

Low Noise Preamplifier for THz Detector

Anyi Rao

(2014' , School of Electronic Science and Engineering, Nanjing University, Nanjing 210023)

Mentor: Lin Kang

(School of Electronic Science and Engineering, Nanjing University, professor, Nanjing 210093)

Abstract: Terahertz technology has important promising applications in the imaging radar, medical diagnosis, battlefield target identification, etc. However, due to the lack of high sensitive and accurate detector, terahertz technology's application is limited. A low noise amplifier is essential part to this problem. The goal of this paper is to design a low noise and accurate preamplifier. My goal is achieved by employing a differential input structure, which greatly improves the noise performance. The requirement of this design is controlling the preamplifier's noise performance less than $10\text{nV}/\sqrt{\text{Hz}}$ to satisfy THz detector Data acquisition circuit's resolution. Preamplifier's input signal has a 4kHz center frequency with 1kHz bandwidth, $100\mu\text{V}$ -1mV amplitude. Its output signal amplitude needs to be greater than 100mV.

Keywords: preamplifier; noise

1 Introduction

THz detector application's block diagram is shown in Figure 1. Preamplifier's design and optimization are needed. Considering the main source of noise is 50Hz power interference noise, active device noise and inactive devices[1] ($\frac{1}{f}$ noise, broadband white noise, etc.), there are two designs:

1.1 Design 1 Two Stage Band-pass Filter

Input stage is a Delyiannis Band-pass Filter[3], as shown in Figure 2.

Then write down the node voltage formulas of this circuit.

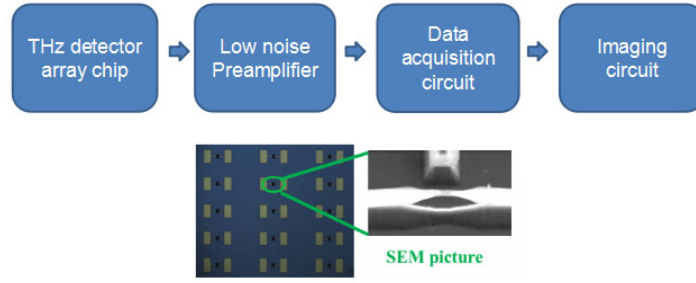


Figure 1: Block diagram of THz detector

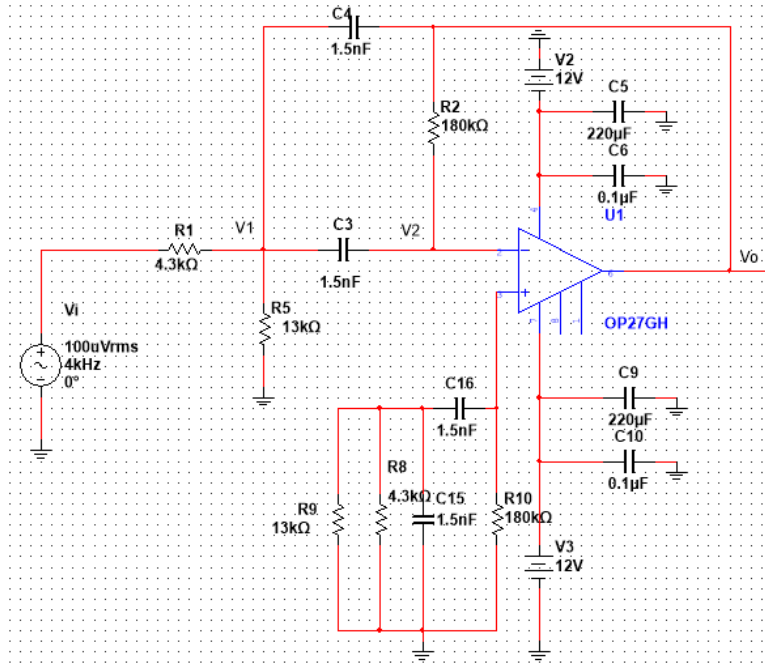


Figure 2: Input stage Delyiannis Band-pass Filter

$$\begin{cases} \left(\frac{1}{R_1} + \frac{1}{R_3} + sC_1 + sC_2 \right) V_1 - \frac{1}{R_1} V_i - sC_1 V_2 - sC_2 V_0 = 0 \\ \left(\frac{1}{R_2} + sC_2 \right) - \frac{1}{R_2} V_o - sC_1 V_1 = 0 \\ V_2 = 0 \end{cases}$$

Transfer function $H(s) = \frac{V_o}{V_i} = \frac{-s/R_1 C_2}{s^2 + s \left(\frac{1}{R_2} \left(\frac{1}{C_1} + \frac{1}{C_2} \right) \right) + \frac{1}{R_2 C_1 C_2} \left(\frac{1}{R_1} + \frac{1}{R_3} \right)}$

Stage II uses wide band band-pass filter, as shown in Figure 3.

