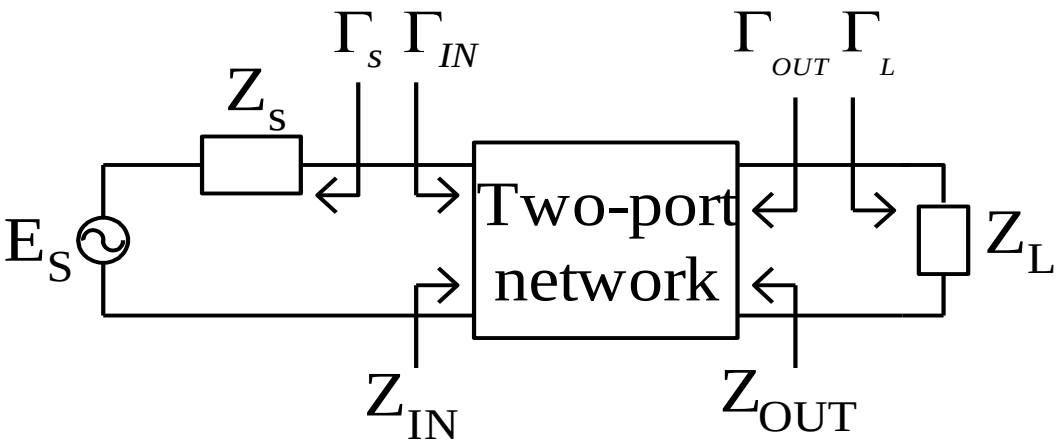


Fig. 3-1 Stability of two-port networks

3.1.2 Impedance Matching Networks

Consider the RF system shown in Fig. 3-2. Here the source and load are 50Ω (a very popular impedance), as are the transmission lines leading up to the IC. For optimum power transfer, prevention of ringing and radiation, and good noise behavior,

Fig. 3-3 illustrates a typical situation in which a transistor, in order to deliver maximum power to 50Ω load, must have the terminations Zs and Z_L. The input matching network is designed to transform the generator impedance (show as 50Ω) to the source impedance Zs, and the output matching network transforms the 50Ω

termination to the load impedance Z_L.

If a broadband matching is required, then other techniques may need to be used. An example of matching a transistor amplifier with a capacitive input is shown in Fig.3-4. Typically, reactive matching circuits are used because they are lossless and because they do not add noise to the circuit will only be matched over a range of frequencies and not at others. The series inductance adds an impedance of $j\omega L$ to cancel the input capacitive impedance. Note that, in general, when impedance is complex $(R + jX)$. Then to match it, the impedance must be driven from its complex conjugate $(R - jX)$.

A more general matching is required if the real part is not 50Ω. For example, if the real part of Z_{in} is less than 50Ω, then the circuit can be matching using the circuit in Fig. 3-5.

Series components will move the impedance along a constant resistance circle on the Smith Chart. Parallel components will move the admittance along a constant conductance circle. The input impedance of a circuit can be any values. In order to have the best power transfer into the circuit, it is necessary to match this impedance to the impedance of the source driving the circuit. The output impedance must be similarly matched. It is very common to use reactive components to achieve this impedance transformation, because they do not absorb any power or add noise. Thus,

series or parallel inductance or capacitance can be added to the circuit to provide an impedance transformation.

Although many different types of matching networks can be designed, the eight Ell sections (also denotes as L sections) shown in Figure 3-6 are not only simple to design but quite practical. The matching networks are lossless in order not to dissipate any of the signal power.

In any particular region on the Smith Chart, several matching circuits will work and others will not. This is illustrated in Figure 3-7, which shows what matching networks will work in which regions. Since more than one matching network will work in any region, how does one choose? There are a number of popular reasons for choosing one over another.

1. Sometimes matching component can be used as dc blocks (capacitors) or to provide bias currents (inductors).
2. Some circuits may result in more reasonable component values.
3. Personal preference. Not to be underestimated, sometimes when all paths look equal, you just have to shoot from the hip and pick one.
4. Stability. Since transistor gain is higher at lower frequencies, there may be a low-frequency stability problem. In such a case, sometimes a high-pass network (series capacitor, parallel inductor) at the input may be more stable.

5. Harmonic filtering can be done with a lowpass matching network (series *L*, parallel *C*). This may be important, for example, for power amplifiers. [7]

