UC San Diego, ECE 100

Lab Project 5: Nyquist plots of the op-amp circuits and the design of the Wien-Bridge Oscillator

Part 1: Nyquist plots of the op-amp circuits

In this part of the lab project, you will generate the Nyquist plots for the loop gain of the voltage follower and differentiator circuits by hand and using MATLAB and will find the phase margins in these circuits.

(a) **Unloaded voltage follower**

Draw the circuit schematic of the voltage follower circuit (Lab project 3). Use the op-amp open-loop gain equation, $A(s) = \frac{A_0}{1+s\tau_A}$, in which $A_0 = 10^5$ and $\tau_A = 0.1$. Assuming that the closed-loop DC gain is 10 V/V, write the equation of the loop gain. Draw the magnitude and phase Bode plots and the Nyquist plot of the loop gain by hand. Then, use MATLAB to draw the Bode and Nyquist plots of the loop gain. Read the **Notes** included in this document to learn more about how to do this.

(b) **Voltage follower circuit with a capacitive load**

Draw the circuit schematic of the **unity gain** voltage follower circuit with a capacitive load (Lab project 3). Assume $R_o = 50\ \Omega$ and $C = 200$ nF. Using the op-amp open-loop gain equation, $a(s) = \frac{a_0}{1+s\tau_A}$, in which $a_0 = 10^5$ and $\tau_A = 0.1$, write the equation of the loop gain. Draw the magnitude and phase Bode plots and the Nyquist plot of the loop gain by hand. Then, use MATLAB to draw the Bode and Nyquist plots of the loop gain. Include the original Nyquist plot and the zoomed-in version (with the phase margin data tip added to the figure) in your report.

(c) **Voltage follower circuit with a capacitive load and a compensation resistor**

Draw the circuit schematic of the **unity gain** voltage follower circuit with a capacitive load and a compensation resistor, R_C. Write the equation of the loop gain in this circuit. Find the value of R_C such that the zero of the compensated loop gain is at the cross-over frequency (unity-gain frequency) of the uncompensated loop gain. Assume $R_o = 50\ \Omega$ and $C = 200$ nF and $A(s) = \frac{a_0}{1+s\tau_A}$, in which $a_0 = 10^5$ and $\tau_A = 0.1$. Draw the magnitude and phase Bode plots and the Nyquist plot of the loop gain by hand. Then, use MATLAB to draw the Bode and Nyquist plots of the loop gain. Include the original Nyquist plot and the zoomed-in version with the phase margin data tip in your report.

(d) **Differentiator circuit with no compensation**

Draw the circuit schematic of the differentiator circuit (Lab project 4). Write the equation of the loop gain using the op-amp open-loop gain equation, $a(s) = \frac{a_0}{1+s\tau_A}$, in which $a_0 = 10^5$ and $\tau_A = 0.1$. Use $R = 100\text{k}\ \Omega$ and $C = 5$ nF in your equation. Draw the magnitude and phase Bode plots and the Nyquist plot of the loop gain by hand. Then, use MATLAB to draw the Bode and Nyquist plots of the loop gain. Include the original Nyquist plot and the zoomed-in version (with the phase margin data tip added to the figure) in your report.

(e) **Differentiator circuit with compensation resistor**

Draw the circuit schematic of the differentiator circuit compensated with a resistor, R_C, in series with the input capacitor (Lab project 4). Write the equation of the loop gain using the op-amp open-loop gain equation, $A(s) = \frac{A_0}{1+s\tau_A}$, in which $A_0 = 10^5$ and $\tau_A = 0.1$. In your equation, use $R = 100\ k\Omega$ and $C = 5$ nF. Find the value of R_C such that the zero of the compensated loop gain is at the cross-over frequency (unity-gain frequency) of the uncompensated loop gain. Draw the magnitude and phase Bode plots and the Nyquist plot of the loop gain by hand. Then, use MATLAB to draw the Bode and Nyquist plots of the loop gain. Include the original Nyquist plot and the zoomed-in version (with the phase margin data tip added to the figure) in your report.

Part 2: Design of the Wien-Bridge Oscillator

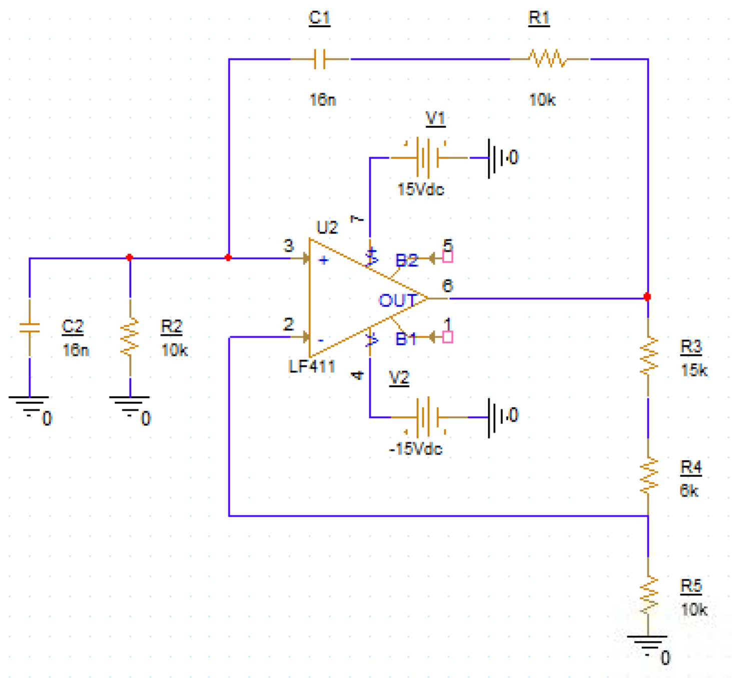
We have given a good deal of thought to preventing circuits from oscillating; but, of course, there are times when you want the circuit to oscillate. Such a circuit is the "Wien-Bridge Oscillator." It has been a popular oscillator in the audio frequency range, and it is of historical interest because it was the first product developed and sold by the Hewlett-Packard company!

