

EE133 - Prelab 2

Using Oscillators to Generate FM Signals

1 Introduction

Now that we have a firm understanding of how amplitude modulation works, we will explore the real core of our project, frequency modulation. In this lab, we will see how oscillators can be made to produce FM signals. We will see that we can either produce modulation at an intermediate frequency that can then be multiplied up to a higher carrier, or we can modulate directly at the carrier frequency.

Last week, we used two signal generators to supply the multiplier with carrier and modulating signals. In a portable system, we must find a way to produce these signals without the help of bulky equipment. This is where oscillators come in. These will be the backbone of your FM transmitter.

There are a variety of flavors of oscillators, and about as many names. Many of the different oscillator types are named for their inventors, who found new and clever ways to vary the feedback path that makes an otherwise perfectly good amplifier into an oscillator. In this lab, we will use a chip-based VCO, or Voltage-Controlled Oscillator, to generate a modulated FM signal. We will also build a Colpitts Oscillator and use it as a VCO as well. Part of this oscillator will include an LC tank similar to the one you built in Lab 1. As you will see, this discrete-based design can be used in the direct creation of FM signals at a carrier frequency.

In addition to these variable oscillators, we'll investigate the properties of a crystal stabilized oscillator. We will be building this using the extra transistor located on the SA612. This will serve as the local oscillator on both the transmitter and receiver boards.

As you can see, we will be soldering a lot of pieces this week. Fortunately, you really only have to implement one discrete-based design. The rest are chip based, for which we will only need to add a few, datasheet defined, components.

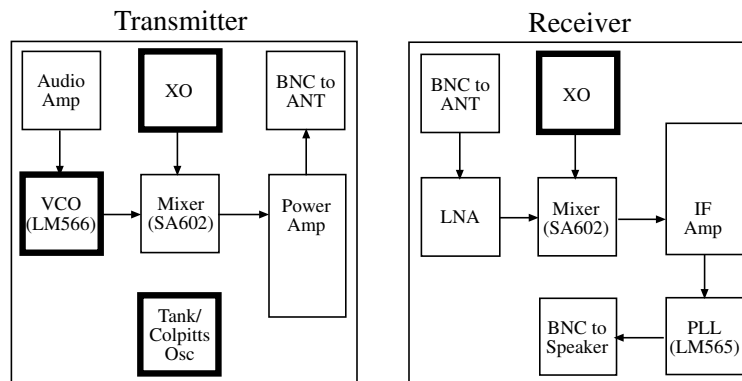


Figure 1: Roadmap for Lab 3

You are *extremely* advised to solder up the majority of components *before* your assigned lab period.

2 Mathematics of Frequency Modulation

In the first part of this lab, you will be investigating some of the mathematical properties of FM, and trying to understand the significance of the various sidebands, the Bessel functions, and the bandwidth of an FM signal.

2.1 Bessel Functions

Consider a frequency modulated signal of the following form:

$$F_{\theta}(t) = V_c \cos(\omega_c t + m_{\theta} \sin \omega_m t) \quad (1)$$

This signal can be expressed via trigonometric identities and series representations of Bessel functions as an infinite series of the following form:

$$\begin{aligned} F_{\theta} = V_c \{ & J_0(m_{\theta}) \cos \omega_c t \\ & + J_1(m_{\theta}) [\cos(\omega_c + 1\omega_m)t - \cos(\omega_c - 1\omega_m)t] \\ & + J_2(m_{\theta}) [\cos(\omega_c + 2\omega_m)t - \cos(\omega_c - 2\omega_m)t] \\ & + J_3(m_{\theta}) [\cos(\omega_c + 3\omega_m)t - \cos(\omega_c - 3\omega_m)t] \\ & + J_4(m_{\theta}) [\cos(\omega_c + 4\omega_m)t - \cos(\omega_c - 4\omega_m)t] \\ & + \dots \} \quad (2) \end{aligned}$$

The first term is the carrier, and each of the subsequent terms represents a side-frequency pair. This representation can aid in predicting the magnitude of modulated carrier and sideband signals. It turns out that for certain values of m_{θ} , the carrier or side-frequency pair can be ‘deleted’ from the signal spectrum. You will see more of what this means in the Matlab exercises.