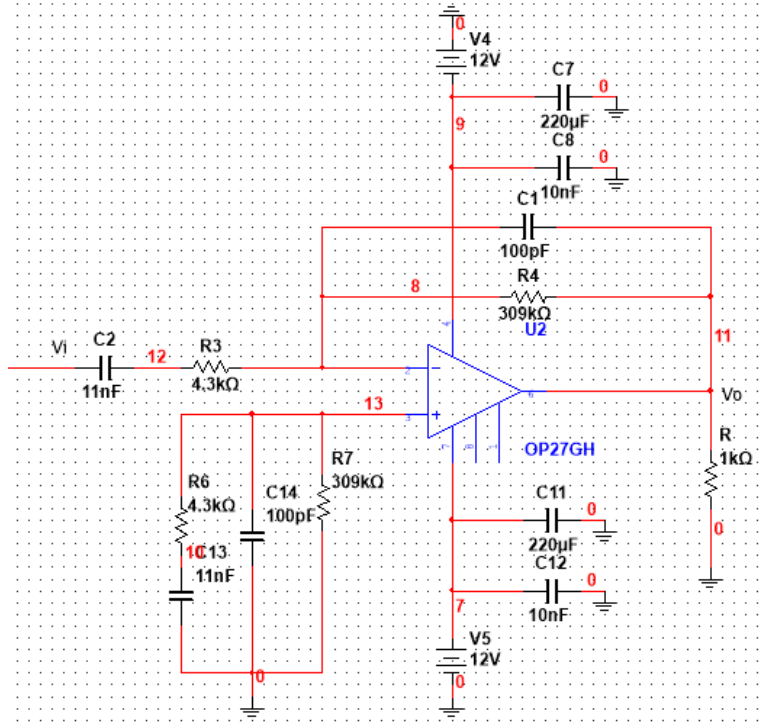


Figure 3: Wide band band-pass filter

Transfer function
$$H(s) = \frac{V_o}{V_i} = -\frac{R_4 \parallel \frac{1}{sC_1}}{\frac{1}{sC_2} + R_3}$$

1.2 Design 2 The Combination of Instrumental Amplifier and Band-pass Filter

Input stage is a differential input circuit, in which Op Amp 1 and Op Amp 2 configure the first level. Op Amp is the second level, as shown in Figure 4.

Transfer function
$$H = \frac{V_o}{V_{in1} - V_{in2}} = -\frac{R_5}{R_3} \left(1 + 2\frac{R_1}{R_g}\right)$$
 Stage II is the Delyiannis band-pass filter of feedback structure. The schematic diagram is shown in Figure 5.

1.3 Notices in the Process of Experiment

Selection of Op Amp chips: Considering the noise of chips and the cost of the experiment, I use OP27G comparing with other chips because of coefficient factor.¹

¹Because the experiment is studies the pros and cons of different method, the experiment is meaningful as long as the Op Amp chip doesn't loss the generality.