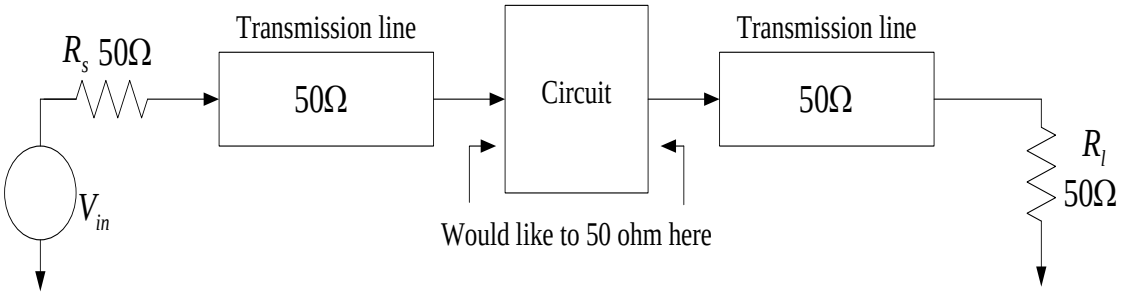


Fig. 3-2 Circuit embedded in a 50-Ω system

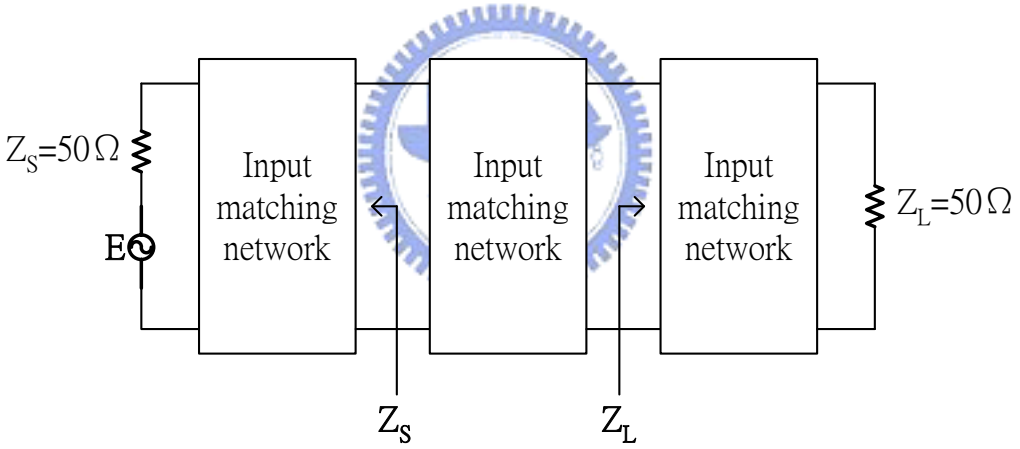


Fig. 3-3 Circuit embedded in a 50-Ω system with matching circuit

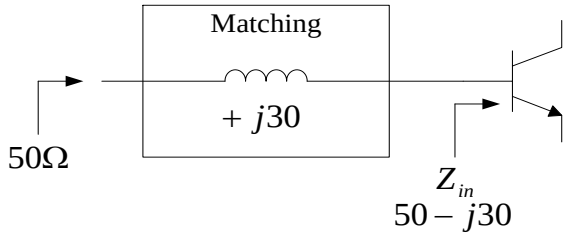


Fig. 3-4 Example of a very sample matching network

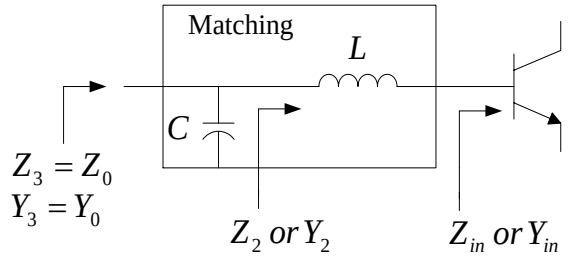


Fig. 3-5 A possible impedance-matching network

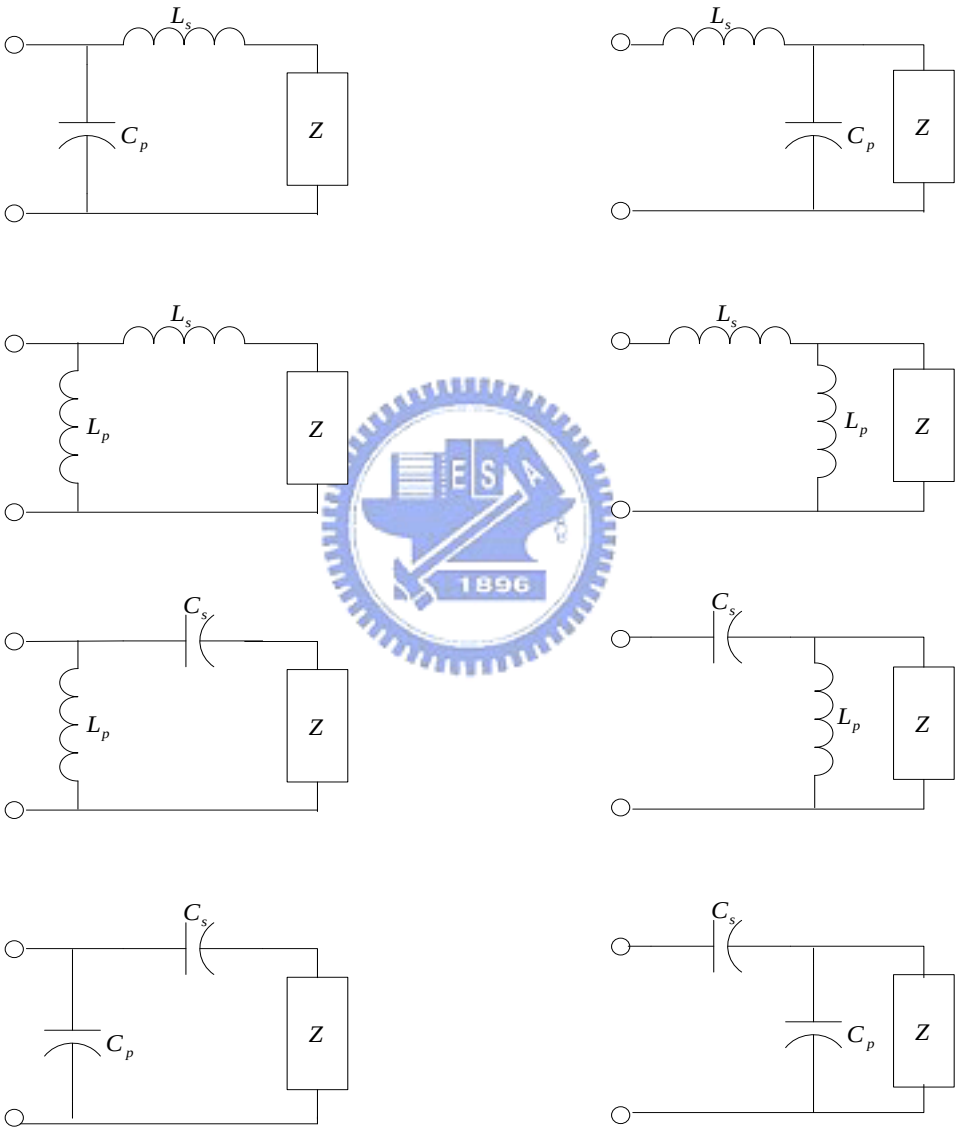


Fig. 3-6 The eight possible impedance-matching networks with two reactive

components

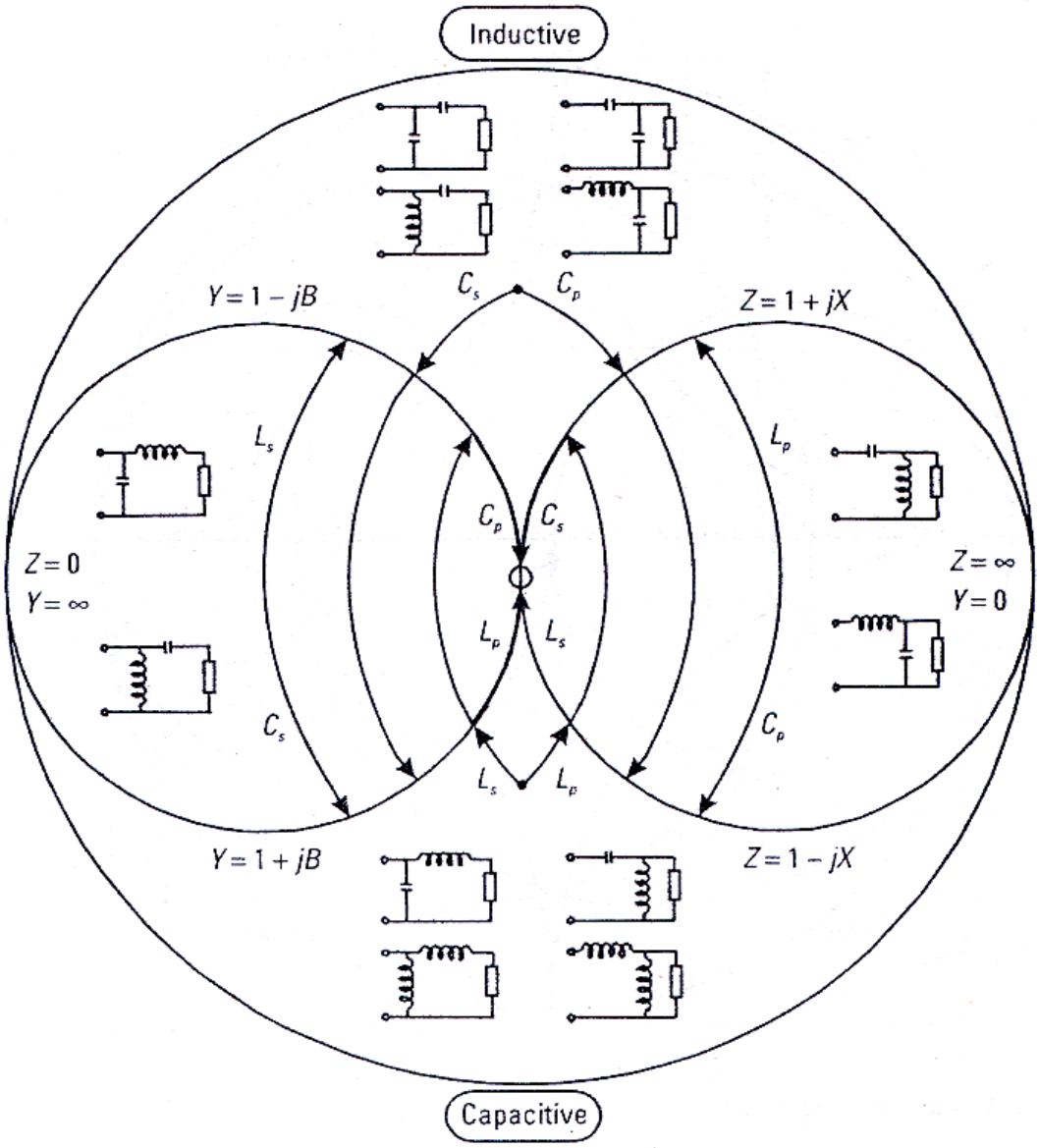


Fig. 3-7 Which ell matching networks will work in which regions

3.2 Conventional LNA design

3.2.1 Narrow band LNA design

In the design of low noise amplifier, there are many important goals. These include noise figure minimization, providing sufficient gain with good linearity, and the reasonable power consumption. Fig. 3-8 illustrates the input stage of the low noise amplifier with source degeneration. A simple calculation is

$$Z_{in} = s(L_g + L_s) + \frac{1}{sC_{gs2}} + \left(\frac{g_{m2}}{C_{gs_2}}\right)L_s \qquad (3.11)$$

If choose appropriate value of inductance and capacitance, then L_g+L_s and C_{gs} will resonate at certain frequency. By choosing L_s appropriately, the real term can be made equal to 50Ω . The gate inductance L_g is used to set the resonance frequency once Ls is chosen to satisfy the criterion of a 50Ω input impedance. The matching method in noise performance is better than using resistance termination of the input end.

The reverse isolation of low noise amplifier determines the amount of LO signal that leaks from the mixer to the antenna. The leakage arises from capacitive paths, substrate coupling, and bond wire coupling. In heterodyne receivers with a high first IF, the image-reject filter and the front-end duplexer significantly suppress the leakage because the LO frequency falls in their stop-band. In homodyne topology, the

leakage is attenuated primarily by the LNA reverse characteristics. Equation (3.7) suggests that stability improves as S_{12} decrease, i.e., as the reverse isolation of the circuit increases. The feedback can be suppressed through the use of a cascade configuration, but at the cost of a somewhat higher noise figure. The common-gate transistor in the Fig.3-8, M_1, plays two important roles by increasing the reverse isolation of the LNA: (1) it lowers the LO leakage produced by the following mixer, and (2) it improves the stability of the circuit by minimizing the feedback from the output to the input [10].

3.2.2 Wide-band LNA design

From Fig.3-9, the R_f is added as a shunt feedback element to the conventional cascade narrow band LNA and L_{load} is used as shunt peaking inductor at the output. The capacitor C_f is used for the ac coupling purpose. The source follower composed of M_3 and M_4, is added for measurement proposes only, and provides wideband output matching. C_1 and C_2 are ac coupling capacitor. The small-signal equivalent circuit at the input of the LNA is shown in Fig. 3-10. The resistor $R_{fM} = R_f /(1 - A_v)$ represents the Miller equivalent input resistance of R_f, where A_v is the open-loop voltage gain of the LNA. From equivalent circuit, the value of R_f can be much larger than that of the conventional resistance shunt-feedback. In the conventional resistance shunt-feedback, the size of R_f is limited as R_{fM} determines the input impedance. One

of the key roles of the feedback resistor R_f is to reduce the Q-factor of the resonating narrowband LNA input circuit. The Q-factor of the circuit shown in Fig. 3-10 can be approximately given by

$$Q_{WB} \approx \frac{1}{\left[R_S + \omega_T L_S + \frac{(\omega_0 L_g)^2}{R_{fM}} \right] \cdot \omega_0 \cdot C_{gs}} \tag{3.12}$$

From (3.12), and considering the inversely linear relation between the -3dB bandwidth and the Q-factor, the narrowband LNA in Fig.3-9 can be converted into a wideband amplifier by the proper selection of R_f. To design a wideband amplifier that covers a certain frequency band, the narrowband amplifier will be optimized at the center frequency. The feedback resistor R_f also provides its conventional roles of flattening the gain over a wider bandwidth of frequency with much smaller noise figure degradation [11].

Another wide-band LNA design schematic is shown in Fig.3-11. In Fig.3-11, is the LNA circuit schematic. We discuss this circuit step by step from the first stage. First, to make $1/g_m = 50\Omega$, the g_m value of common gate amplifier is going to be fixed at certain trans-conductance. An additional stage is required to provide sufficient gain over the desired band. A shunt feedback common source amplifier is used in the second stage for this purpose. The first step is the selection of transistor size and bias

condition of the M1 to yield $\mathrm{Re}|Z_{11}| = 1/g_m = 50\Omega$. This ensures input matching condition for wide-band of frequency. But this condition is violated with optimum noise condition. There is a trade-off between noise and impedance matching in the LNA circuit. One of the major problems in the wide bandwidth amplifier design is the limitation imposed by the gain-bandwidth product of the active device. We know that any active device has a gain roll off at high frequency because of the gate-drain and gate-source capacitance in the transistor. This effect degrades the forward gain as the frequency increases and eventually the transistor stops functioning as an amplifier at the high frequency. Therefore the second design step is the selection of optimal bias point of second stage of LNA so that it operates at its maximum f_T. In addition to this $|S_{21}|$ degradation with frequency other complications that arises in wide-bandwidth amplifier design includes, increase in reverse gain $|S_{12}|$ and noise figure at high frequency. Negative feedback configuration is used to reduce these effects and increase the bandwidth. An inductor L is connected in series with R_f such that after certain frequency the negative feedback decreases in proportion to the S_{21} roll-off. This technique improves gain flatness at high frequency. The load inductance of L_1 and L_2 replace the resistor load which is used conventionally. The magnitude of the inductor's impedance increases as frequency increases. This increase inductor impedance compensates the active device gain degradation that occurs at high

frequency [12].