2.2 Carson’s Rule

Although the representation given above is comprised of an infinite number of signals, in reality the higher order terms die off quickly. It is because of this that FM signals can be bandlimited without serious distortion. An approximation to the bandwidth required by an angle-modulated signal is known as Carson’s Rule:

$$B = 2f_m(m_{\theta} + 1) = 2(\Delta f + f_m) \quad (3)$$

where B is the bandwidth, f_m is the modulating frequency, m_{θ} is the modulation index, and Δf is the frequency deviation of the sideband from the carrier.

2.3 Matlab Plots

Now we will look at some FM signals in Matlab. You will need to use two scripts, which are both posted on the web page under ”Spice Decks.” Copy the two scripts into the same directory, because one needs to call on the other. You will specify your parameters in `fm_script.m`, but you should leave `get_fm_spect.m` untouched (you can peruse the file if you want to see the Matlab commands for creating an FM spectrum). The parameters of the signal are as follows:

- f_s , the frequency of the modulating signal
- f_c , the carrier frequency
- V_c , the carrier voltage
- m , the modulation index

To generate a signal, just start Matlab, open the script, and fill in the correct values for your parameters. Run the script, and the correct frequency content should appear. The script has been written to give you a ”nice” display. There may be some variation on what’s ”nice” for your computer, however, so you may have to tweak yours.

- Plotting FM frequency spectra:** Obtain plots of the frequency spectrum of an FM signal with these parameters: $f_s = 1kHz$, $f_c = 30kHz$, $m = 0$, $V_c = 1V$. Observe the outputs you obtain for the following values of m , keeping all other parameters constant: $m = 1, 2.40, 3.83, 5.14$.
- Bessel Functions:** Do your spectra agree with the Table of Bessel Functions found in the notes on FM Modulation? You can also try comparing your results to the plots of the Bessel functions in the lecture handouts. Remember that Bessel functions are supposed to predict the amplitude of the carrier and sidebands for a certain modulation index. Is there a reason why we gave you these particular values of m to play around with?
- Carson's rule:** For each of the signals, calculate the bandwidth of the transmission using Carson's rule in Equation 3 above. Comparing the values obtained with the spectra observed, is Carson's rule a good indicator of bandwidth? Is there variation of the accuracy of the rule with the value of m ?

3 The LC Tank

We will be building a Colpitts oscillator, which requires a narrow band resonator, as we will see below. Our resonator will consist of an LC parallel filter, or an 'LC tank'. In this section, you will be designing an LC tank to resonate at 24.3MHz. Figure 2 shows the ideal and actual implementations for this circuit. In the actual circuit, notice that L is replaced by a series combination of L_1 and L_2 , as well as the parasitic series resistance, R_s .

We would like our oscillator to drive a 50Ω load, but to do this, we need to tap the inductor so as not to load the output too much. You can think of this as an impedance transformation of the load resistance to some higher equivalent resistance that appears at the top of the tank (for high Q, we wish the parallel resistance of a tank circuit to be high).