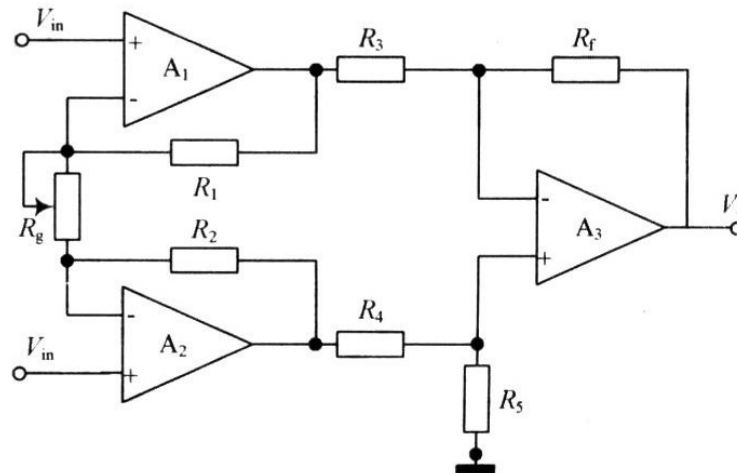


Figure 4: Differential input circuit

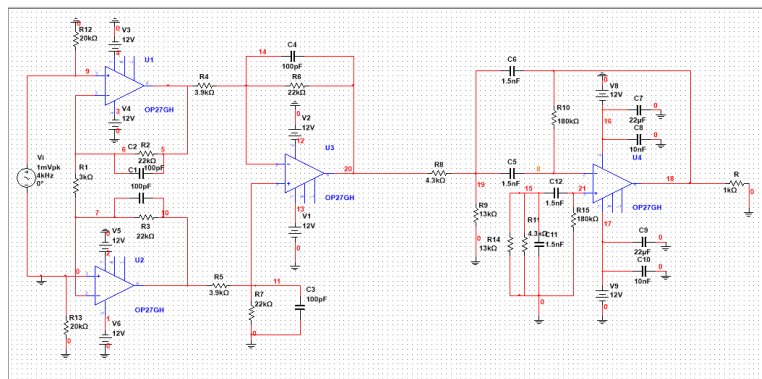


Figure 5: Design 2

Selection of resistors: Metal film resistor.

It is of small size, low noise, good stability, but the cost is higher, allowing the error is $\pm 0.5\%$.

Selection of capacitors: Ceramic capacitor and electrolytic capacitor.

Ceramic capacitor has a lower capacitance value generally, while the electrolytic capacitor's capacitance value is generally larger.

The symmetrical design: Due to the introduction of the negative feedback, Op Amp inverting terminal grounds directly. Therefore, it will lead the noise from land into the amplifying circuit. Obviously, this isn't what we expect. Accordingly it connects the resistors and

Item	Frequency	typ max
Input voltage noise e_n nV/ \sqrt{Hz}	f=10Hz	3.8 8.0
	f=30Hz	3.3 5.6
	f=1000Hz	3.2 4.5
Input current noise e_n nA/ \sqrt{Hz}	f=10Hz	1.7 –
	f=30Hz	1.0 –
	f=1000Hz	0.4 0.6
Gain of band width (GBW)	f	5.0 8.0

Table 1: Chip Parament

f_0 /kHz	BW/kHz	A_0
4.0	1	1500

Table 2: Theoretical Calculation of Design 1

capacitors which are the same as the in-phase terminal causing a symmetrical structure. The structure can inhibit the inverting and noninverting input common mode signal, then restrain the noise.

Op Amp power supply decoupling design: Because Op Amp's power supply comes directly from the power box, so the fluctuations of the amplitude of power voltage will affect Op Amp performance to a certain extent. Accordingly, it uses two small capacitors in parallel way to remove the interference of the high-frequency and low-frequency noise.²

2 Theoretical Calculation and Multisim Simulation

2.1 Design 1

The resistance capacitance values is shown in Figure 6. According to the theoretical formula, we get Table 2.

From Multisim Simulation, AC Analysis, I get Table 3.

²In addition, there are many points needed to pay attention, such as PCB layout drawing, the practical operation process, which will be described in later.

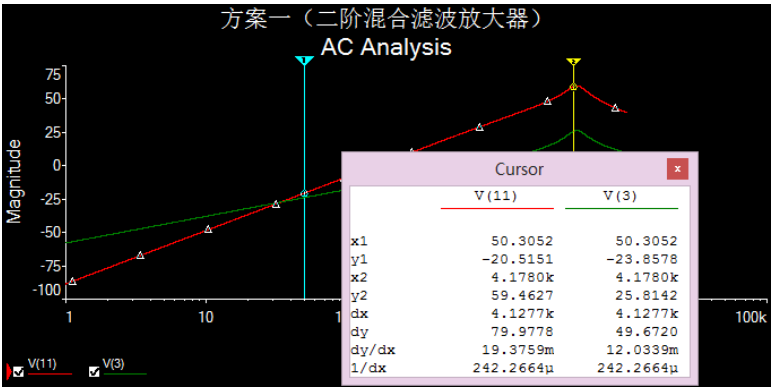


Figure 6: Instrumentation Amplifier

f_0/kHz	BW/kHz	A_0
4.0	1.02	1500.4

Table 3: Multisim Simulation Experiment of Design 1

From Figure 6, V_3 shows the amplitude frequency response of input stage (Delyiannis Filter, V_{11} shows the total amplitude frequency response. The restrain to 50Hz signal in the input reaches -79.9778dB, namely it reduces noise by 1×10^4 . This is exactly what we need.

