Analog Circuits Handbook
模拟电路手册

Yi Ding

(University of Chinese Academy of Sciences, Beijing 100049, China)

丁毅

(中国科学院大学，北京 100049)

2024.11 – ...

序言

Preface

Contents

Chapter 1 Measurement Methods

1.1 General Measurement Methods for Impedance

Relevant recources:

(1) Digilent Reference: Using the Impedance Analyzer
(https://digilent.com/reference/test-and-measurement/guides/waveforms-impedance-analyzer)

(2) 3 Ways to Measure Inductance (https://www.wikihow.com/Measure-Inductance)

(3) Inductance Measurement Using an LCR Meter and a Current Transformer Interface
(https://www.nist.gov/system/files/documents/calibrations/WaltripProceedings.pdf)

1.2 Inductor DCR (Direct Current Resistance) Measurement

DCR, standing for direct current resistance, just as its name implies, is the resistance of the inductor when a DC current flows through it. It is different from the equivalent series resistance (ESR) measured in the last section, which is the real part of the impedance of the inductor for AC signals. Concretely speaking, we use DCR to calculate the DC bias (large-signal) of a certain circuit, but use ESR to calculate the small-signal response.

1.3 High-precision Inductance Measurement

High-precision Measurement circuit is shown in Fig.1.1 and the inductor DCR (or ESR) does not affect the measurement result.

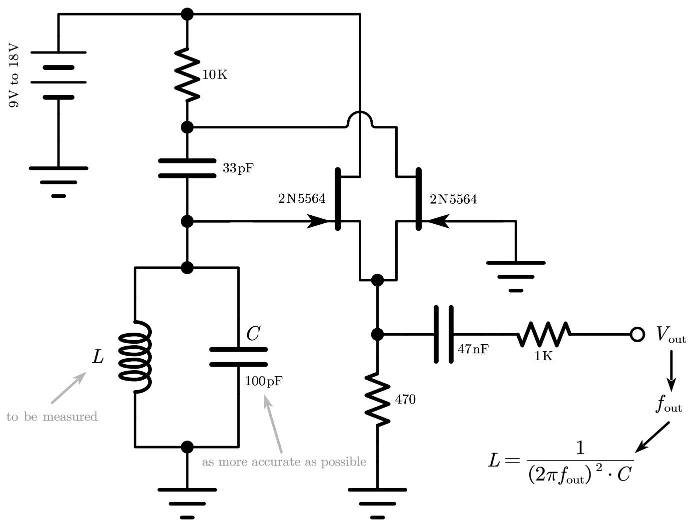


Figure 1.1: High-precision inductance measurement circuit

Below is an output frequency reference with $C = 100\text{pF}$ for different inductance range. Simply change the value of C to meet your requirements. For a precise measurement, high-precision capacitor is recommended.

Table 1.1: Output Frequency Reference for Different Inductance Range

L	1 nH	1 uH	1 mH	1 H	1000 H
f_{out}	503.29 MHz	15.915 MHz	503.29 KHz	15.915 KHz	503.29 Hz

1.4 Basic Current Sensing Circuit

Relevant recources:

(1) Current Sensing Circuit Concepts and Fundamentals (https://ww1.microchip.com/downloads/en/AppNotes/01332B.pdf)

(2) An Engineer's Guide to Current Sensing (https://www.ti.com/lit/eb/slyy154b/slyy154b.pdf?ts=1735114071672&ref_url=https%253A%252F%252Fcn.bing.com%252F)

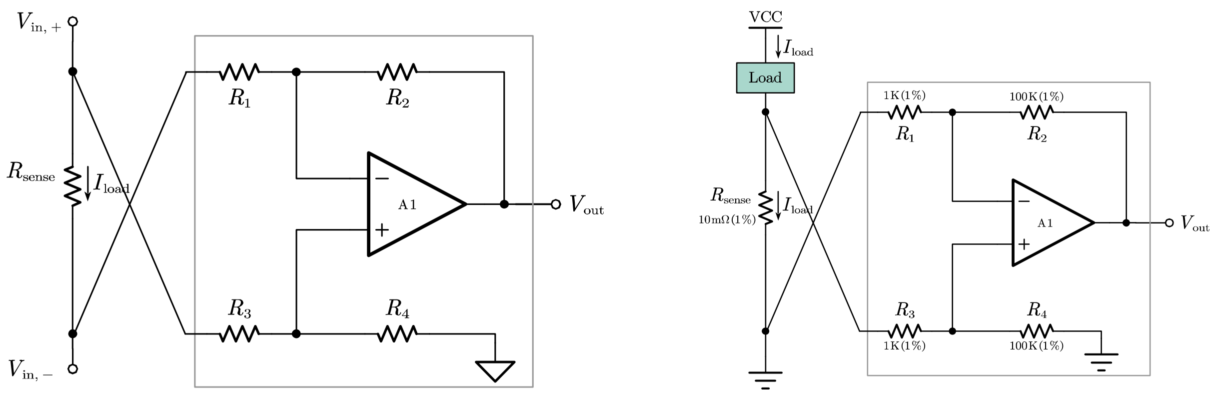
(3) Switch Mode Power Supply Current Sensing - Part 3: Current Sensing Methods (https://www.analog.com/media/en/technical-documentation/tech-articles/A59723-Part3-Switch-Mode-Power-Supply-Current-Sensing-Part-3-Current-Sensing-Methods.pdf)

(4) Understanding Current Sensing Applications & How to Choose the Right Device (https://www.ti.com/lit/ml/slap178/slap178.pdf?ts=1735143793758&ref_url=https%253A%252F%252Fcn.bing.com%252F)

(5) Current Sensing Techniques: A Review (https://ieeexplore.ieee.org/document/4797906)

1.4.1 General Considerations

Depicted in 1.2, we use a difference amplifier to sample the current. A general operational amplifier can be fine, but a precision one is recommended for better performance. We will see the error difference between the two in the following subsection.