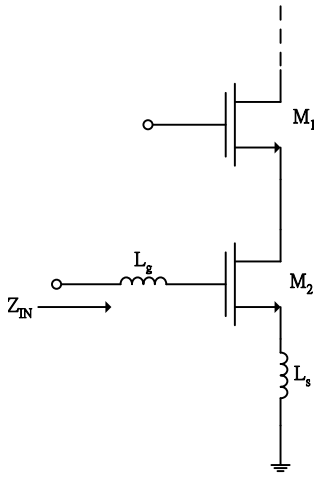


Fig.3-8 Common source stage use inductance degeneration

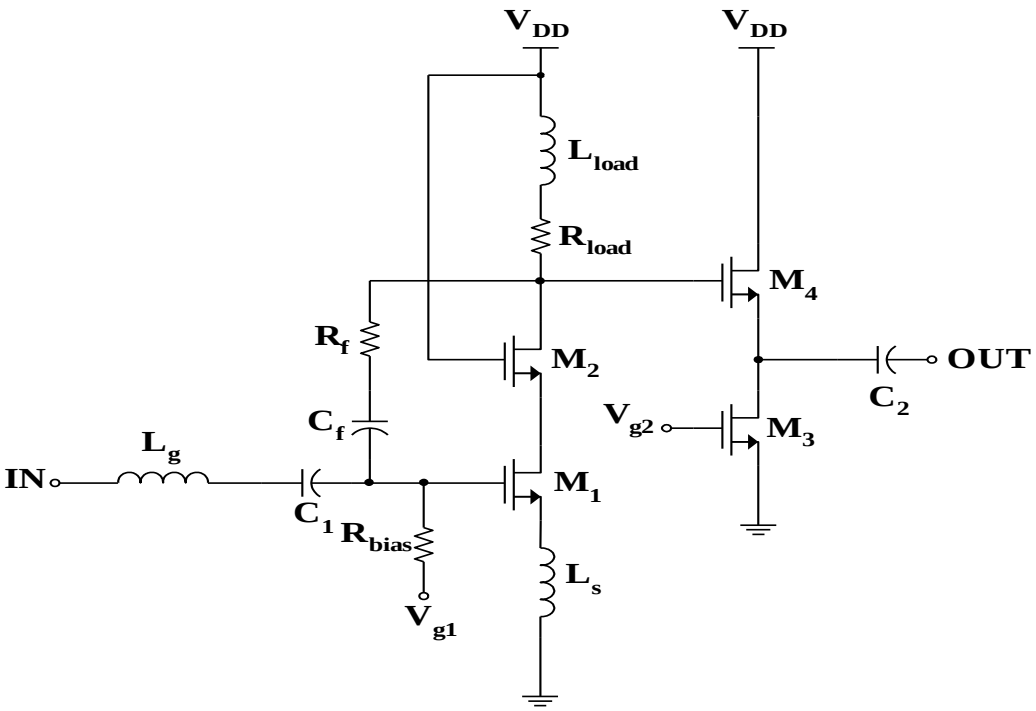


Fig.3-9 Wide-band LNA circuit schematic

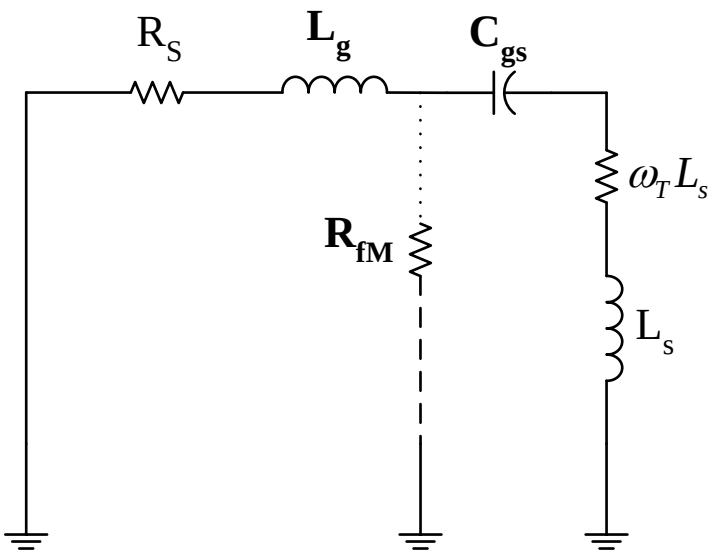


Fig. 3-10 Small-signal equivalent circuit at the input

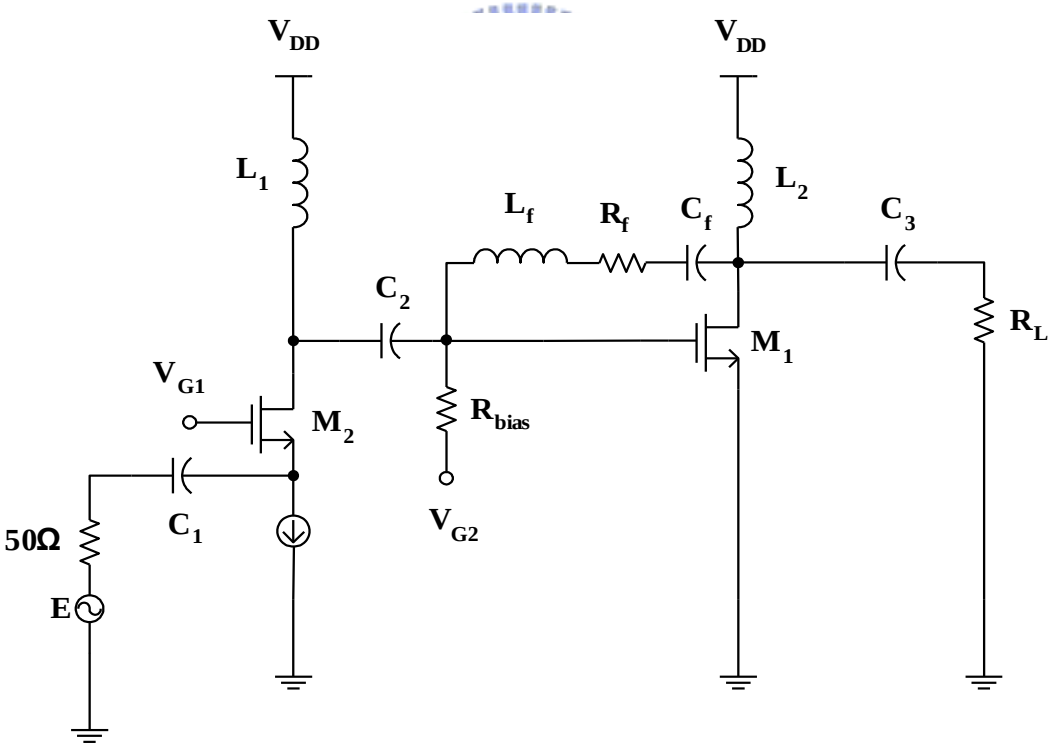


Fig. 3-11 Another wide-band LNA schematic

Chapter 4

UWB CMOS LNA Design

4.1 Circuit topology

In designing a broadband amplifier, feedback and distributed configuration are most widely used. In the chapter, feedback configuration was used instead of distributed configuration because it is more adequate for integration due to better uniformity and stability at frequencies below 12GHz. In addition, the cascode structure has been considered as the best topology for wideband applications because of its advantages

We design UWB LNA with cascode R-C feedback structure that the R-C feedback connected between input and Lg, like Fig. 4-1. From Fig. 4-1, we can observe that this is a three stage low noise amplifier. First stage used the cascode R-C feedback structure. The advantages of cascode structure are high gain, wider bandwidth, better stability and reverse isolation. The cascode configuration is being used to reduce the high frequency roll-off of the input devices due to the Miller effect. It can also be performed the input/output matching independently. The R-C feedback (R_f & C_f) in cascode circuit improves the S11 of the circuit and stabilizes the common-gate without reducing the gain. Above all, it can satisfy the requirements of wideband system for both noise and power simultaneously by carefully feedback

resistance.

The MOS M1 dominates the noise performance. These two MOS (M1 and M2) have little effects with each other. This LNA for UWB applications was designed using TSMC 0.18um RF CMOS technology and Fig. 4-2 shows the chip layouts.