2.2 Design 2

The value is shown in Figure 5, and Table 4 is got according to theoretical formula. Table 5 is got according to Multisim AC Analysis, Figure 7 V_{18} shows the amplitude frequency response of output in logarithmic representation The restrain to 50Hz signal in the the output is -50.1755dB, which is worse than design 1. But Design 2 uses differential input, which has high common mode rejection ratio. This property makes things better.As shown in Figure 8, I measure the common mode gain

f_0/kHz	BW/kHz	A_0
4.0	1	2000

Table 4: Theoretical Calculation of Design 2

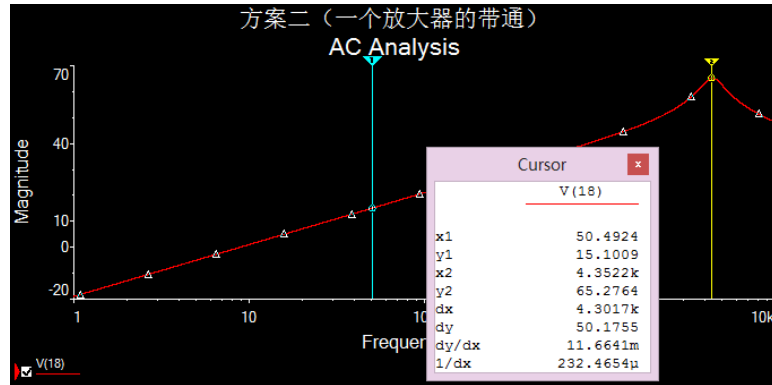


Figure 7: Differential input amplifier

f_0/kHz	BW/kHz	A_0
4.0	0.99	1997.5

Table 5: Multisim Simulation Experiment of Design 2

when I put the 1V 50Hz noise signal in both input terminal.

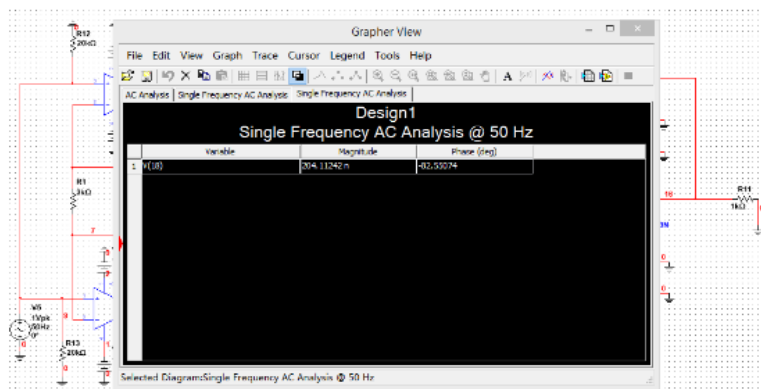


Figure 8: Common-mode Voltage Gain

And we get $V_o = V_{18} = 204.1 \text{ nV}$

$\therefore V_2 = V_5 = 1 \text{ V}$

$\therefore A_c = \frac{V_o}{V_i} = 2.041 \times 10^{-7}$

2.3 Theoretical Noise Analysis of Amplifier

From the device manual of OP27-G[4], as shown in Figure 9, OP27-G $\frac{1}{f}$ normalized voltage noise $e_{form}=1.2\text{nV}/\sqrt{\text{Hz}}$, wide width white voltage noise $e_{BB}=3\text{nV}/\sqrt{\text{Hz}}$, wide width white current noise $e_{inBB}=0.3\text{pA}/\sqrt{\text{Hz}}$

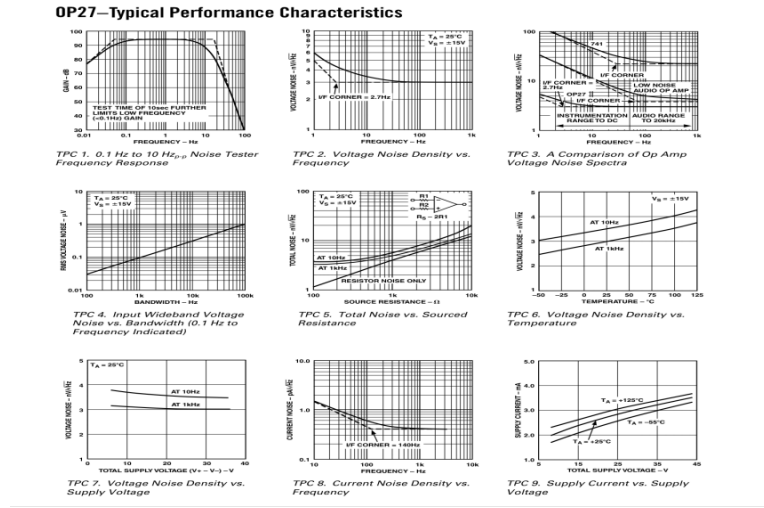


Figure 9: OP27-G Noise performance from device manual

2.3.1 Design 1

Configure the Thevenin equivalent circuit by using Multisim. The input resistance