(a) Simple current sensing schematic **(b) Typical application (low-side current sensing)**

Figure 1.2: A Simple current sensing circuit

To gain a better ouput performance at low load current, ± VCC is recommended for the power supply of the amplifiers. Remark that $R_1 = R_3$ and $R_2 = R_4$ are required for the difference amplifier. In this case, the output voltage is given by:

$$V_{\text{out}} = \frac{R_2}{R_1} \cdot I_{\text{load}} R_{\text{sense}} \Longrightarrow I_{\text{load}} = \frac{V_{\text{out}}}{R_{\text{sense}} \cdot \frac{R_2}{R_1}} \tag{1.1}$$

setting $R_1 = R_3 = 1\ \text{K}\Omega$, $R_2 = R_4 = 100\ \text{K}\Omega$ and $R_{\text{sense}} = 0.01\ \Omega$ yileds:

$$I_{\text{load}} = V_{\text{out}} \tag{1.2}$$

where the unit of I_{load} is ampere (A) and the unit of V_{out} is volt (V).

1.4.2 LTspice Simulation

Now we consider two different operational amplifiers, NE5532 and OP07, to run the simulation in LTspice, where NE5532 is a general operational amplifier and OP07 is a precision one. Here are the specific details of their parameters:

Table 1.2: Parameters of OP07 and NE5532

OPA	V_{io}	I_{io}	I_{b}	A_v	Slew Rate
OP07	60 uV	0.8 nA	1.8 nA	400 V/mV	0.3 V/us
NE5532	0.5 mV	10 nA	200 nA	200 V/mV	9 V/us

(1) V_{io}: input offset voltage;
(2) I_{io}: input offset current;
(3) I_{b}: bias current;
(4) A_v: large-signal open-loop gain;
(5) Slew Rate: the maximum rate of output voltage change.

Construct the simulation circuit in LTspice as shown in Fig.1.3 and run the DC sweep. We can see the output and error difference between the two operational amplifiers in Fig.1.4, where the relative error η is defined as:

$$\eta = \frac{V_{\text{out}} - I_{\text{load}}}{I_{\text{load}}} \times 100\,\% \tag{1.3}$$

Since OP07 has a low input offset and bias, the measurement error is negligible. By contrast, the error with NE5532 is a bit obvious. This difference becomes more significant with $I \in [0,\ 0.1A]$, shown in Fig.1.5.

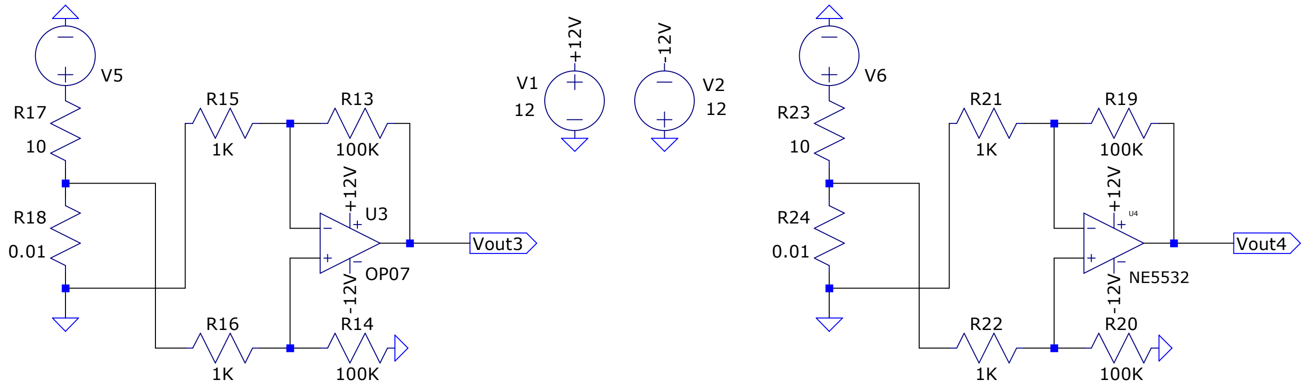
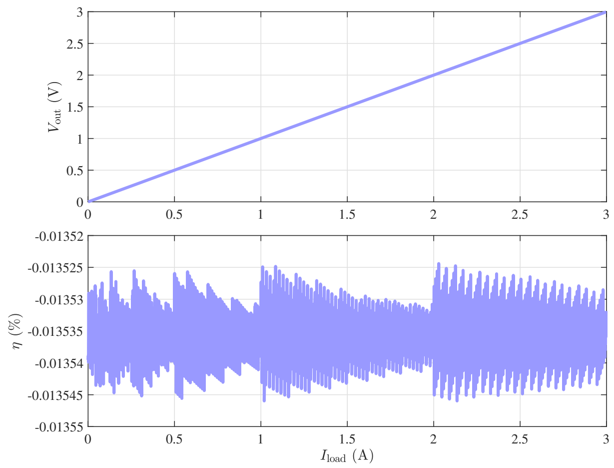
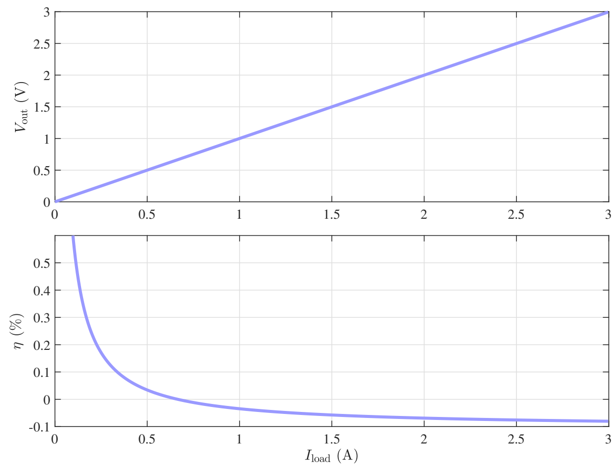