Ignoring the parasitic resistance R_s , then,

$$R_{eq} \approx 50\Omega \left(\frac{L_1 + L_2}{L_2} \right)^2$$

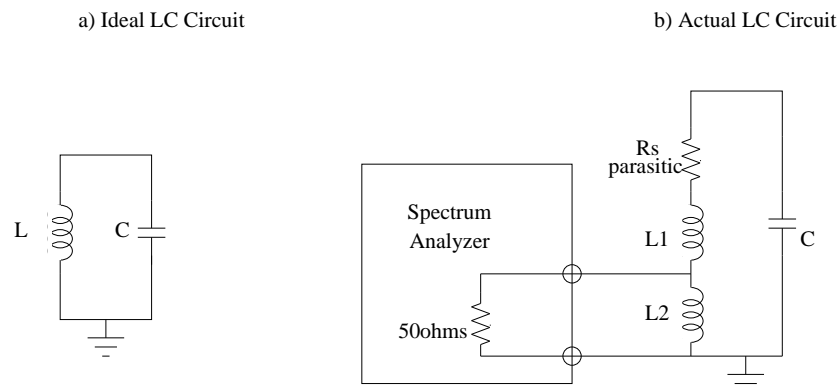


Figure 2: LC Resonators: Ideal and Actual Implementations

- Find C for 24.3MHZ Resonance:** Using $L_1 = 1\mu H$ and $L_2 = 0.1\mu H$, find the correct value for C to resonate at around 24.3MHz. Compute R_s from the Q values given for the inductors (see the web page for a list). (Remember that $Q_{series} = \frac{\omega L}{R_s}$ for an inductor.) To simplify the calculations, you may ignore the series resistance of the $.1 \mu H$ inductor.

2. **Equivalent Tank Impedance:** Using the impedance transformation methods discussed in class, what is the equivalent series resistance of the 50Ω impedance across L_2 ? You can now add this to the series resistance of L_1 and transform the new series R to a parallel R_t . What is this R_t , the resistance seen by the LC tank? What is the Quality Factor of the tank (Q_t) based on your value for R_t ? (Notice that we are assuming that the capacitors are ideal and do not severely degrade our value for Q. This is not a perfect assumption, but it will be sufficient for our use. Try measuring the Q of a silver-mica capacitor to be sure.)
3. **HSPICE AC Analysis:** Plot the AC frequency response with a current source input and a voltage output (both at the top of the LC Tank). Remember to only use capacitance values that are available on the web page. What value for C gets you closest to 24.3MHz? What is the 3-dB bandwidth of your LC tank? What is the corresponding Q_t ? How well do your hand-calculations match the simulations (give percent error)? (Again, the HSPICE input file is available on the web page.)

4 Theory behind the Colpitts Oscillator

The Colpitts Oscillator is the most famous in a family of one-transistor oscillator circuits. A useful block diagram representation of the oscillator is shown in Figure 3.

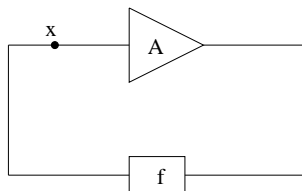


Figure 3: Block Diagram of Colpitts Oscillator

This simplified feedback circuit allows us to see how oscillation could possibly occur. Suppose we break the loop at point (x) and input a sinusoid at any amplitude. This signal, V_{in} , would be multiplied by the gain (A) and then the feedback factor (f) before coming back to point (x). If this “loop gain” Af was *exactly* equal to 1, with its phase *exactly* zero, the sinusoid would repeat forever and we would have our oscillation. This condition that $Af = 1$ with the phase of $Af = 0$ is the only possible condition for oscillation. If A times f was larger than 1, the amplitude of the sinusoid would grow exponentially with time and if $Af < 1$, the amplitude would decay exponentially to zero.

The Colpitts oscillator translates to this block diagram very well. Looking at Figure 4, we see a simplified version of the Colpitts that you’ll be building. You might notice that the transistor’s base is grounded, hinting that we probably have a common base amplifier. Recalling that the small-signal gain for this amplifier is $A_v = g_m R_{load}$ (note that this gain is positive), it is easy to see how this could make up the gain block (A), of the block diagram. The feedback (f), in this case, is the capacitive divider formed by C1 and C2. Recalling a bit about capacitive dividers, we remember that $f = \frac{C_1}{C_1 + C_2}$. In order for our condition for oscillation to be satisfied, we must set $A = 1/f$, or $g_m R_{load} \frac{C_1}{C_1 + C_2} = 1$. Also, since the phase must exactly equal zero, this oscillation can only occur at the resonant frequency of the LC tank, or at $\omega = \frac{1}{\sqrt{LC}}$.