$$\text{is } R_{eq} = \frac{U_x}{I_x} = \frac{U_i - I_x R_s}{I_x} = 1.468\text{k}\Omega$$

$$\text{Voltage noise } \frac{1}{f} \text{ noise: } e_f = e_{form} \sqrt{\ln \frac{f_H}{f_L}} = 3.7\text{nV}$$

wide width white voltage noise

$$e_{nBB} = e_{BB} \cdot \sqrt{BW_n} = 118.8\text{nV}$$

$$e_{n-v} = \sqrt{e_f^2 + e_{nBB}^2} = 118.9\text{nV}$$

Current noise Consider the wide width white current noise

$$e_{n-i} = R_{eq} \cdot e_{inBB} \cdot \sqrt{BW_n} = 17.5\text{nV}, \text{ and the } \frac{1}{f} \text{ noise of current is small enough to be ignored.}$$

Resistors thermal noise $e_{n-R} = \sqrt{4KTR_{eq}BW_n} = 0.197\mu V$

(where k is Boltzman constant, t is temperature in Kelvin's scale)

$$e_{n-RTL} = \sqrt{e_{n-v}^2 + e_{n-i}^2 + e_{n-R}^2} = 0.230\mu V$$

2.3.2 Design 2

The noise model of differential circuit is shown in Figure 10.

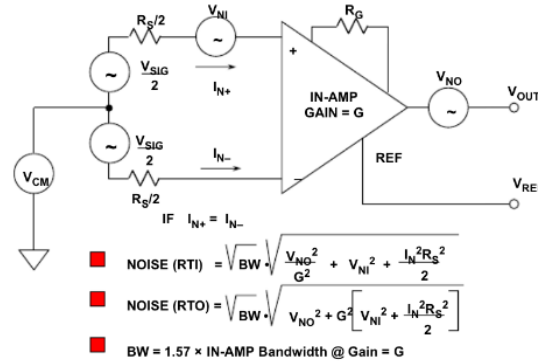


Figure 10: Differential Circuit Noise

$$\text{NOISE(RTL)} = 914.39\text{nV}$$

Delyiannis band pass filter is both used in design 1 and design 2.

$$e_{n-RTL} = \frac{N1G1G2 + N2G2}{G1G2} = N1 + \frac{N2}{G1} = 0.915\mu V$$

2.4 Deficiencies of Theoretical Noise Analysis

Theoretical analysis only considers the noise of Op Amp. There are lots of factors which are not included in, such as that the power line will conduct noise, ground plane will conduct noise, and the air radiation noise. Hence, the low Op Amp noise is advantageous to subsequent practical PCB circuit product and practical experiment. And this paper will continue to discuss the noise in the following practical experiment.