Figure 1.3: LTspice simulation of simple current sensing circuit

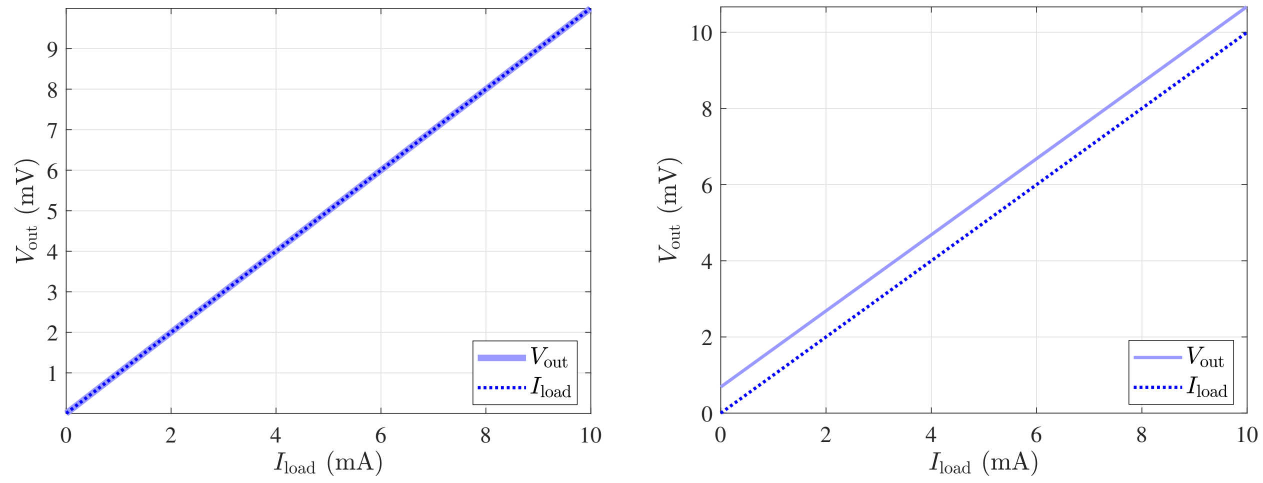


(a) Simulation results for OP07



(b) Simulation results for NE5532

Figure 1.4: Simulated ouput voltage and reletive error for $I_{\text{load}} \in$ [0A, 3A]



(a) Simulation results for OP07 (b) Simulation results for NE5532

Figure 1.5: Simulated ouput voltage and reletive error for $I_{\text{load}} \in [0\text{mA},\ 10\text{mA}]$

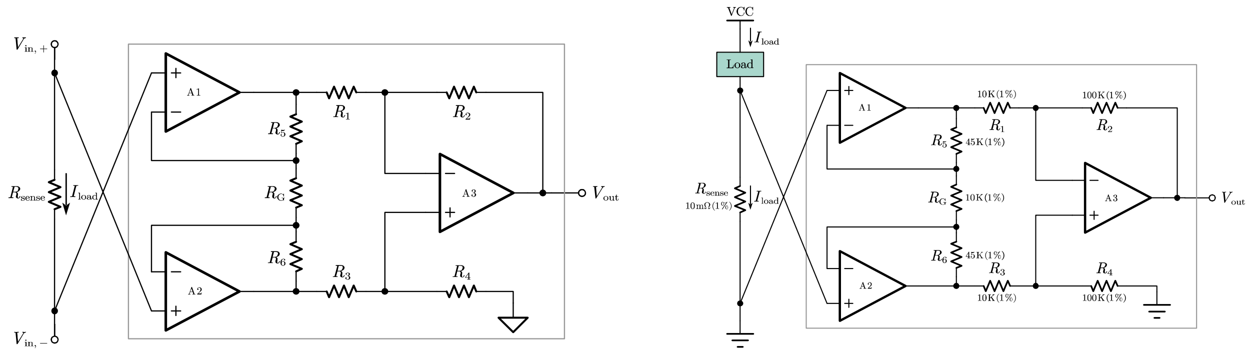
1.4.3 Actual Circuit Test

1.5 Precision Current Sensing Circuit (Ammeter)

> **Relevant recources:** same as in the previous section (Sec.1.4 Simple Current Sensing Circuit)

1.5.1 General Considerations

Depicted in Fig.1.6, we use an instrumentation amplifier comprised of three amplifiers to precisely sample the current of a load. $R_1 = R_3$, $R_2 = R_4$, $R_5 = R_6$ are necessary, and $R_G \in [0.2\ \text{K}\Omega,\ 2\ \text{K}\Omega]$, $R_{\text{sense}} \in [1\text{m}\Omega,\ 1\ \Omega]$ are recommended for better performance.



(a) Circuit schematic (b) Typical application

Figure 1.6: Precision current sensing circuit

The output voltage is given by:

$$V_{\text{out}} = \left(1 + \frac{2R_5}{R_G}\right)\frac{R_2}{R_1} \cdot I_{\text{load}} R_{\text{sense}} \Longrightarrow I_{\text{load}} = \frac{R_1}{R_2 R_{\text{sense}} \cdot \left(1 + \frac{2R_2}{R_1}\right)} \cdot V_{\text{out}} \tag{1.4}$$

1.5.2 LTspice Simulation

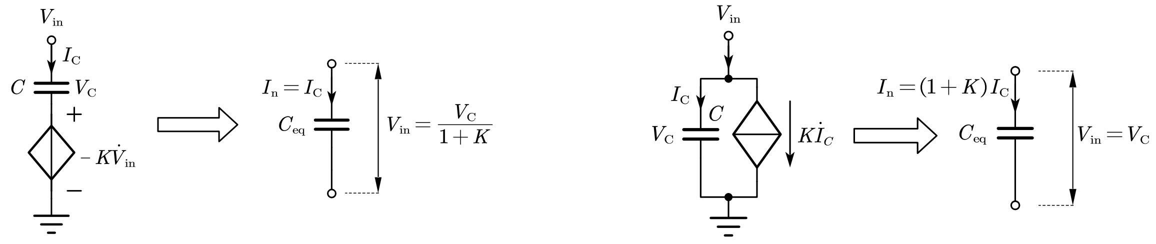
1.5.3 Actual Circuit Test

Chapter 2 Capacitors

2.1 Capacitance Multiplier

2.1.1 Basic Circuits and Principles

This part refers to references [?], [?] and [?]. Below are two basic concepts for capacitance multiplication:



(a) Voltage Mode **(b) Current Mode**

Figure 2.1: Basic Two Concepts for Capacitance Multiplier

Thus, we obtain the equivalent capacitance as:

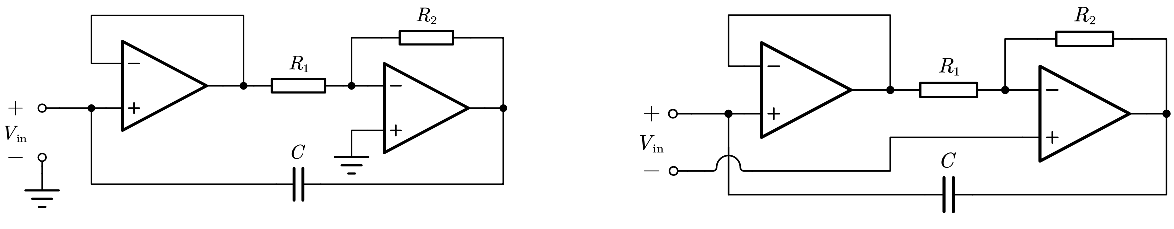
$$\text{voltage mode:} \quad C_{\text{eq}} = \frac{I_{\text{n}}}{SV_{\text{n}}} = \frac{I_C}{S\frac{V_C}{1+K}} = (1+K)\,C, \quad K > 0 \tag{2.1}$$

$$\text{current mode:} \quad C_{\text{eq}} = \frac{I_{\text{n}}}{SV_{\text{n}}} = \frac{(1+K)I_C}{SV_C} = (1+K)\,C, \quad K > 0 \tag{2.2}$$