Oscillators work in two steps:

a. **Gain Stage:** Noise is amplified by a gain factor K, determined by resistors connected to the output signal and the negative terminal of the op-amp.

b. **Filter:** The amplified noise gets filtered to allow only certain frequencies to pass, and the new signal gets fed back into the amplifier.

The basic oscillator is sketched below. It is based on an amplifier with a voltage gain $K = 3$, here shown implemented as a follower with gain. The feedback is through the RC network shown in the circuit diagram, where the two resistors R and the two capacitors C are the same.



System Level Design:

(a) Show analytically that the feedback factor is $B(s) = \frac{sRC}{1+3sRC+(sRC)^2}$. The loop gain will be $T(s) = -KB(s)$. Using MATLAB, plot the Nyquist plot for this loop gain. You will see a circular trace that goes right through the -1 point on the real axis twice, meaning there are two poles on the $j\omega$ axis. Include this Nyquist plot in your report. If $K > 3$, the poles are in the RHP; if $K < 3$, the poles are in the LHP. The resonant frequency of the oscillator is $\omega_0 = 1/RC$. To make a real oscillator, we need to consider two values of K: first, K has to be slightly greater than 3 so that the instability starts up the oscillating sinusoidal wave from noise; secondly, when the sinusoidal signal reaches the desired amplitude, we need to decrease K until it is exactly equal to 3 to prevent any further unstable growth of the sine wave.

(b) Simulate the oscillator in Simulink with a gain block of 3.1 and a feedback transfer function of $\frac{s}{1+3s+s^2}$. Include a saturation element and set the upper limit at 13v and the lower limit at -13 V. To get the oscillation to start in Simulink, add a small step of amplitude 0.1 V to the amplifier input. Plot the output on the scope. Include screenshots of your block diagram and the scope graph in your report. Read the **Notes** included in this document to learn more about how to do this.

Circuit Level Simulation:

(a) A nonlinearity can reduce the effective gain of the amplifier. For example, we can let the amplifier clip at the power-supply voltages. This will limit the output amplitude to about ± 13 V, and there will be noticeable clipping as the output is "flat" at points where we expect to see maxima and minima, as you will observe when you perform the following analysis in PSpice.

To observe the amplifier clipping the output, use the LF411 op-amp. Notice the resistors of the gain stage are a 15 $k\Omega$ and a 6 $k\Omega$ in series. This will give you $K = 3.1$, with poles in the RHP and thus unstable oscillation. For the filter components, choose $R = 10$ $k\Omega$ and $C = 16$ nF. This will give you a resonant frequency of 1 kHz.

You will need to run the transient simulation long enough that the oscillation builds up and saturates at the final amplitude.

- To get the oscillation started, in the time domain analysis settings in Pspice, select "Skip the initial transient bias point calculation." In LTspice, select "Skip the solution of the initial operating bias point."
- Sometimes Pspice and LTspice do not sample the waveform as finely as you would like. You can adjust the "Maximum Step Size" in the transient options. A max step size of 10us works well in this case.
- In Pspice simulations, you need to give the amplifier input voltage at the non-inverting input terminal a non-zero initial condition (say a few mV). To do so, you can specify the non-zero initial condition (voltage) on the capacitor connected from the input to ground by double-clicking on the capacitor and adjusting its properties. This step is not required for LTspice simulation.
- Run the transient response for about 100 ms. The oscillation should start from nothing and grow to full amplitude in about 30-40 ms.

Include pictures of your circuit diagram and the simulated growing oscillation in your report. Focus on a few cycles of the full amplitude oscillation and expand it to show the wave shape. You will see that it looks a bit flat on top. Clipping the wave this way prevents the oscillation from unstable growth but gives a distorted waveform. Include a picture of the expanded clipped waveform in your report.