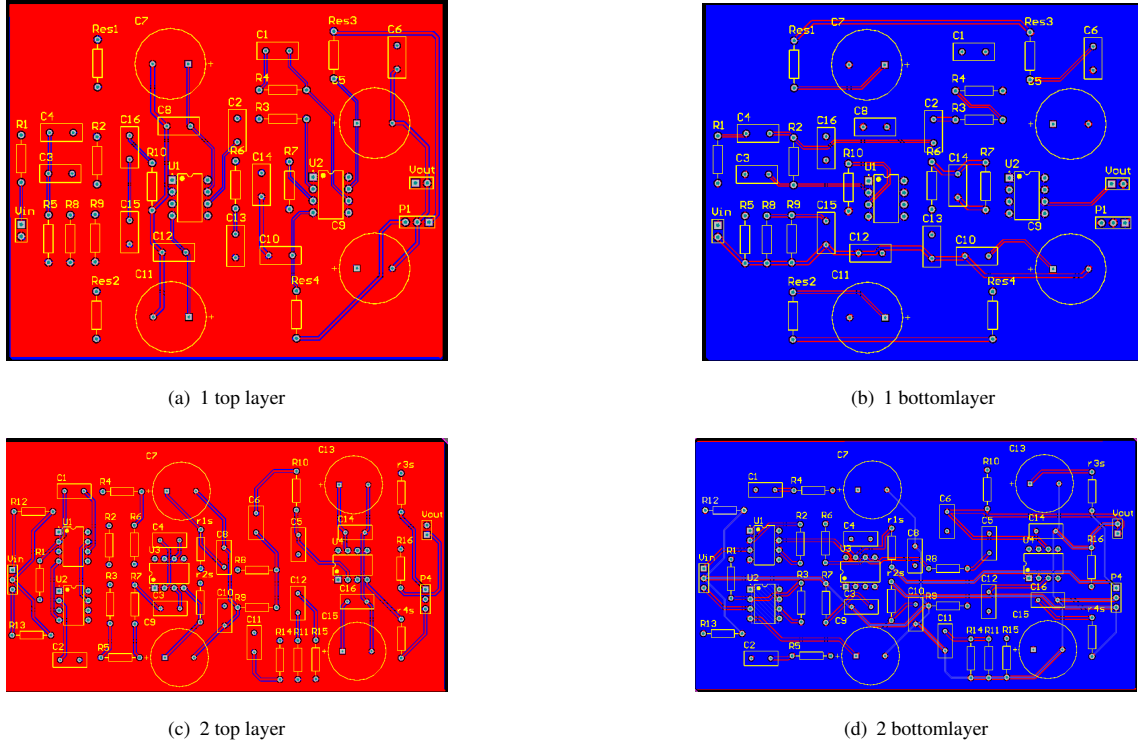


Figure 11: The PCB

3 Practical Experiment

Apparatus Oscilloscope Tektronix TDS1012B-SC, power GWINSTEKGPO-33030, signal source KEITHLEY3390, multimeter FLUKE8808A.

Temperature $T=25^{\circ}\text{C}$

PCB Picture The width of power line and ground line is 20mil, as shown in Figure 11. Especially the symmetry of design 2 is better than design 1, the wires between different layers is orthogonal.

After getting the actual PCB, weld the component and PCB according to the schematic diagram. Then measure the experimental data of design 1, as shown in Table 6.

Poly the data by MATLAB, as shown is Figure 12.

Taylor expand the amplitude frequency response between 3kHz-5kHz

$$f(x) = -0.2342 \times x^5 + 5.7029 \times x^4 + -53.3265 \times x^3 + 240.4312 \times x^2 - 524.0543 \times x + 443.8479$$

f/kHz	1.0	1.5	2.0	2.5	3.0	3.1	3.2	3.3	3.4	3.5
V_{p-p}/V	0.07	0.15	0.26	0.44	0.74	0.84	0.84	1.06	1.18	1.30
f/kHz	3.6	3.7	3.8	3.9	4.0	4.1	4.2	4.3	4.4	4.5
V_{p-p}/V	1.45	1.66	1.74	1.80	1.80	1.76	1.68	1.58	1.46	1.34
f/kHz	4.6	4.7	4.8	4.9	5.0	5.5	6.0	6.5	7.0	8.0
V_{p-p}/V	1.26	1.18	1.08	1.04	1.02	0.78	0.61	0.50	0.44	0.37

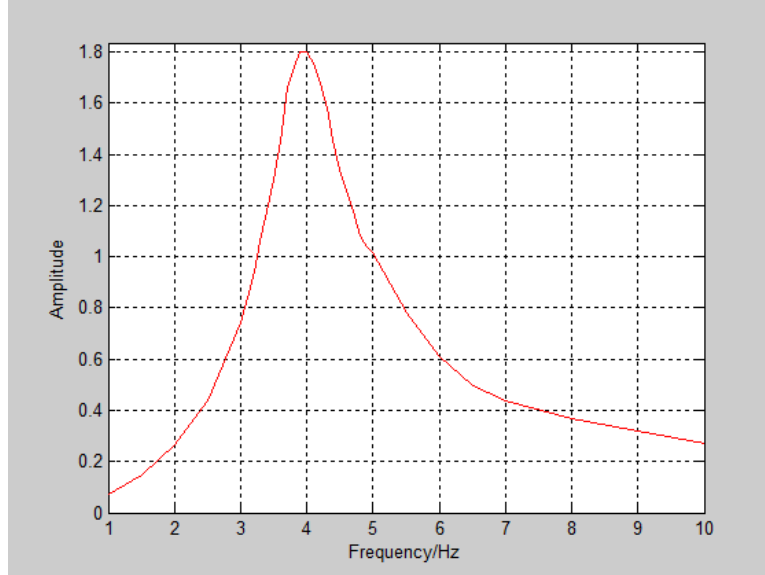
Table 6: Desing 1 Magnitude Response $V_{ip-p}=1.08mV$ 