A simple implementation of cap multiplier, depicted in Fig.2.2 (a), combining a unit-gain buffer (voltage fllower) and a inverting amplifier, uses a voltage mode. yielding the equivalent capacitance:

$$A = -\frac{R_2}{R_1} = -K \Longrightarrow C_{\text{eq}} = \left(1 + \frac{R_2}{R_1}\right) C \tag{2.3}$$

where $A = -\frac{R_2}{R_1}$ is the closed-loop gain of the inverting amplifier. Since inverting amplifier has a low input impedance, the unit-gain buffer is a necessary. To change it into a two-terminal element, just replace GND with the negtive terminal of the input voltage, e.g. $V_{\text{in},-}$, as shown in Fig.2.2 (b).



(a) One-terminal Cap Multiplier **(b) Two-terminal Cap Multiplier**

Figure 2.2: A Simple Implementation of Capacitance Multiplier

Refer to reference [?], [?] and [?] for more advanced circuits.

2.1.2 Multisim Simulation

Considering Figure 2.2 (b), set the parameters as Table 2.1. Then we connect it to the RC series circuit to perform a AC sweep to test the capacitance value. The Simulation Circuit is shown in Figure 2.3.

Table 2.1: Simulation Parameters of Capacitance Multiplier

C	R_1	R_2	Operation Amplifier
10 nF	1 KΩ	11 KΩ	LM258P

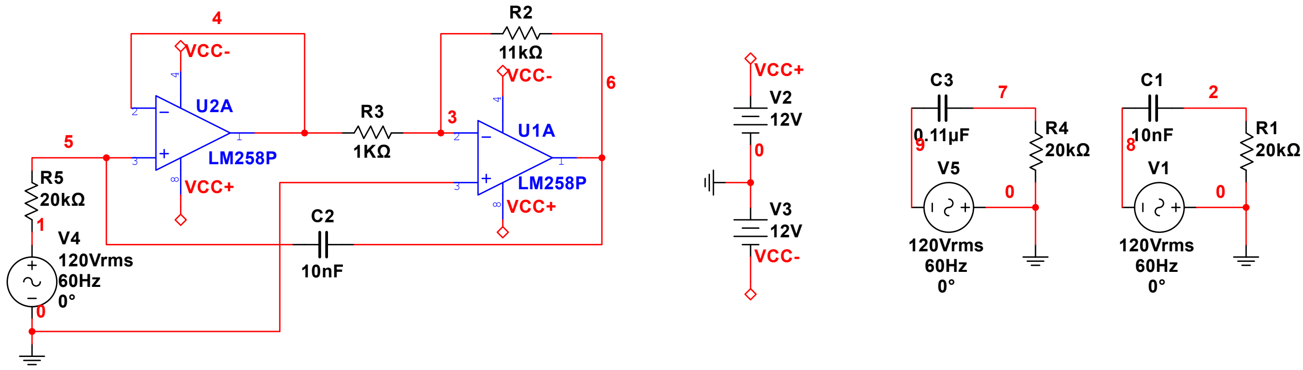


Figure 2.3: Simulation Circuit of Cap Multiplier

Export the simulation data and plot the frequency response (bode plot) of the series RC circuit, as shown in Figure 2.4. The theoretical value of the capacitance of the cap multiplier is 110 nF, and the simulation result confirmed this point.

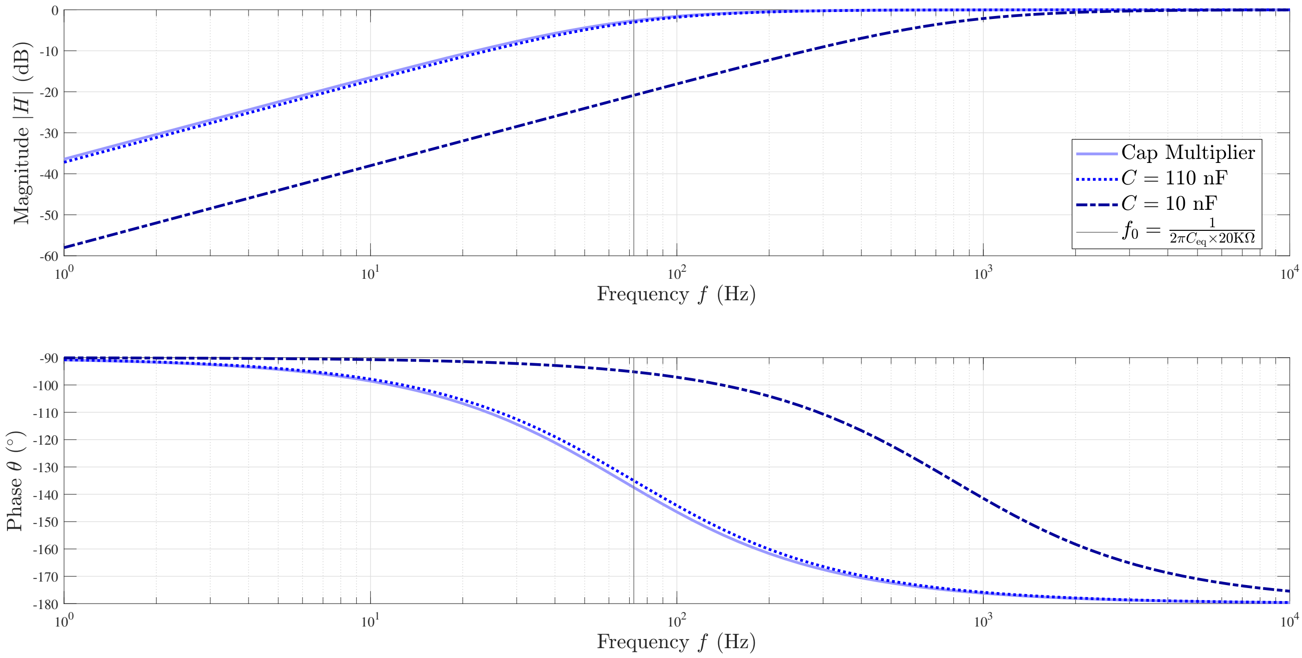


Figure 2.4: AC Sweep of the Cap Multiplier

Chapter 3 Inductors

3.1 Gyrator-Based Inductor

Relevant resources:

(1) Class AB Gyrator-Based Active Inductor (https://doi.org/10.1109/INMMIC.2015.7330365)

(2) Gyrator Based Inductor (https://www.academia.edu/4246523/gyrator_based_inductor)

Chapter 4 Oscillators

4.1 The Wien Bridge Oscillator

You must have seen that a number of resistors and capacitors can be connected together with an inverting amplifier to produce an oscillating circuit. Wien bridge oscillator is one of the simplest sine wave oscillators which uses an RC network in place of the conventional LC tuned tank circuit to produce a sinusoidal output waveform.

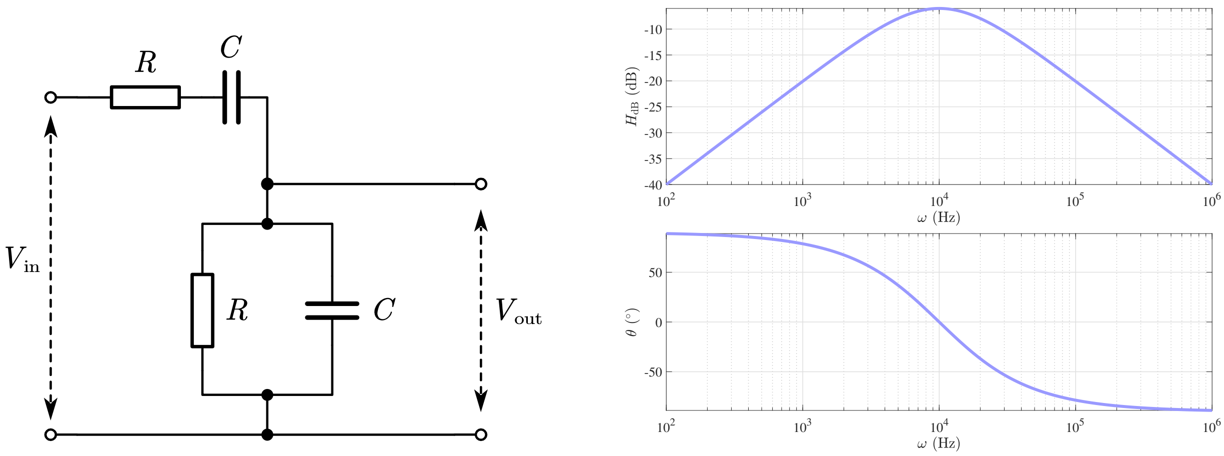
The Wien Bridge Oscillator is based on a noninverting amplifier, using a series RC circuit connected with a parallel RC of the same component values as a feedback circuit.

4.1.1 Basic Circuit and Principles

Consider the RC circuit in Figure 4.1 (a), the voltage gain H of the series RC circuit is:

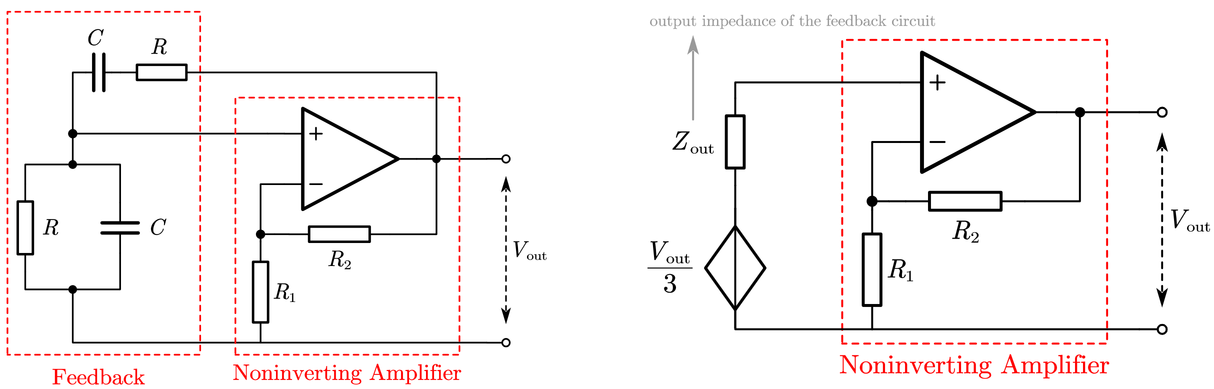
$$H(j\omega) = \frac{R \parallel (\frac{1}{j\omega C})}{R + \frac{1}{j\omega C} + R \parallel (\frac{1}{j\omega C})} = \frac{1}{1 + \frac{(R^2 - \frac{1}{\omega^2 C^2}) + \frac{2R}{j\omega C}}{\frac{R}{j\omega C}}}, \quad H|_{\omega=\frac{1}{RC}} = \frac{1}{1 + \frac{\frac{2R}{j\omega C}}{\frac{R}{j\omega C}}} = \frac{1}{3} \tag{4.1}$$

Defined to obtain a $0°$ phase shift, the resonant frequency f_0 is the key to Wien bridge oscillator. And at the point we have $H = \frac{1}{3}$. Let's set $R = 10\ \text{K}\Omega$, $C = 10$ nF and sketch the bode plot of this RC circuit in Figure 4.1 (b).



(a) RC Feedback Circuit **(b) Bode Plot of the RC Circuit**

Figure 4.1: Wien Bridge Oscillator's Feedback Circuit



(a) Wien Bridge Oscillator **(b) Equivalent Circuit of Wien Bridge Oscillator**

Figure 4.2: Wien Bridge Oscillator and Its Equivalent Circuit

Since a noninverting amplifier has extreme high input impedance and low output impedance, the coupling effect of the two circuits is negligible. In other words, the output impedance of noninverting amplifier (combing with the input impedance of feedback circuit), and the output impedance of feedback circuit (combing with input impedance of noninverting amplifier) can be ignored. Thus, the Wien bridge oscillator, depicted in Figure 4.2 (a), has the equivalent circuit shown in Figure 4.2 (b).