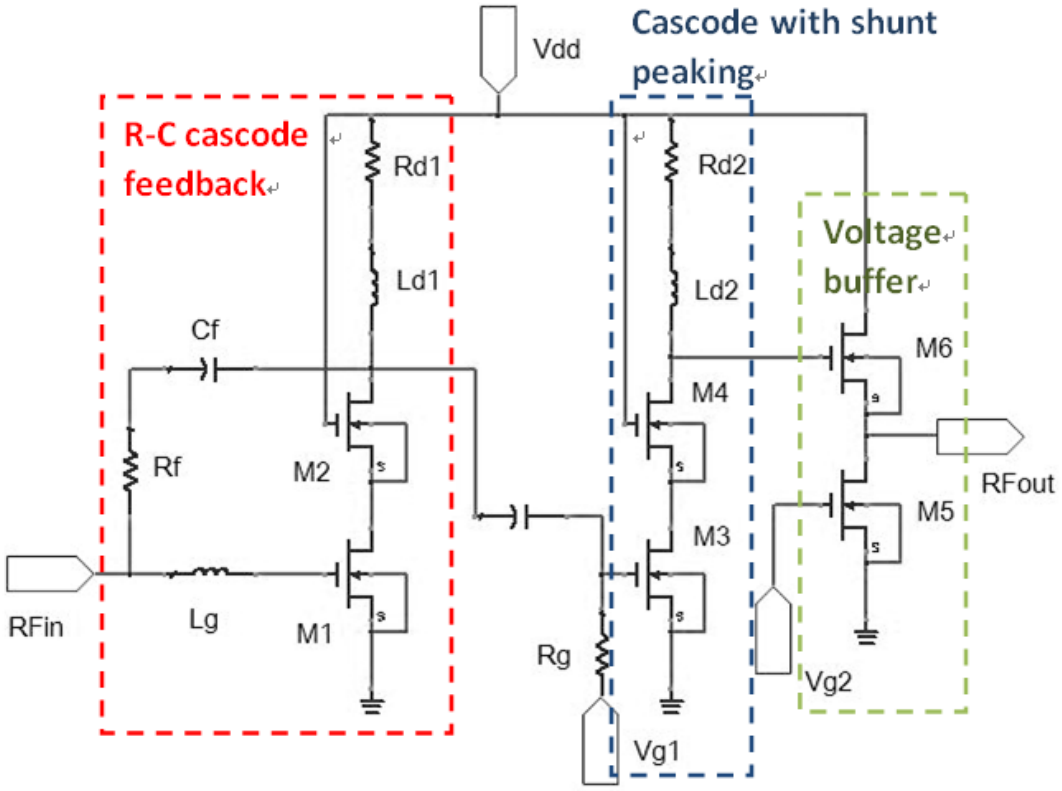


Fig. 4-1 Circuits diagram

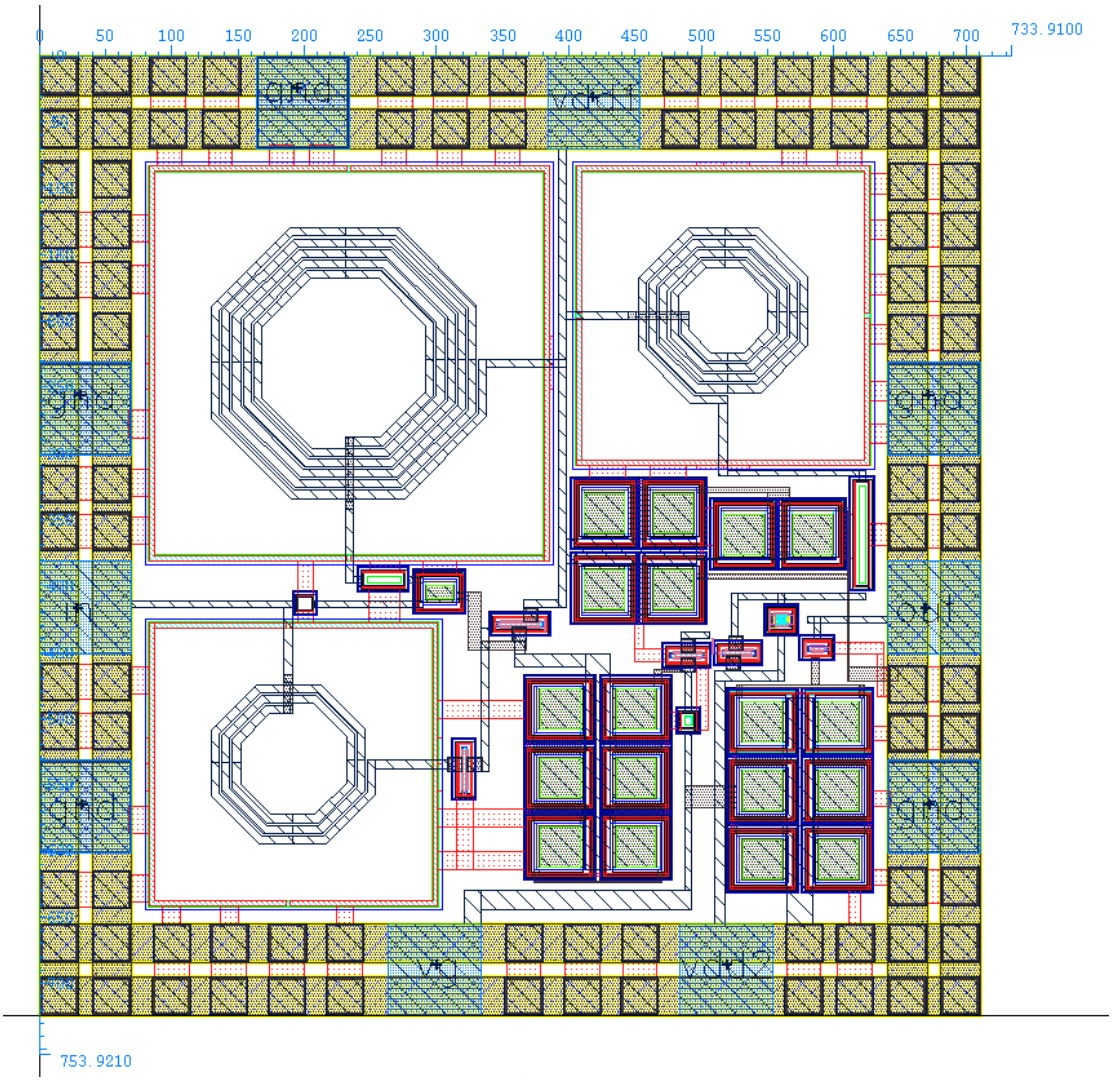


Fig. 4-2 The chip layout

4.2 Design procedures

4.2.1 Noise analysis

Three main contributors determine the noise performance of the R-C cascode feedback topology: the gate inductor Lg, the feedback resistor Rf and the noise of the amplifying device M1. We optimize the noise contribution from M1 relies on the choice of its width for a given bias current.

MOS transistor noise sources are shown in Fig. 4-3(a). The noise generator $\overline{i^2_d}$ is

$$\overline{i^2_d} = 4kT\left(\frac{2}{3}g_m\right)\Delta f + k\frac{i^a{}_D}{f}\Delta f \tag{4.1}$$

$4kT\left(\frac{2}{3}g_m\right)\Delta f$ is thermal noise component, and $k\frac{i^a{}_D}{f}\Delta f$ is flicker noise component. And noise generator $\overline{i^2_g}$ is

$$\overline{i^2_g} = 2qI_G\Delta f \tag{4.2}$$

The input-referred in a conventional way and replaced with two correlated noise generators, as shown in Fig. 4-3(b), because the current gain of the source degeneration is $\beta(j\omega) = \frac{\omega_T}{j\omega}$, and the cutoff frequency is $\omega_T \approx \frac{g_m}{C_{gs}}$, so $\overline{i^2_{ia}}$ is

$$\overline{i^2_{ia}} = \overline{i^2_g} + \frac{j\omega C_{gs}}{g_m}\overline{i^2_d} \tag{4.3}$$

And $\overline{v^2_{ia}}$ is

$$\overline{v^2_{ia}} = \frac{\overline{i^2_d}}{g_m} \tag{4.4}$$

One common design that the R-C feedback connected between Lg and

MOS1's gate (Fig. 4-4(a)) and this work (Fig. 4-4(b)) are showed below, and we can compare with their equivalent circuit of the input stage for noise calculation.

In Fig. 4-4(a), the feedback of the common design topology:

$$v_{ib1} = v_{ia} \tag{4.5}$$

$$i_{ib1} = i_{ia} + i_f \tag{4.6}$$

Then,

$$v_{i1} = v_{ib1} + i_{ib1} \cdot R_{Lg} + v_{Lg} = v_{ia} + i_{ia} \cdot R_{Lg} + v_{Lg} + i_f \cdot R_{Lg} \tag{4.7}$$

$$i_{i1} = i_{ib1} = i_{ia} + i_f \tag{4.8}$$

Thus, the total equivalent noise voltage and current of this feedback is

$$\overline{v^2_{i1}} = \overline{v^2_{ia}} + \overline{v^2_{Lg}} + \overline{i^2_{ia}} \cdot R_{Lg}{}^2 + \overline{i^2_f} \cdot R_{Lg}{}^2 \tag{4.9}$$

$$\overline{i^2_{i1}} = \overline{i^2_{ia}} + \overline{i^2_f} \tag{4.10}$$

In Fig. 4-4(b), the feedback of this work topology:

$$v_{ib2} = v_{ia} + i_{ia} \cdot R_{Lg} + v_{Lg} \tag{4.11}$$

$$i_{ib2} = i_{ia} \tag{4.12}$$

Then,

$$v_{i2} = v_{ib2} = v_{ia} + i_{ia} \cdot R_{Lg} + v_{Lg} \tag{4.13}$$

$$i_{i2} = i_{ib2} + i_f = i_{ia} + i_f \tag{4.14}$$

Thus, the total equivalent noise voltage and current of this feedback is

$$\overline{v^2_{i2}} = \overline{v^2_{ia}} + \overline{v^2_{Lg}} + \overline{i^2_{ia}} \cdot R_{Lg}{}^2 \tag{4.15}$$

$$\overline{i^2_{i2}} = \overline{i^2_{ia}} + \overline{i^2_{f}} \tag{4.16}$$

where

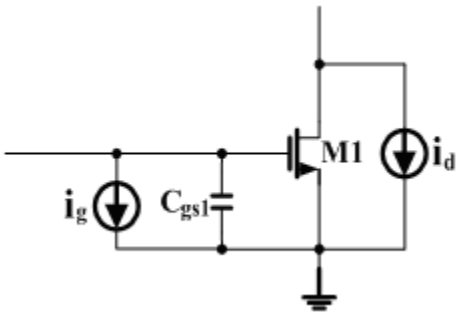
$$\overline{i^2_{f}} = 4kT\frac{1}{R_f}\Delta f \tag{4.17}$$

$$\overline{v^2_{Lg}} = 4kTR_{Lg}\Delta f \tag{4.18}$$

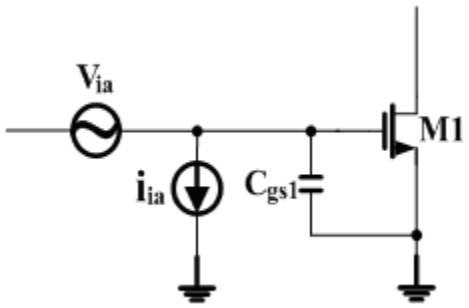
The noise factor is

$$F = 1 + \frac{\overline{v^2_{i}}}{4kTR_S\Delta f} + \frac{\overline{i^2_{i}}}{4kT\frac{1}{R_S}\Delta f} \tag{4.19}$$

From equation (4.11) and (4.15), we can observe that the total noise of the common design is greater than this work due to the $\overline{i^2_{f}} \cdot R_{Lg}{}^2$ item. Fig. 4-5 shows the noise figure: the common design versus this work. We can observe that the noise figure of the topology of this work is better than the common one.