Figure 12: Magnitude Response of Design 1

$$f'(x) = -(1171 \times x^4)/1000 + (57029 \times x^3)/2500 - (22515113018341197 \times x^2)/140737488355328 + (264356900080137 \times x)/549755813888 - 2304815185744049/4398046511104$$

Solve $f'(x)=0$

$$x=3.9712$$

It is coherent with the theory.

Then measure the circuit and get the experimental data of design 2, as shown in Table 7.

Poly the data by MATLAB, as shown is Figure 13.

Taylor expand the amplitude frequency response between 3kHz-5kHz.

$$f(x) = 1.7369 \times x^4 - 27.8764 \times x^3 + 164.5621 \times x^2 - 422.6665 \times x^1 + 400.0305$$

$$f'(x) = (17369 \times x^3)/2500 - (209073 \times x^2)/2500 + (1447503539529895 \times x)/4398046511104 -$$

f/kHz	1.0	1.5	2.0	2.5	3.0	3.1	3.2	3.3	3.4	3.5
V_{p-p}/V	0.21	0.38	0.48	0.71	1.10	1.22	1.31	1.44	1.58	2.08
f/kHz	3.6	3.7	3.8	3.9	4.0	4.1	4.2	4.3	4.4	4.5
V_{p-p}/V	2.32	2.56	2.76	2.88	2.96	2.92	2.84	2.682	.58	2.36
f/kHz	4.6	4.7	4.8	4.9	5.0	5.5	6.0	6.5	7.0	8.0
V_{p-p}/V	2.16	2.04	1.98	1.86	1.64	1.18	1.16	1.02	0.84	0.70

Table 7: Design 2 Magnitude Response $V_{ip-p}=1.45mV$

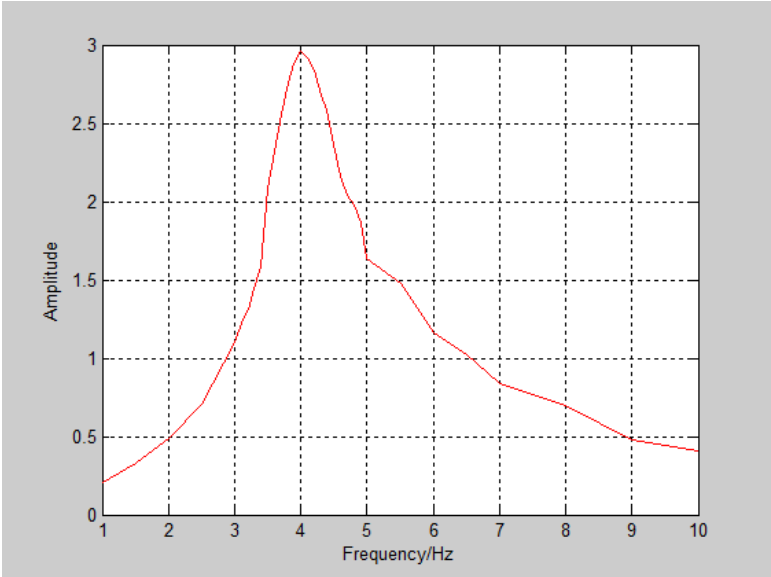


Figure 13: Magnitude Response of Design 2

7435627702742155/17592186044416

Solve $f'(x)=0$

$x=4.0632$

It is coherent with the theory.

f_0/kHz	V_{imax}	V_{omax}	A_c	K_{CMR}/dB	
50	1	0.718	7.18×10^4	2.04×10^3	129

Table 8: Common-mode Voltage Gain and Common-mode Rejection Ratio

Design	$V_{o-noise}/mV$	BW/kHz	A_o	A_e	$V_{i-noise}/\mu V$
1	1.302	1.3	1.67×10^3	20.8	62.6
2	0.212	1.3	2.04×10^3	27.6	7.68

Table 9: Noise Analysis

4 Noise Analysis

Complete the circuit and ensure it working normally. Then the input terminal is grounded, and the output voltage is measured by the multimeter FLUKE8808A, recording the maximum value of the output voltage.