The oscillation frequency f_0 of the Wien Bridge Oscillator is given by:

$$\omega_0 = \frac{1}{RC}, \quad f_0 = \frac{1}{2\pi RC} \tag{4.2}$$

As the voltage gain of noninverting amplifier is:

$$A_v = 1 + \frac{R_2}{R_1} \tag{4.3}$$

yielding **the start-oscillation condition**:

$$A_v > 3 \Longleftrightarrow R_2 > 2R_1 \tag{4.4}$$

Assuming R_2 is slightly greater than $2R_1$, and there is a noise signal consists of a series of frequency, including $f_0 = \frac{1}{2\pi RC}$. Then at the selected resonant frequency f_0, $\frac{1}{3}A_v > 1$, so the positive feedback will cancel out the negative feedback signal, causing the circuit to oscillate, until it reaches a voltage saturation (dependent on power supply). However, at the other frequency, $\frac{1}{3}A_v < 1$ so the negative feedback will cancel out the positive, resulting other frequency signal fading away.

The closer the ratio $\frac{R_2}{R_1}$ is to 2^+, the better the waveform, but the longer the start-up time. Define $a = \frac{1}{3}\left(1 + \frac{R_2}{R_1}\right)$ as the periodic gain, a not-bad approximation for the start-up time is:

$$t_{\text{start}} = \frac{1}{f}\log_{a^3}\left(\frac{V_{\text{limit}}}{V_{\text{noise}}}\right) - 0.02 \tag{4.5}$$

where the unit of t is seconds, V_{limit} is the limit amplitude of output voltage, V_{noise} is the amplitude of noise.

By the way, if R_2 exceeds $2R_1$ too much, for example $R_2 = 3R_1$, the output waveform will be seriously distorted. Also, due to the slew rate limitations of operational amplifiers, frequencies above 1 MHz are unachievable without the use of special high frequency op-amps.

4.1.2 Multisim Simulation of the Basic Circuit

Set the parameters in Figure 4.2 (a) as below, and run the simulation.

Table 4.1: Simulation Parameters of Wien Bridge Oscillator

R	C	R_1	R_2	Operation Amplifier	VCC
10 KΩ	10 nF	10 KΩ	20.1 KΩ	LM258P	± 12 V

The start-up time is about 570 ms, shown in Figure 4.3.

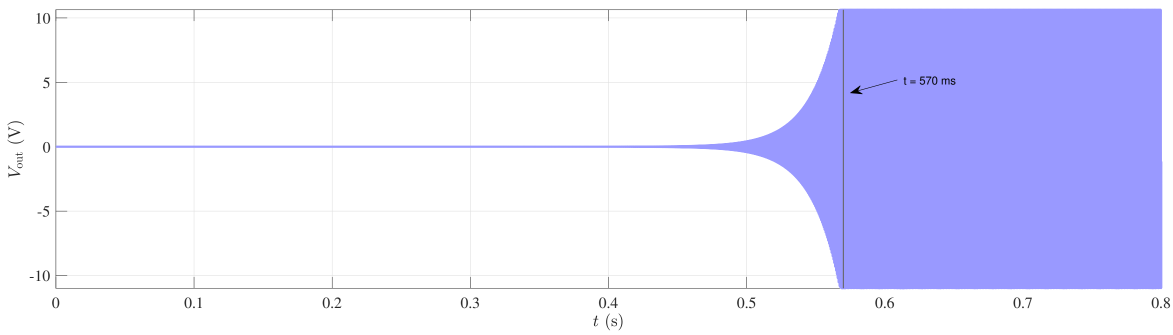


Figure 4.3: Start-up Time of Wien Bridge Oscillator

Export the simulation data, and perform a spectrum and distortion analysis in Matlab. Then we obtain the waveform and spectrum shown in Figure 4.4.

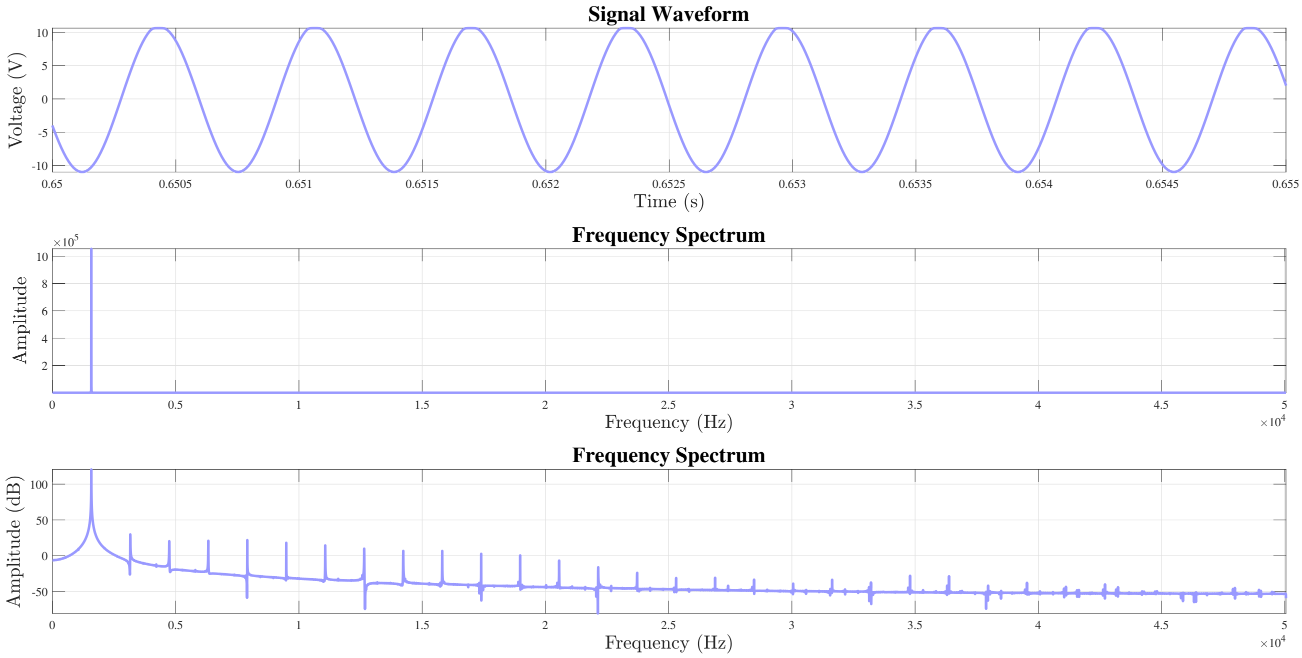


Figure 4.4: Spectrum Analysis of the Simulation Circuit

As we can see, the main output waveform is a sine wave at the resonant frequency f_0, the simulated oscillation frequency is:

$$
\begin{aligned}
f_{\text{simu}} &= 1.5758 \text{ KHz} \\
f_{\text{theo}} &= 1.5915 \text{ KHz} \\
\eta &= \frac{f_{\text{simu}} - f_{\text{theo}}}{f_{\text{theo}}} = -0.98\ \%
\end{aligned}
\tag{4.6}
$$