(b) The performance of oscillators is traditionally judged by the frequency domain analysis. That is, we do a Fourier transform and compare the power in the fundamental frequency with the power in the harmonics. This can be done in PSpice using the Fourier option in the Trace tab or by clicking the FFT option in the menu bar. Modify the simulation profile to save the simulation data after 60 ms, then simulate the circuit again and select the FFT option. Next, set the y-axis of the FFT plot to log scale. Measure and record the amplitude of the 1 kHz spike and the amplitude of the 3 kHz spike; clearly mark these points on your FFT plot. Include a picture of your FFT plot in your report. The ratio of those (in dB) is an important measure of performance. Find the performance of this oscillator using

$$P = 20\log\left(\frac{V_{out}@1kHz}{V_{out}@3kHz}\right)$$

(c) Allowing the amplifier to saturate at the power supply voltage has a couple of disadvantages. It means that the amplitude depends on the power supply level, which also means that it is quite large. We would often prefer a smaller amplitude output to avoid slew rate limiting at high frequencies. We can obtain a smaller amplitude by clipping the voltage across the 6 kΩ resistor with parallel diodes. Use the 1N4148 diode. Put one in parallel with the resistor in one polarity and the other in parallel but with the opposite polarity. This will prevent the voltage across the resistor from exceeding about 0.7 V and thus limit the signal amplitude. What amplitude do you get with this modification? Expand a couple of cycles. Can you see any sign of clipping? Make a copy of your circuit diagram and of the simulated growing oscillation for your report. Re-measure and re-record the power ratio of the 1 kHz and 3 kHz harmonics (in dB). How much has it changed?

In fact, the original HP oscillator used a softer nonlinearity and obtained significantly lower distortion. They used a lamp filament which increases resistance as it is heated. Diodes have a much harder limiting characteristic.

(d) Try increasing the frequency by a factor of 10. Take a screenshot and ensure the frequency is clearly displayed for the report. Can you increase frequency by another factor of 10? You should not reduce the resistor R below 1 kΩ because it will draw too much current from the op-amp. However, you can change the capacitance C to compensate. What is the highest frequency you can reach? What seems to limit the maximum frequency?

Measurement:

(a) Build your oscillator and give it a test. It will start by itself because the circuit noise will be sufficient. Take a screenshot of the output waveform (or save the oscilloscope display) for your report. Include a picture of your circuit setup in your report.

(b) Add two parallel diodes across the 6 kΩ resistor. How does the output waveform change? Take a screenshot of the output waveform (or save the oscilloscope display) for your report. Include a picture of your circuit setup in your report.

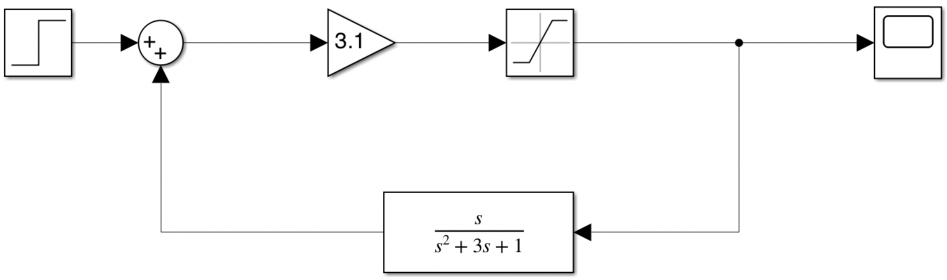
(c) What happens if you put the two parallel diodes across the 15 kΩ resistor instead of the 6 kΩ resistor? Take a screenshot of the output waveform (or save the oscilloscope display) for your report.

(d) Connect the diodes across the 6 kΩ resistor and try increasing the frequency by a factor of 10. Take a screenshot (or save the oscilloscope display) and make sure the frequency is clearly displayed for the report. Can you increase frequency by another factor of 10? Note, you should not reduce the resistor R below 1 kΩ because it will draw too much current from the op-amp. However, you can change the capacitance C to compensate. What is the highest frequency you are able to reach? Ensure the error in the expected frequency compared to the actual frequency is within 10%. What seems to limit the maximum frequency? What about lower frequencies? Can you get it down to 1 Hz?

Notes:

Note 1: In MATLAB, use the `nyquistplot` function to draw the Nyquist plot of the loop gain. Right-click on the plot, select "Characteristics," and then select "All Stability Margins." This will draw a circle with a radius of 1 centered at the origin. It can be used to show whether the Nyquist plot encircles the -1 point. You will need to zoom in on the graph to see the circle. Click on the intersection of the unit circle and the Nyquist plot to see the phase margin. Include the original Nyquist plot and the zoomed-in version (with the phase margin data tip added to the figure) in your report.

Note 2: Below is a picture of how the block diagram should be built in Simulink.



Use the following MATLAB Simulink tutorials to familiarize yourself with this package.

`https://youtu.be/iOmqgewj5XI`
`https://youtu.be/a_DW7xznPco`

Report:

Make sure to include the following in your report:

- Calculation of feedback factor (Part 1)
- Nyquist plot (Part 1)
- Circuit diagram and simulated growing oscillation, no diodes (Part 2a)
- Expanded clipped oscillation waveform (Part 2a)
- FFT plot with 1 kHz and 3 kHz points marked on plot (Part 2b)
- Circuit diagram and simulated growing oscillation, with diodes (Part 2c)
- Measurements

This list is not all-encompassing, check with your TAs, but serves as a helpful checklist.