A few issues were overlooked in the preceding analysis, and we should take a moment to consider them. First of all, you may have been wondering if the capacitive divider shifts the resonant frequency of the tank. The answer is yes, and we simply have to calculate the new tank capacitance to be $C_{total} = C + \frac{C_1 C_2}{C_1 + C_2}$ to account for this. Secondly, recalling that the input impedance looking into the emitter is $\frac{1}{g_m}$, you may wonder how this affects R_{load} . Since $\frac{1}{g_m}$ appears across the capacitive divider formed by C_1 and C_2 , we can transform this impedance up so it appears across the tank, just as we did in Lab 1 with the tapped inductors. After some math crunching (and some simplification), we find that the equivalent impedance

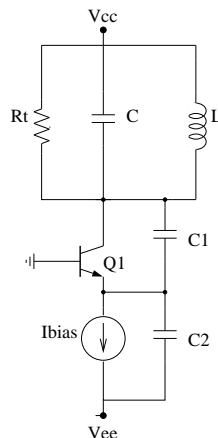


Figure 4: Simple Colpitts Oscillator

across the tank from the transistor is $R_{eq} = \frac{1}{g_m} \left(\frac{C_1 + C_2}{C_1} \right)^2$. Our new expression for the oscillation condition can now be rewritten.

$$g_m \left(R_{tank} \parallel \frac{1}{g_m} \left(\frac{C_1 + C_2}{C_1} \right)^2 \right) \frac{C_1}{C_1 + C_2} = 1 \quad (4)$$

This can be rewritten in terms of conductances using $n = \frac{C_1}{C_1 + C_2}$ and $G_{tank} = \frac{1}{R_{tank}}$.

$$\frac{g_m n}{(G_{tank} + g_m n^2)} = 1 \quad (5)$$

And we now have a very simple expression for the loop gain, Af .

One last major issue needs to be considered. It would be difficult to control this loop gain to be precisely 1. Therefore, it would seem difficult to sustain a stable oscillation. In this case, we don't have to be this precise, because the oscillator corrects for our mistakes. If we make $Af > 1$, we are sure that the amplitude of any sinusoid will rise exponentially with time. Once the amplitude begins to rise to values on the order of 100mV or greater, our assumption of small-signal operation is no longer valid. What happens next is that the small-signal g_m , begins to get smaller with increasing amplitude. Eventually, the new transconductance, called G_m , will decrease until the loop gain is *exactly* equal to 1. So we take advantage of this fact to have the oscillator amplitude limit itself. As Prof. Lee would say, Murphy never gives us a free lunch, but sometimes he does give us a light, nutritious snack to munch on.

We will finally try to estimate the amplitude of oscillation for this Colpitts oscillator. Because many people have looked at this problem before, we have a good approximation for what this large signal G_m may be. As long as $V_{tank} \gg V_t$, then to a good approximation $G_m = \frac{2g_m V_t}{nV_{tank}}$. If we plug this expression into our new expression for the loop gain,

$$\frac{G_m n}{(G_{tank} + G_m n^2)} = 1 \quad (6)$$

we find the following expression for the amplitude of oscillation, V_{tank} .

$$V_{tank} = \frac{2g_m(1-n)V_t}{G_{tank}} \quad (7)$$

Since $g_m = \frac{I_c}{V_t}$, this expression reduces to the following.

$$V_{tank} = 2I_c(1-n)R_{tank} \quad (8)$$

And we are done!

5 Analyzing the Discrete-Component Colpitts Oscillator

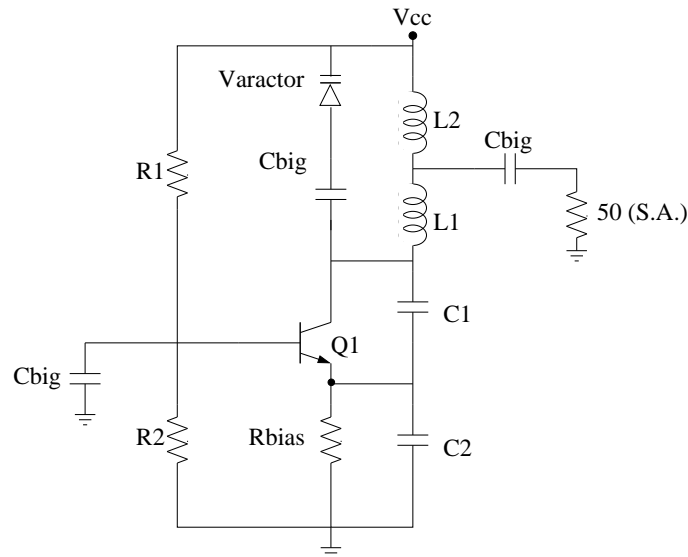


Figure 5: Colpitts Oscillator (with biasing and varactor)