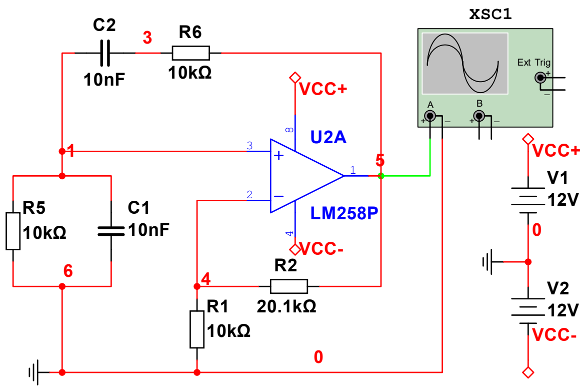
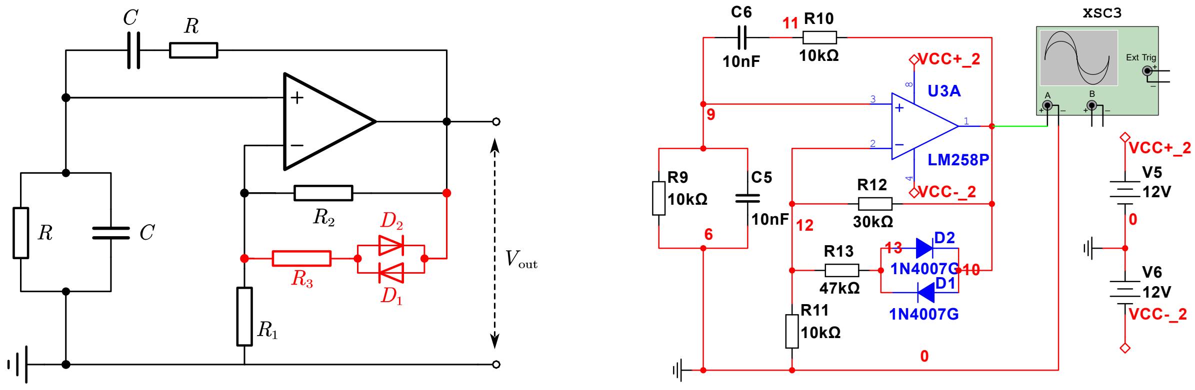


Figure 4.5: Simulation Circuit of Wien Bridge Oscillator

4.1.3 Decrease the Start-up Time

We have noted that the output waveform is distorted when R_2 exceeds $2R_1$ too much, but the start-up time is too long when R_2 is too closed to $2R_1$. Therefore, we need to optimize the circuit to achieve a better waveform and shorter start-up time, exemplified in Figure 4.6.

In Figure 4.6 (a), we added a resistance R_3 and two diodes D_1 and D_2. When the output voltage amplitude is less than the threshold voltage of diodes V_{D}, the diodes are off, and the circuit is the same as the basic circuit. When the output is greater than V_{D}, the diodes are on, and the resistance of R_3 is added to the circuit (parallel with R_2), which reduces the gain of amplifier.



(a) Optimized Circuit (b) Simulation Circuit

Figure 4.6: Optimize the Start-up Time of Wien Bridge Oscillator

Simulation circuit is shown in Figure 4.6 (b), the start-up time is reduced to about 10 ms (see Figure 4.7), and the output waveform is shown in Figure 4.8.