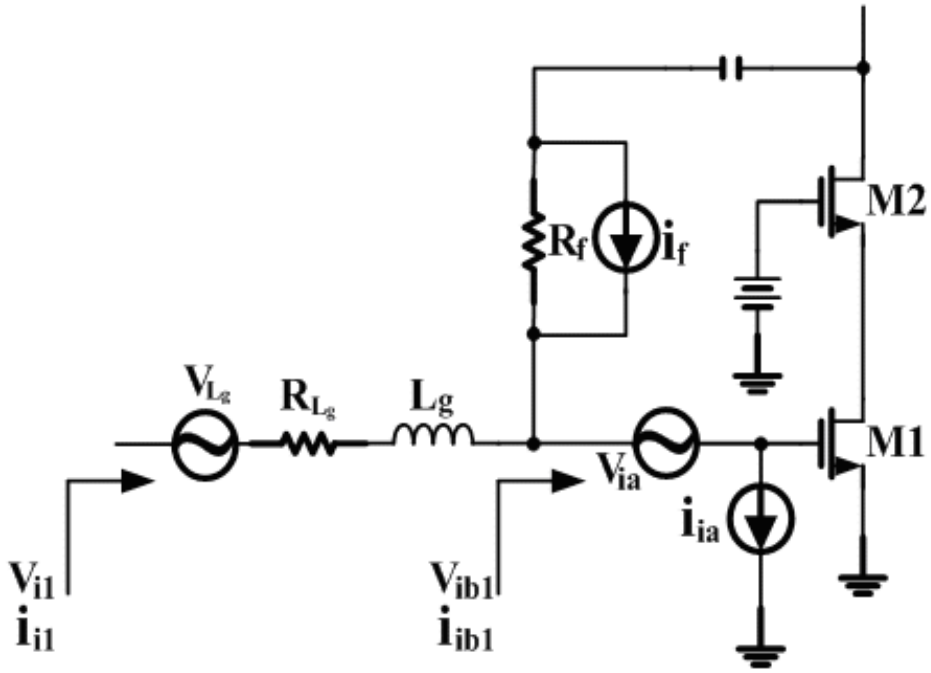
(a)



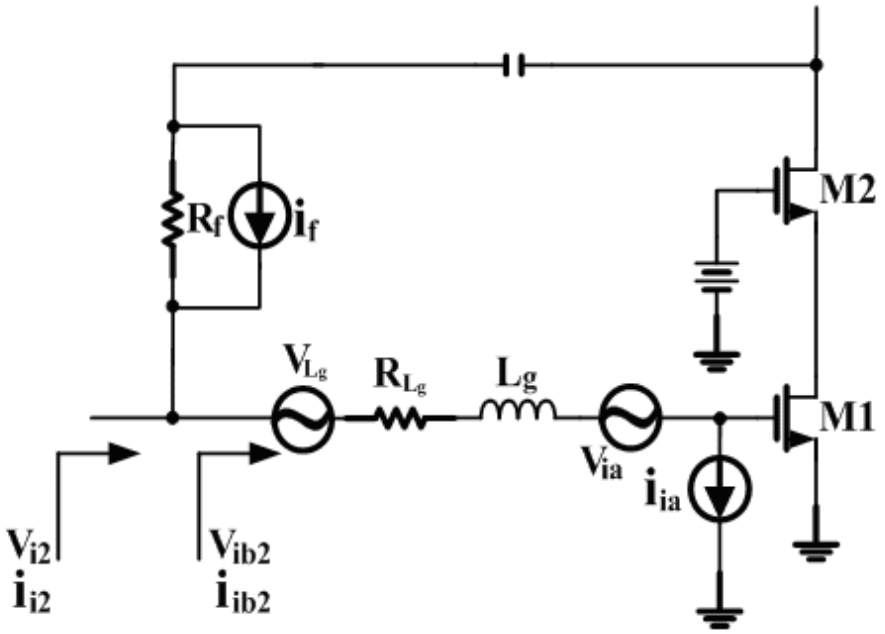
(b)

Fig. 4-3 Noise model for the amplifying transistor M1 (a) M1 noise sources (b)

Input-referred equivalent noise generators



(a)



(b)

Fig. 4-4 Equivalent circuit of the input stage for noise calculation

(a) one common design (b) this work

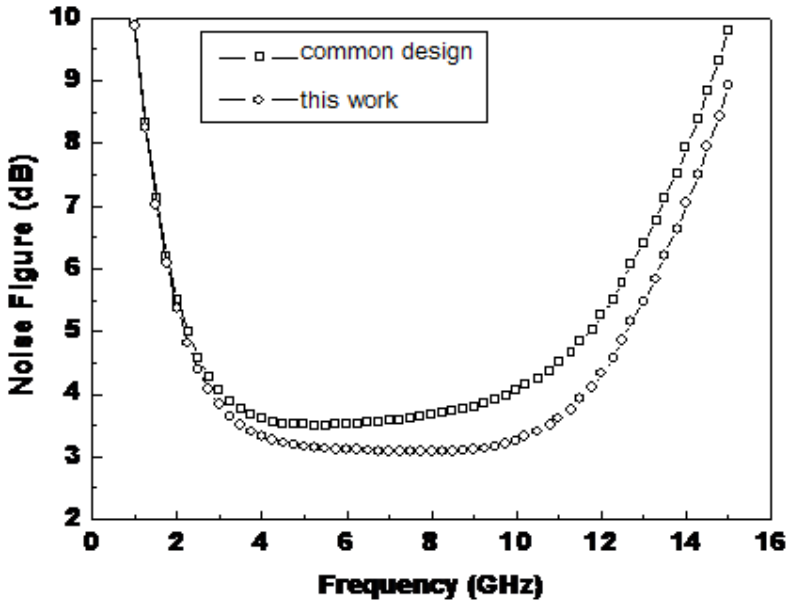


Fig. 4-5 Noise figure: One common design v.s. This work

4.2.2 Input and output match

Input matching:

In feedback topology, the small signal equivalent is shown in Fig. 4-6, where

$$Z'_{in} = sL_g + 1/sC_{gs1} \tag{4.20}$$

$$g_{m,eff} \cong g_{m1} \times 1 = g_{m1} \tag{4.21}$$

$$Z_L = R_{d1} + sL_{d1} \tag{4.22}$$

$$Z_f = R_f + 1/sC_f \tag{4.23}$$

For this configuration, the input impedance and the gain can be calculated to be

$$Z_{in} = \left[\frac{Z'_{in} \cdot Z_f}{Z_f + (A_V - 1) \cdot Z'_{in}} \right] \tag{4.24}$$

$$A_V = \frac{Z_L + g_{m,eff} \cdot Z_f \cdot Z_L}{Z_L - Z_f} \tag{4.25}$$

Since the circuit parameters Z'_{in} and A_V are frequency dependant, the characteristics of Z_f will vary accordingly over the frequency band. Thus, select Z_f and combination of R、C components，perfect matching and gain can be achieved.

Output matching:

The third stage is decided to use source follower buffer to make $\frac{1}{g_{m4}} = 50\,\Omega$ for output match.

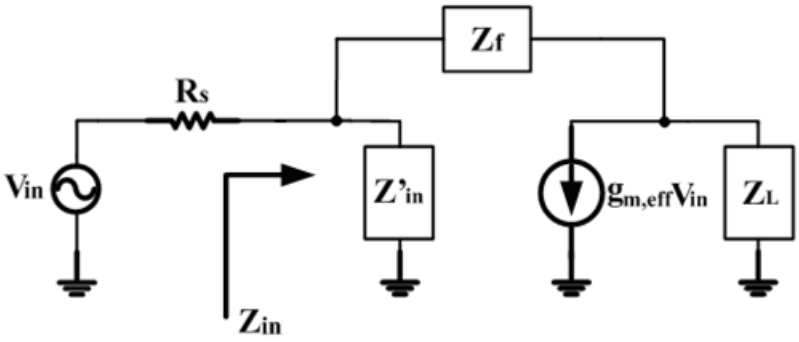


Fig. 4-6 Feedback configuration

4.2.3 Shunt peaking

A model of shunt peaking amplifier is shown in Fig. 4-7. The capacitance *C* may be taken to represent all the loading on the output node, including that of a subsequent stage. The resistance *R* is the effective load resistance at that node and the inductor provides the bandwidth enhancement. It's clear from the model that the transfer function v_{out}/i_{in} is just the impedance of the *RLC* network, so it should be straightforward to analyze. The addition of an inductance in series with the load resistor provides an impedance component that increases with frequency, which helps offset the decreasing impedance of the capacitance, leaving net impedance that remains roughly constant over a broader frequency range than that of the original *RC* network. The impedance of the *RLC* network may be written as

$$Z(s) = (sL + R) /\!/ \frac{1}{sC} = \frac{R[s(L/R)] + 1}{s^2 LC + sRC + 1} \tag{4.26}$$

L_d must be sizable to have large gain and must be small so that it resonates C_{out} out of band. R_d is chosen to place the zero frequency ($\omega_z = R_d/L_d$) as close as

to the lower edge of the band to improve the gain.

We introduce a factor *m*, defined as the ratio of the *RC* and *L/R* time constant:

$$m = \frac{RC}{L/R} \tag{4.27}$$

Then, the transfer function becomes

$$Z(s) = \frac{R[s(L/R)]+1}{s^2LC + sRC + 1} = \frac{R(\tau s + 1)}{s^2\tau^2 m + s\tau m + 1} \tag{4.28}$$

where $\tau = L/R$.

The magnitude of the impedance, normalized to the DC value as a function of frequency, is then

$$\frac{|Z(j\omega)|}{R} = \sqrt{\frac{(\omega\tau)^2 + 1}{(1-\omega^2\tau^2 m)^2 + (\omega\tau m)^2}} \tag{4.29}$$

so that

$$\frac{\omega}{\omega_1} = \sqrt{\sqrt{(-\frac{m^2}{2} + m + 1)^2 + m^2} + (-\frac{m^2}{2} + m + 1)} \tag{4.30}$$

where ω_1 is the uncompensated -3dB frequency. Because the load is designed to achieve flat gain over the whole bandwidth, chose $m = 1 + \sqrt{2} \approx 2.414$, then can lead to a bandwidth that is about 1.72 times as large as the un-peaked case. Therefore, both a maximally flat response and a substantial bandwidth extension can be obtained simultaneously at least for the shunt-peaked amplifier. [8]

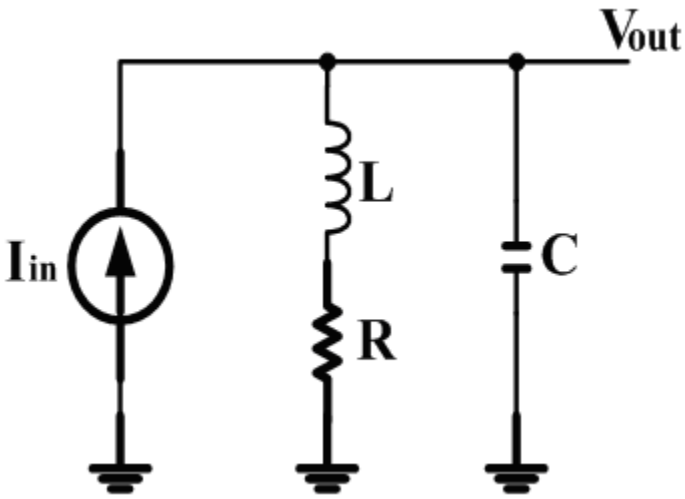


Fig. 4-7 Model of shunt-peaked amplifier

4.3 Simulation Result

Fig.4-8 shows the simulated input and output reflection coefficients. S11 is lower than -10dB between 3.1 and 10.6GHz. The output buffer achieves excellent matching such that S22 is lower than -12.57dB from 3.1GHz to 10.6 GHz. Fig. 4-9 is the power gain versus frequency, and the maximum power gain is 18.43dB in our simulation results. Since the output source follower drives a matched load, the voltage gain of the core amplifier is exactly 6dB higher than S21. The -3dB bandwidth is 0.4~9.9GHz for the simulation. The noise figure (NF) of this UWB LNA is shown in Fig.4-10. The noise figure is as low as 2.8dB at 10.6GHz, while the average noise figure in-band is about 3.7dB. Fig.4-11 and 4-12 show the simulated reverse isolation S12 and stability factor respectively. The two-tone test results for third-order intermodulation distortion are shown in Fig.4-13. The test is performed at 5.5GHz. IIP3 is to 5.19dBm, and the input referred 1-dB compression point (ICP) is -2dBm. These results imply excellent linearity of our LNA. The proposed UWB LNA dissipate 17.2mW with a power supply of 1.8V.