So now we'll apply that knowledge to simulate and (eventually) build a discrete-based oscillator. We will use the LC tank designed above for the resonator. Note that the impedance of the next stage (the power amplifier) will be roughly 50Ω , so we will want to use the tapped inductor configuration to take our output. In Figure 5, the components have the following values: $C_1 = 60pF$, $C_2 = 120pF$, $R_1 = R_2 = 2.2k\Omega$, and $R_{bias} = 6.8k\Omega$. $L_1 = 1\mu H$ and $L_2 = .1\mu H$ as before. The current through the transistor is $I_{bias} < 1mA$. As usual, $V_{cc} = 9V$. $I_{bias} = 0.5mA$, and $n = \frac{1}{3}$, and a center frequency around 24.3MHz. We will use $C_{big} = 1000pF$.

NOTE: Since any new additional capacitance will push our center frequency below 24MHz, we will replace the C capacitor from lab 1 with the series equivalent of C_1 and C_2 in Figure 5. (In order to tune our circuit, we will physically realize C_1 as a capacitor (most likely a 33pF) in parallel with a tunable 7.5-30pF variable capacitor.)

You might note the funny looking symbol that looks like a cross between a diode and a capacitor. That component is called a varactor, or variable capacitor. Unlike physically tunable capacitors, the varactor is tuned by a voltage signal. Varactors are often made from reverse-biased diodes (as we vary the voltage across the diode, we vary the size of the depletion region, and therefore the depletion capacitance). Because of this feature of having a voltage-controlled capacitance, a varactor will allow us to tune the center frequency of our Colpitts oscillator according to some input signal, thus creating an FM output.

Now we will characterize some features of the Colpitts oscillator.

1. **The Varactor Range:** Look at the data sheet for the 2N3904 NPN transistor. What is the range of input capacitance, C_{ibo} , for a voltage range of 0-3V? This is the capacitance value of interest, since we will be connecting the transistor as a diode.

2. **The New Resonant Frequency of your Circuit:** Ignoring the varactor capacitance for now, what is the resonant frequency of this circuit? Include $C_\pi = 2pF$ in your calculation. How much frequency deviation would the varactor allow you to obtain?
3. **Small-Signal Gain:** What is the loop gain of this circuit for the small-signal case? Will the oscillations grow with time?
4. **Output Voltage:** Eventually, the amplitude will stop growing as G_m decreases. For $I_c = .5mA$, what is the resulting V_{tank} ? If we tap the output inductors, what is the output amplitude as seen by the spectrum analyzer? What is the output power into the 50Ω input of the spectrum analyzer (in Watts and in dBm)? (Hint: The power delivered to the effective resistance of the tank circuit is the same as the power delivered to the 50Ω spectrum analyzer input, since ideally the reactive components don't dissipate any power!).
5. **HSPICE Analysis:** Simulate this circuit using the HSPICE deck provided and the Toshiba 2SC3302 bipolar transistor. Use the midrange value for the varactor capacitance.
 - (a) How do your calculations compare with your HSPICE results? Look at the output file to find the collector current. Is this higher than you expect? Why? What is the expected V_{tank} for this current?
 - (b) What is the total DC power consumption of this oscillator?
 - (c) Separately plot the output voltage across the tank and the voltage across the 50Ω impedance. Do the results make sense to you?
 - (d) Now plot the collector current vs. time. Why does the current have the shape that it does?
 - (e) Change the varactor capacitor to the maximum and minimum capacitance values - what variation in frequency can you obtain?
 - (f) Now change the bias resistor to $1k\Omega$ and look at the voltage and current waveforms again. Is the result what you'd expect?

/noindent In order to isolate the bias voltage on the varactor from the bias voltage on the collector of the transistor, we will need to put the varactor in series with a large silver-mica capacitor (since the circuit operates at high frequencies). You will need to pick a capacitor that is large enough so its series combination with the varactor is nearly the value of the varactor.