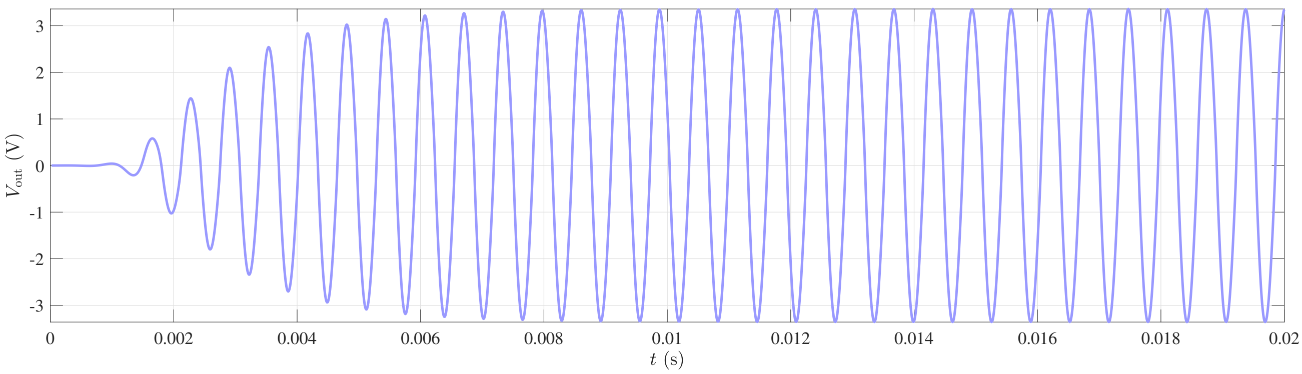


Figure 4.7: Optimized Start-up Time

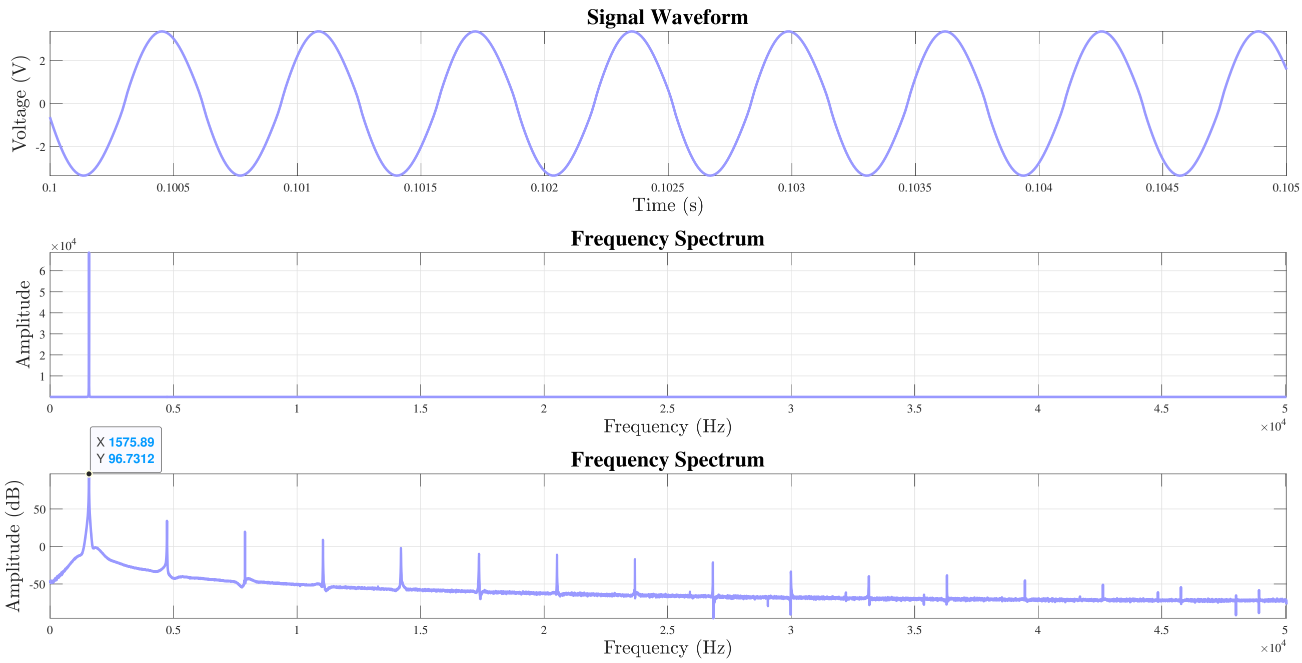


Figure 4.8: Spectrum Analysis of Optimized Circuit

Although the output frequency still focuses on f_0 and the distortion completely disappears, we have to note that the amplitude of the output waveform is significantly reduced. If larger amplitudes are desired, a resistor can be added to divide V_{out} to a suitable level, leading to a larger output amplitude (see reference [?] page 661). For more alternative methods to optimize the start-up time, see https://blog.csdn.net/qq_29356039/article/details/132611987.

4.1.4 Generate a Square Wave

An R_2 greater than $2R_1$ will result in the output waveform being clipped at the output voltage limitations. In other words, if we let $R_2 \gg 2R_1$, the waveform becomes a square wave. To prove our surmise, reset $R_1 = 1$ KΩ, $R_2 = 30$ KΩ in Table 4.1, without changing the other parameters. We obtain the output waveform shown in Figure 4.9.

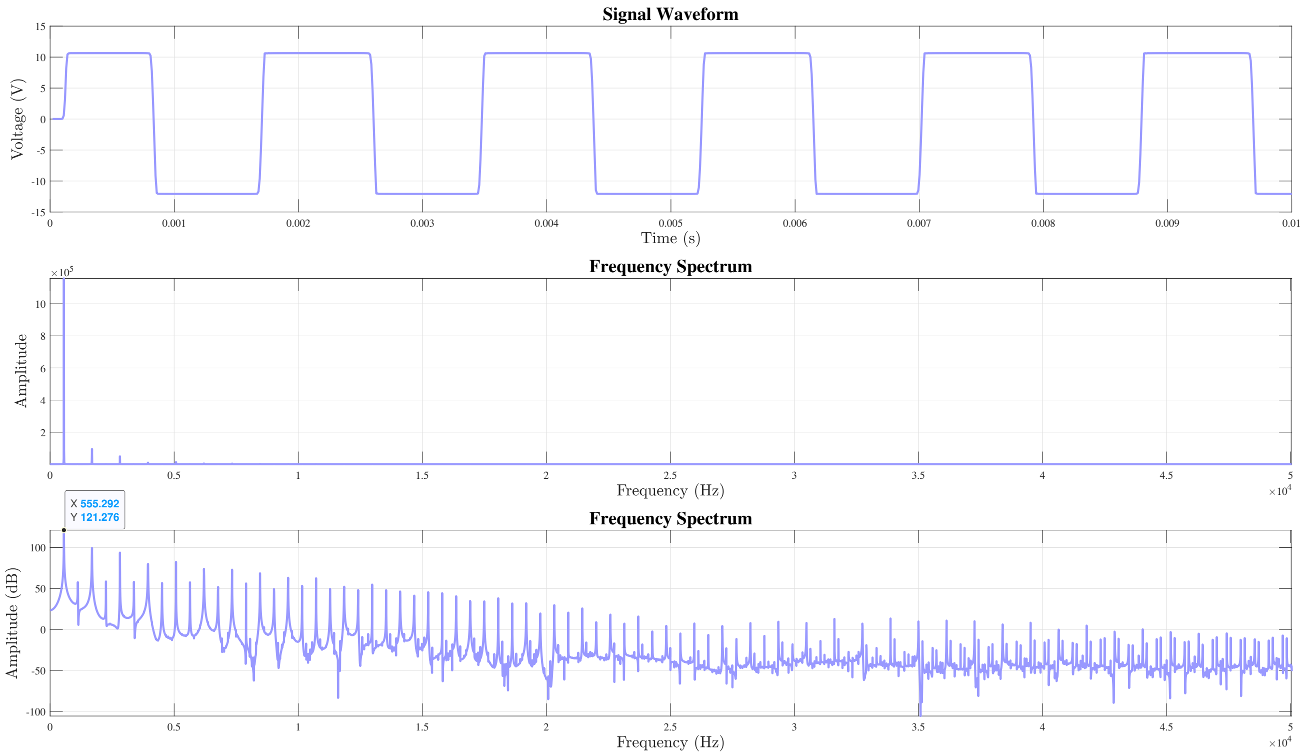


Figure 4.9: Square Wave Output of Wien Bridge Oscillator

Since the frequency of the square wave is difficult to calculate and control, the Wien Bridge Oscillator is not suitable for generating square waves in practical applications. To obtain an output signal with DC offset, see https://blog.csdn.net/qq_29356039/article/details/132611987 for more details.