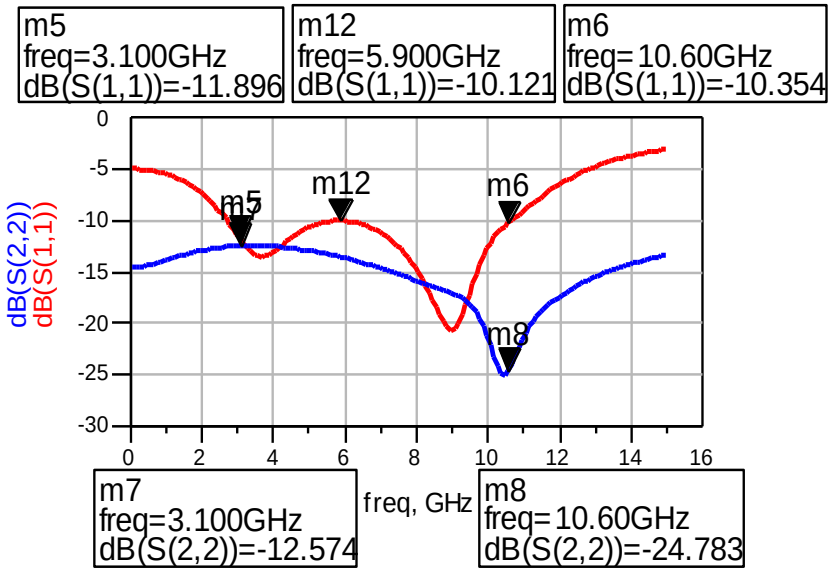


Fig. 4-8 Simulated S11&S22

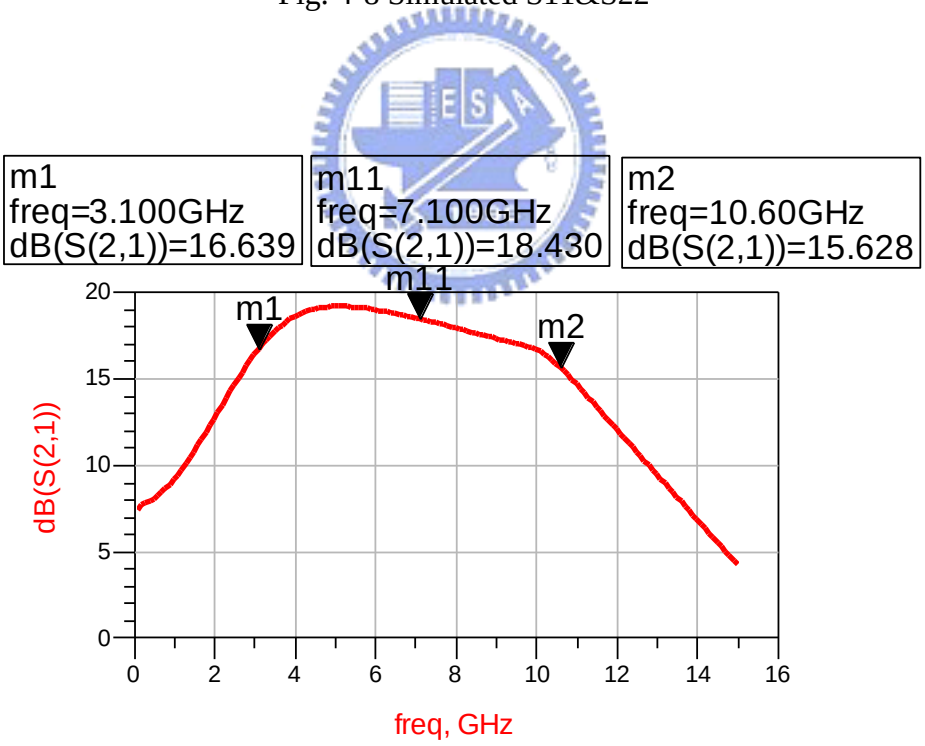


Fig. 4-9 Simulated S21

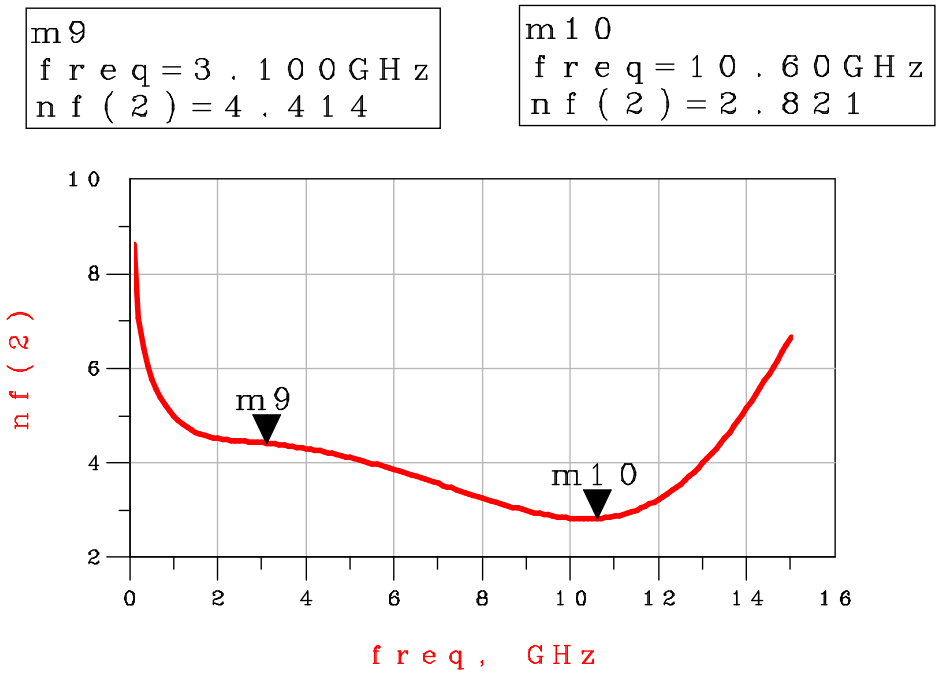


Fig. 4-10 Simulated NF

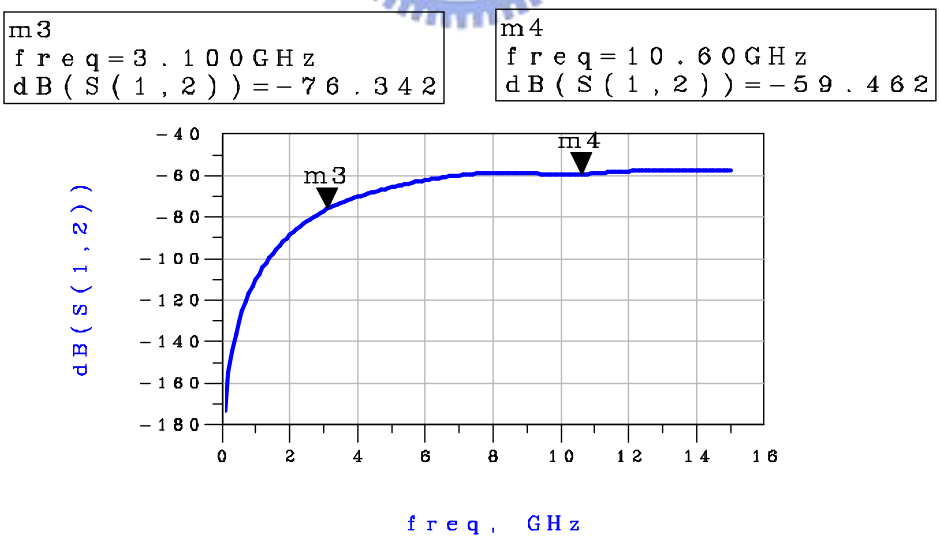


Fig. 4-11 Simulated S12

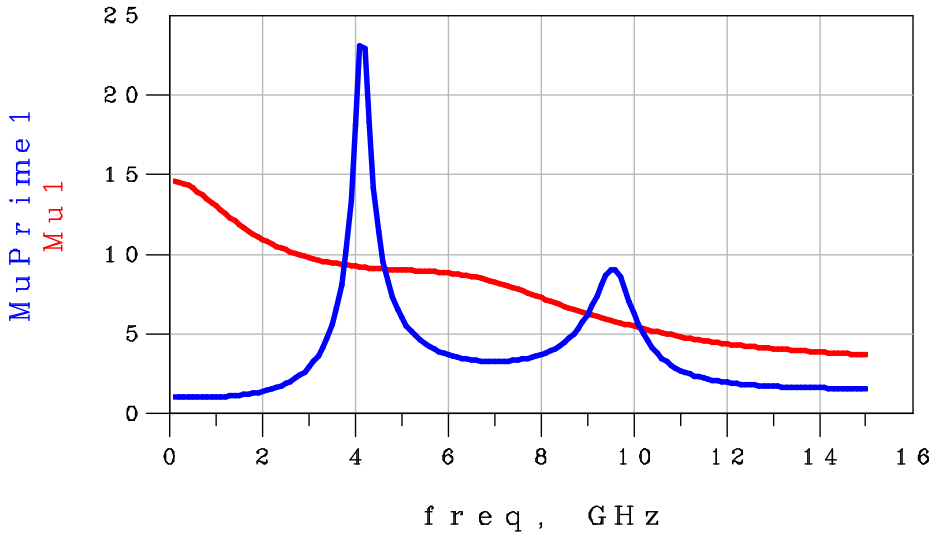


Fig. 4-12 Simulated stability

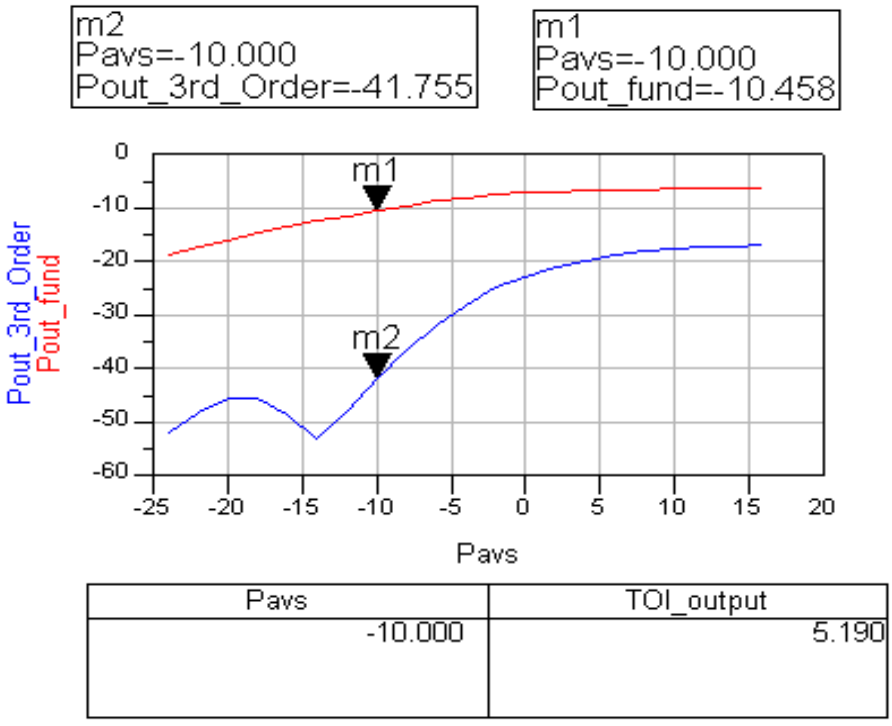


Fig. 4-13 Two tones test

4.4 Measurements and Conclusions

The following Fig.4-14 ~ Fig.4-19 are the measurement result which are slightly different from simulation. Which imply good accuracy of simulation and good circuit design. The some of the gain compression at high frequency showing in Figure 4-14 maybe due to the underestimate of the load resistor parasitic.

The bandwidth of this work with considering matching and gain is from 3.1 to 10.6 GHz, while the average gain is about 10dB. Fig. 4-16 shows the measurement result of S11 and the Fig. 4-17 shows the measurement result of S22. Output matching is achieved well from 3.1 to 10.6 GHz. The average S11 is about -8dB and the S22 can bellow -9.6dB. Fig. 4-18 shows the measured noise figure. The noise performance is very flat and the minimum noise figure is 5.03dB at 7GHz. The noise figure can be better if we solve the resistor parasitic. Fig.4-20 shows the die photo of this circuit. Total power consumption is 17mw which the vg is 0.7V and vdd1 and vdd2 are 1.8v. Table 4.1 is the measurement result summary. By the capacitor-resistance feedback with series inductive peaking we proposed, a good input and output matching, broadband, a low power consumption amplifier is developed for UWB system applications.