We will now move onto a more conventional design of the Colpitts Oscillator.

6 Building the Colpitts Oscillator with the SA612

Now that we've analyzed a discrete-based Colpitts, we will use our experience to design a crystal-based oscillator. In lab, we will compare the oscillation characteristics with the one you designed in the previous section.

- **Using the SA612-based Colpitts:** Read the SA612 Data Sheet and Application Notes on the EE133 Web Page to find out how crystal oscillators work and how to build one using the SA612. You will be building a fundamental-mode crystal oscillator. You can use two $30pF$ capacitors as your feedback path. Sketch the schematic in your lab notebook.

7 LM 566 Voltage Controlled Oscillator

The LM 566 is a general-purpose Schmitt Trigger based Voltage Controlled Oscillator (VCO), similar to the one discussed in class. It is capable of working up to $1MHz$, and producing both square and triangle wave outputs. Keep this in mind as you examine the results out of the multiplier.

These are great devices for the creation of FM signals at low frequencies. We can apply a changing voltage to the input and the output oscillation frequency varies. To make good use of it, we'll need to know how to set it up and what to expect from it. We are going to use the VCO to produce a base oscillation at $300kHz$. Then we're going to figure out what type of signal we're going to need from the audio microphone circuit in order to produce a change of about $100kHz$ ($50kHz$ deviation on either side).

Let's do that now:

1. **Familiarizing Yourself with the LM566:** Download and read the LM566C datasheet from the EE133 Web site. All of the information you need to answer the following questions is contained in those pages.
2. **Calculate the Center Frequency:** The center frequency is set by an RC network at pins 6 and 7. Using the equation given on the datasheet, design a network with a center frequency $f_0 = 300kHz$. Use $V_+ = 9V$ and $V_5 = 7.5V$.
3. **Calculating the Frequency Sensitivity:** Calculate k , the frequency sensitivity in kHz/V . How much does the control voltage need to deviate to produce a $\Delta f = 50kHz$? Note: You may have to go searching through the datasheet to find this information.
4. **Predicting DC Power Drain:** How much DC power will this chip consume? (Again, you may have to go searching for this information.)

8 Soldering Recommendations

A few words about soldering for this week:

- Once you have analyzed your circuit and confirmed that it'll work, you are advised to solder up the Colpitts oscillator in office hours before your lab period and verify that it is at least oscillating. Be VERY careful about your construction. These circuits are very touchy. You'll need to bypass your power supply lines very well and also keep your leads short.
- Of lesser importance is to solder up the SA612 based oscillator and the LM566 VCO onto your transmitter board. These generally work with very little tinkering.
- Do not connect any of the blocks together. You will do that after you have characterized them all.

9 The Future and Some Design Choices

After you build the Low-Noise Amplifier in the next lab, you will have completed the main building blocks for an FM transmitter/receiver pair. The last few blocks will provide additional amplification and filtering to allow for greater distance of transmission.

In this class, we're going to allow you to choose from a couple of transmitter designs to suit your available time and taste.

1. The first option is the standard chain that's been outlined in the labs. Your audio signal will modulate the output of the VCO around an intermediate frequency of $300kHz$, and then this will be mixed up to a frequency near $24.3 MHz$. This signal will then enter the power amp before transmission. Most transmitters that use this route will obtain $2-4dBm$ of output power, but have high stability. As you saw in Lab 2, one of the biggest problems is that the mixer has an upper limit to the power of signal it can take before creating distortion. In addition, the LM566 VCO outputs square waves, and not a pure tone, so there are some additional filtering concerns with this approach.

2. The second option is to use the Colpitts Oscillator to modulate your audio signal directly up to 24.3MHz. As you can see from the prelab, one could apply a voltage directly across the varactor, and create an FM signal around 24.3MHz. Unfortunately, since this signal is not based around a crystal oscillator, the carrier frequency can drift significantly, even within a single use of the transmitter. The added complexity makes it a headache to solder and debug. On the other hand, students have obtained 10-15dBm of output with this, nearly a factor of 10 better than the alternative!

You may choose either method as your method of modulation. If you do, remember to remove the power to the other circuit once you've characterized it.