Take advantage of $BW_{equip} \times A_e = BW \times A_o \times k$ and calculate the equivalent magnification A_e , where BW_{equip} is measureable bandwidth of FLUKE8808A $BW_{equip} = 150\text{kHz}$, BW is the bandwidth of the circuit, k is an empirical value, A_o is the center frequency of the amplification circuit[2].

Carrying out measurement to design 1 $V_{o-noise} = 1.302mV$

Carrying out measurement to design 2 $V_{o-noise} = 0.212mV$

It can be seen from the comparison as shown in Table 9, the equivalent noise of design 2 is better than design 1, because the design 2 uses the differential input mode, sacrificing two times the components than design 1 for the better effects.

Though the design 1, which sets filtering as its main target, is better in theoretical analysis and simulation, it is difficult to get rid of the practical noise from each circuit stage in practical experiment process. And it is easy to control the noise in the practical experiment because of its symmetry. Therefore, design 2 has more advantages.³

Additionally, When drawing PCB, design 2 is easy to be mirror-symmetrical because of the differential input mode. This enhances the effect of symmetrical design.

Design	theory	Multisim	experiment	error
1Amplitude Maxium value A_o	1.5×10^3	1.5004×10^3	1.67×10^3	11.3%
2Amplitude Maxium value A_o	2.0×10^3	1.9975×10^3	2.04×10^3	2.0%
$2K_{CMR}/\text{dB}$	—	199.69	129	35.4%

Table 10: Error Analysis

5 Error Analysis

In order to make the experiment more convincing, it introduces a small signal in the input terminal by using series connecting allotting circuit. This circuit will cause additional error to the experiment. Measurement is carried out, as shown in Table 10. The error of design 2 is smaller than design 1.

6 Conclusions

Deficiencies The empirical formula for noise analysis $BW_{equip} \times A_e = BW \times A_o \times k$ is not the best, though it is efficient. Because in the noise analysis, the most important thing is to get the equivalent input noise. Assuming the relationship between the voltage of input noise signal and the frequency can be depicted by function $f(x)$, the relationship between the magnification and the frequency can be depicted by function $g(x)$. After amplifier circuit's effect, the relationship between the voltage of output noise signal and the frequency can be depicted by the function $H(x)$. Therefore, the measured voltage of output noise by multimeter FLUKE8808A is $V_{o-noise} = \int h(x)dx = \int f(x) \cdot g(x)dx$. $V_{o-noise}$ and $g(x)$ is given, by calculating $V_{i-noise} = \int f(x)dx$, the accurate value will be got.

Expectation In agreement with other functions, the noise performance can be improved to two order of magnitude (100 times). Therefore, the differential input mode will have a

³Taking an overview of the two design, in order to get larger amplification and make the result more obvious, the experiment uses higher resistance values. This will introduce a greater resistance thermal noise, which is disadvantageous to the overall noise.

broad prospect, and it will be applied to the THz detector and it can be used in a practical way to a certain extent, so the THz technology will have further application.

References

- [1] 康华光.电子技术基础模拟部分[M].北京:高等教育出版社,2008.
- [2] 倪乃琛,沈保罗.噪声和电声测试技术[M].上海:同济大学出版社,1989.
- [3] LOW-NOISE HIGH-SPEED PRECISION OPERATIONAL AMPLIFIERS,SLOS100E FEBRUARY 1989.
- [4] Device manual of OP27-G, 2010.

针对太赫兹检测器的低噪声放大器研究

饶安逸

(南京大学电子科学与工程学院电子系2014级, 南京 210023)

指导老师: 康琳

(南京大学电子科学与工程学院, 教授, 南京 210093)

摘要: 太赫兹技术, 在雷达成像、医疗诊断、战场目标识别等领域具有重要的应用前景。然而, 由于高灵敏检测器等关键技术的瓶颈限制了太赫兹技术的实际应用。由于检测器的噪声性能小于 $10\text{nV}/\sqrt{\text{Hz}}$, 为使检测器阵列获得实际应用, 需要开发配套的低噪声放大电路。本文通过研究发现相对于传统方法, 使用差分输入方法可以大大提高噪声性能。其中太赫兹检测器输出信号为中心频率为4kHz、带宽1kHz、幅度 $100\mu\text{V}$ -1mV的小信号, 输出阻抗为 $\text{k}\Omega$ 量级, 后级采集电路的输入信号幅度至少需要100mV。

关键字: 前置噪声放大; 噪声