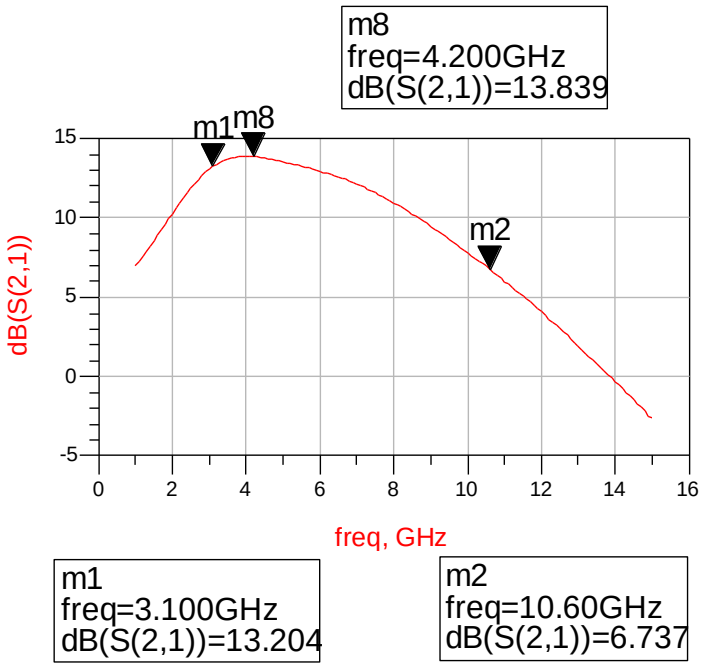


Fig. 4-14 Measured S21

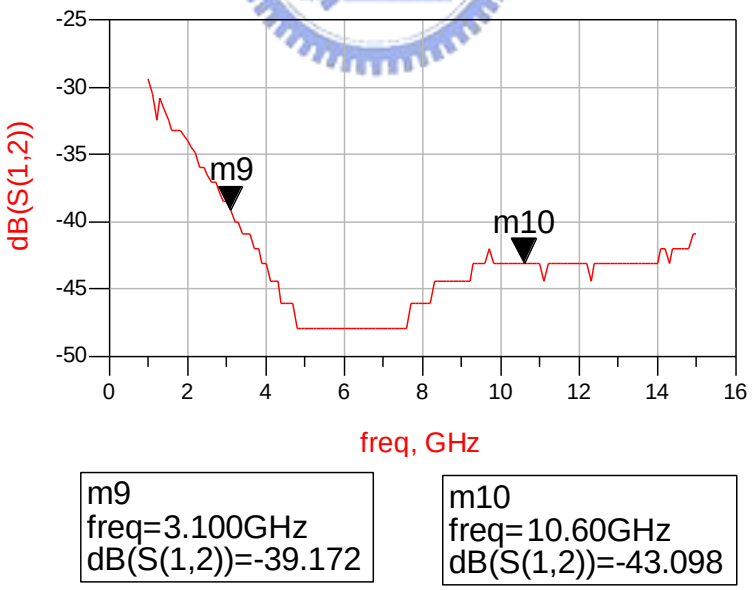


Fig. 4-15 Measured S12

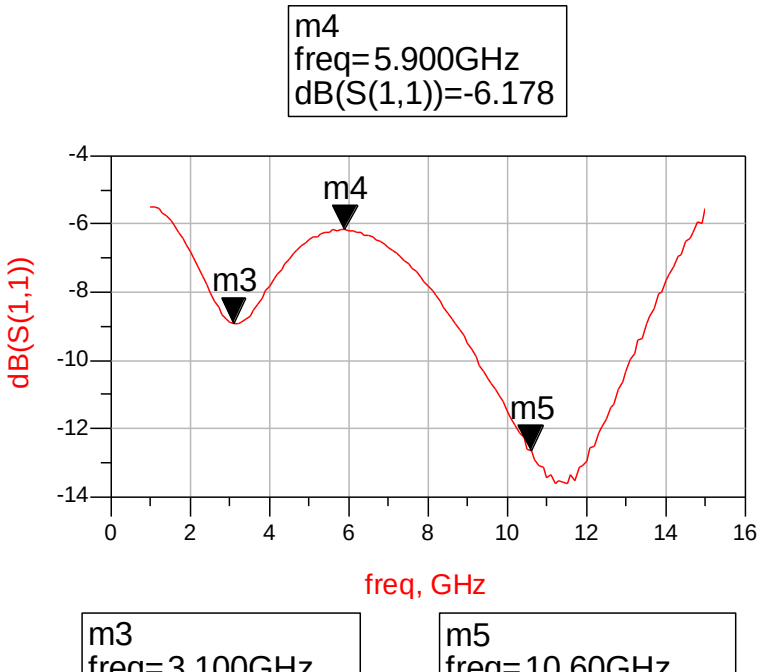


Fig. 4-16 Measured S11

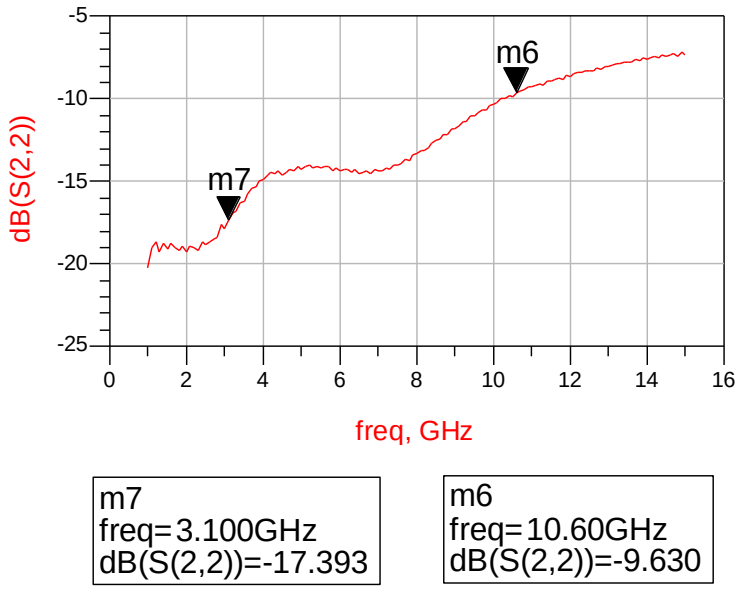


Fig. 4-17 Measured S22

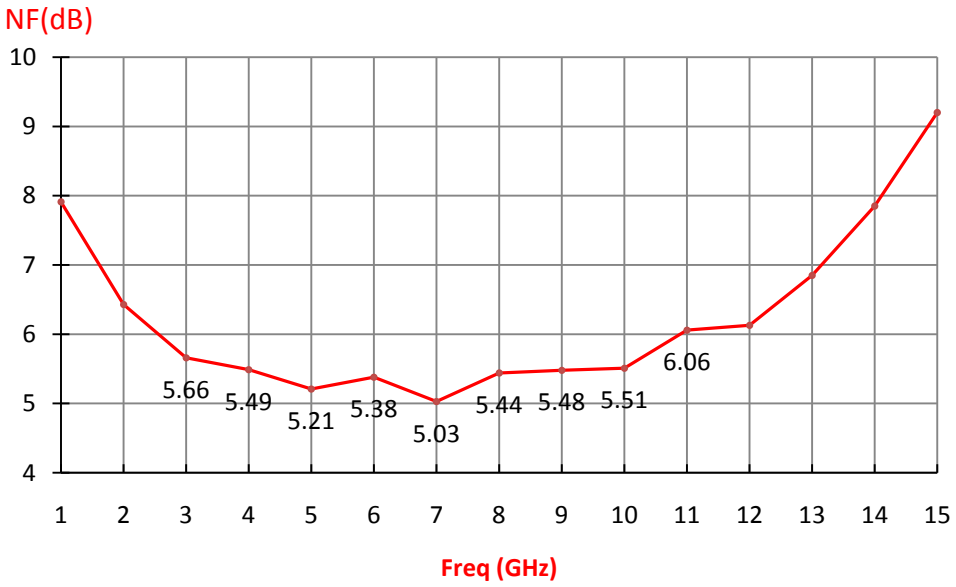


Fig. 4-18 Measured noise figure

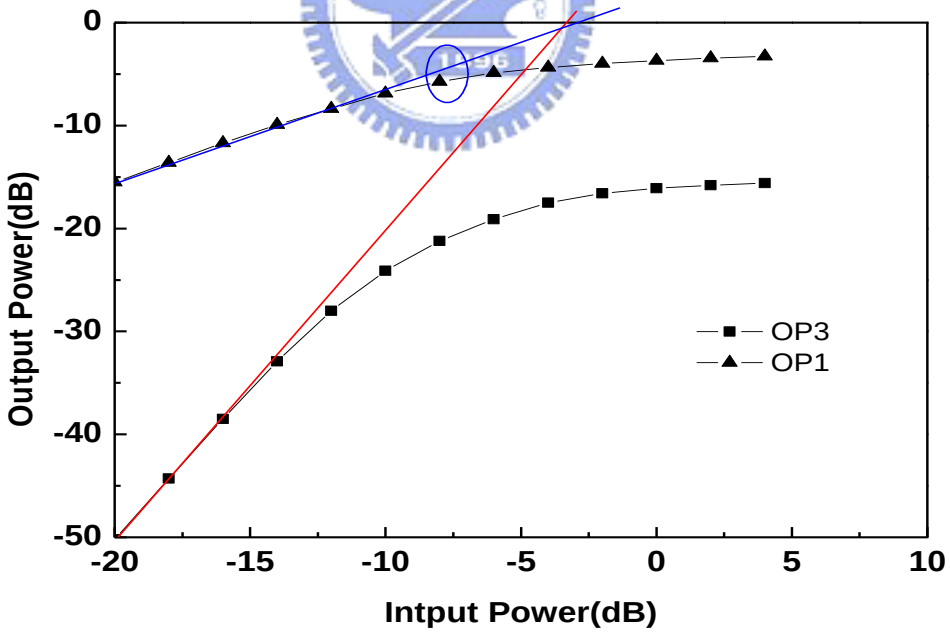


Fig. 4-19 Measured linearity

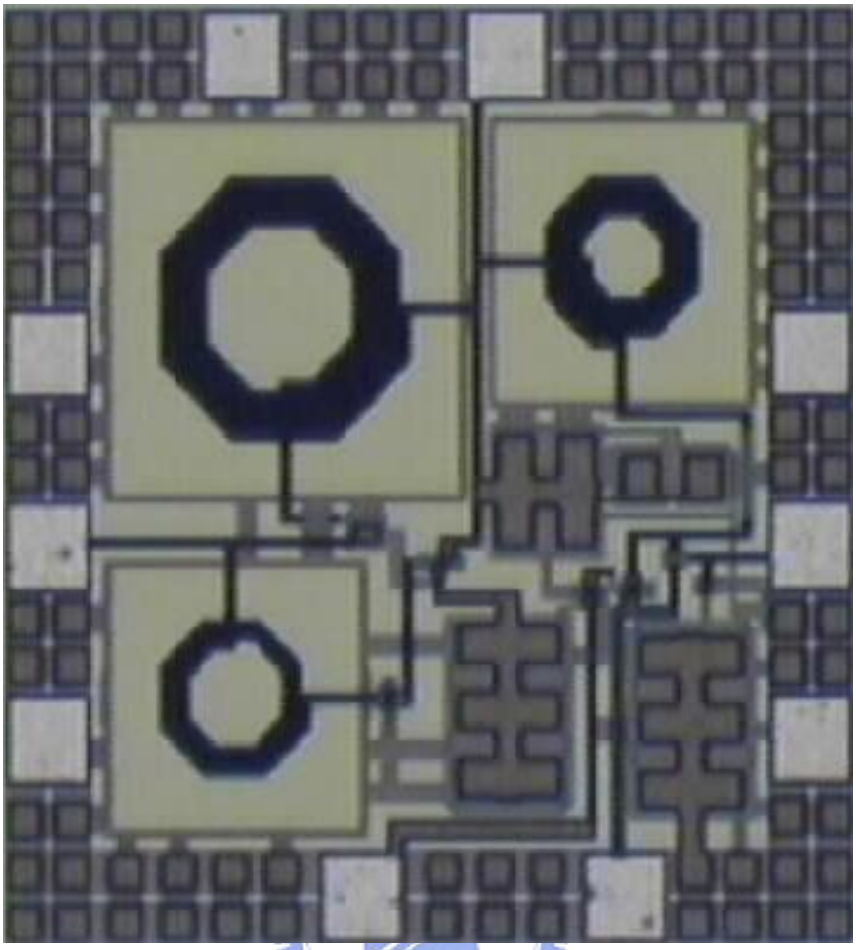


Fig.4-20 Die Photo

B.W. (GHz)	Gain (dB)	NF (dB)	S11 (dB)	S22 (dB)	IIP3 (dBm)	Pdc (mW)
3.1~10.6	6.73~13.20	5.03~5.66	-9.18~-12.65	-9.63~-17.39	-3	17

Table 4.1 Measured results summary

Chapter 5

Summary

By the capacitor-resistance feedback with series inductive peaking we proposed, a good input and output matching, broadband, a low power consumption amplifier is developed for UWB system applications.

Table 5.1 is the comparison of broadband LNA performance. We can find out by this table, by using R-C feedback with series inductive peaking technology, can pull to being wide very big arrival frequently. All the advantages are important for UWB system considerations.

Ref.	**B.W. (GHz)**	**Gain (dB)**	**NF (dB)**	**S11 (dB)**	**S22 (dB)**	**IIP3 (dBm)**	**Pdc (mW)**	**Tech.**	**year**
[13]	2.4~9.5	9.3	4~9	<-9	<-20	6.7	9*	.18 CMOS	2004
[14]	0.6~22	8.1	4.3~6	<-8	<-9	NA	52	.18 CMOS	2003
This work	3.1~10.6	10	5.03~5.6	<-7	<-9	-3	17	.18 CMOS	2006

Table 5.1 Comparison of broadband LNA performance　　* LNA core only

Reference

[1] A. Rofougaran, G. Chang, J. Rael, et al. “A single –chip 900MHz spread spectrum wireless transceiver in 1mm CMOS-part I: architecture and transmitter design.” IEEE J. Solid State Circuits, vol. 33, pp.513-534, April 1998.

[2] J. Rudell, et al.,“A 1.9GHz wide band IF double conversion CMOS receiver for cordless telephone applications” IEEE J. Solid-state Circuits, vol.32, pp.2071-2088, Dec.1997.

[3] P. Orsatti, F. Piazza, Q. Huang, and T. Mosrimoto, “A 20 mA receive 55 mA transmit GSM transceiver in 0.25-mm CMOS,” in In Int. Solid-State Circuits Conf. Dig. Tech. Papers.(San Francisco), pp. 232-233,Feb. 1999.

[4] C.Yoo and Q.Huang, “A common-gate switched,0.9W class E power with 41% PAE in 0.2μm CMOS.” In 2000 Symposium on VLSI circuits,(Honolulu, HI),pp.56-57, June 2000.

[5] P. Miliozzi, K. Kundert , K. Lampaert , P. Good, and M. chian, “A design system for RFIC: Challenges and solutions.” Proceedings of the IEEE, Oct.2000.

[6] B. Razavi, *RF Microelectronics*, 1st ed. NJ, USA: Prentice-Hall PTR, 1998.

[7] John Rogers, Calvin Plett, *Radio frequency integrated circuit design*. Boston :Artech House,c2003.

[8] T. H. Lee, *The Design of CMOS Radio-Frequency Integrated Circuits*, 1st ed. New York: Cambridge Univ. Press, 1998.

[9] G. Gonzalez, *Microwave Transistor Amplifiers Analysis and Design*, 2nd ed. NJ: Prentice-Hall, Inc. 1997.

[10] D. K. Shaeffer and T. H. Lee, "A 1.5-V, 1.5-GH_Z CMOS Low Noise Amplifier," *IEEE J. Solid-State Circuits*, vol. 32, no. 5, pp; 745-759, May, 1997.

[11] C-W. Kim, M-S. Kang, P. T. Anh, H-T. Kim and S-G. Lee, "An Ultra-Wideband CMOS Low Noise Amplifier for 3-5-GH_Z UWB System," *IEEE J. Solid-State Circuits*, vol. 40, no. 2, February, 2005.

[12] S. Vishwakarma, S. Jung and Y. Joo, "Ultra Wideband CMOS Low Noise Amplifier with Active Input Matching," *IEEE Ultra Wideband Systems*, 2004. Joint with Conference on Ultrawideband Systems and Technologies. Joint UWBST & IWUWBS. 2004 International Workshop on 18-21 May 2004, pp. 415-419.

[13] A. Bevilacqua and A. M. Niknejad, "An ultra-wideband CMOS LNA for 3.1 to 10.6 GHz wireless receiver," in IEEE ISSCC Dig. Tech. Papers, 2004, pp. 382 – 383.

[14] R.-C. Liu, K.-L. Deng, and H.Wang, "A 0.6 – 22 GHz broadband CMOS distributed amplifier," in Proc. IEEE Radio Frequency Integrated Circuits (RFIC) Symp., June 8 – 10, 2003, pp. 103 – 106.

Vita

姓名：陳懿範

性別：女

出生年月日：民國71年10月30日

籍貫：台北市

住址：台北市信義區吳興街284巷28弄36號2樓

學歷：國立交通大學電子工程學系

（90年9月~94年6月）

國立交通大學電子研究所固態電子組

（94年9月~96年6月）

論文題目：

應用於超寬頻3.1-10.6 GHz之無線接收端之低雜訊放大器之設計

An ultra-wideband CMOS LNA for 3.1 to 10.6 GHz wireless receivers