ECE-311
Experiment #1

Getting To Know the Laboratory's Equipment

Objective:

In this exercise you will become familiar with the communication laboratory's equipment, which will be used in the hardware experiments during this semester. The three main pieces of equipment are the function generator, the oscilloscope, and the spectrum analyzer. The other piece of equipment that you will use is a power supply. The function generator and spectrum analyzer will be the main focus of your investigations since it is assumed you are all expert oscilloscope users and power supply adjusters.

In addition, you will review and use your knowledge and skills in measuring and calculating the power of a signal.

Introduction:

Your laboratory teaching assistant will give you a brief lecture and demonstration on the use of the function generator and spectrum analyzer.

Pre-Lab:

Calculate the theoretically expected average and RMS values for the waveforms of *Part One* in the "Task" section.

Calculate the theoretically expected powers for the sinusoids of *Part Two* in the "Task" section. Express you answers in Watts and dBm. Note, that, for the most part, you will be using 50 Ω terminations for this course. I.e., it is assumed that the signals are developed across a 50 Ω resistor.

Task:

Part One:

Use the function generator to produce the following periodic waveforms:

Waveform	Fundamental Frequency	Peak-to-Peak Amplitude	d.c. Offset
Square wave (Rectangular Wave with 50% duty cycle)	1 MHz	3 volts	0 volts
Rectangular Wave with 20% duty cycle	5 MHz	5 volts	*
Triangular Wave	100 KHz	2 volts	*
Sawtooth Wave	100 KHz	3 volts	0 volts

*Adjust the d.c. offset so the "baseline " of the signal is zero volts.

a) Display each waveform on the oscilloscope. First, show one period and then ten periods.
b) Use the oscilloscope's "soft" keys to measure the signal's frequency, average value, and RMS value. Record these values.
c) Draw a sketch of each waveform. Label all critical points. (One period will be adequate.)

Part Two

1. Produce a 1 MHz sine wave with an amplitude of one (1) volt (2 volts peak-to-peak) and an average value of zero. Use the oscilloscope to verify the signal's parameters.
2. Next, use the spectrum analyzer to measure the signal's amplitude in dBmV and volts and its power in dBm. Record these measurements
3. Repeat the previous two steps for sinusoids of amplitudes of two (2) and three (3) volts.
4. Repeat the previous three steps for a sinusoid of 5 MHz.

Note that you will have taken data for six different sinusoids.

If you are having difficulty doing any of these measurements, ask for help from your laboratory teaching assistant.

Data Analysis:

For *Part Two's* data, is there any relationship between the theoretically calculated and measured amplitudes and powers? There may not be any obvious correlation between the two because of either an unforeseen systematic error in taking the data or a mistake in the circuit model used to do the theoretical calculation. However, if you choose one signal as a reference signal (say the unit amplitude sinusoid) and take the ratio of the other signals' amplitudes (or powers) to it. The expected and measured values of these ratios should be very close. This illustrates a very important fact in communication systems; it is the relative signal levels that are important not absolute levels.

Report:

In your report for *Part One* include the sketches of the periodic waveforms you studied and a comparison of the calculated and measured average and RMS values.

In your report for *Part Two* include a table (see below) comparing the expected and measured values for the amplitudes and powers of the various sinusoids along with the ratios you calculated in the "data analysis" section.

Freq. (Hz)	Expected Amplitude (dBmV)	Measured Amplitude (dBmV)	Calculated Power (dBm)	Calculated Relative* Power (dB)	Measured Power (dBm)	Measured Relative* Power (dB)
●	●	●	●	●	●	●
●	●	●	●	●	●	●
●	●	●	●	●	●	●

*Use the unit amplitude sinusoid as the reference.

$P_{relative}$ (dB) = P (dBm) − $P_{reference}$ (dBm)

===
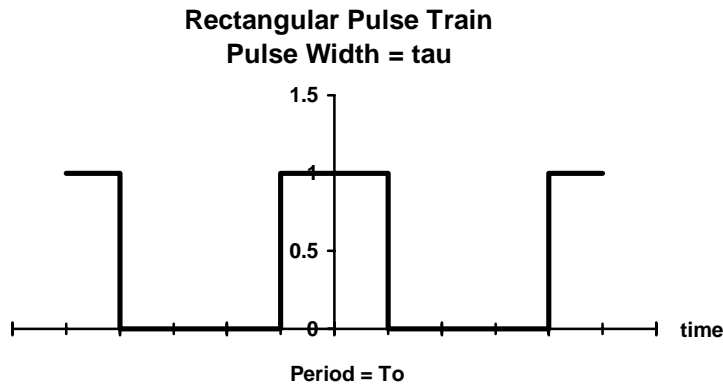<div align="center">USEFUL INFORMATION</div>

Decibel Formulae:

$$\text{Amplitude (in dBm)} = 10\log\left|\frac{P}{1mW}\right| \qquad \text{Amplitude (in dBmV)} = 20\log\left|\frac{V}{1mV}\right|$$

<div align="center">P is in Watts   V is in Volts</div>

Duty Cycle Defined:

Rectangular Pulse Train
Pulse Width = tau

Period = To

$$\text{Duty Cycle (in \%)} = (\tau/T_o) \times 100$$

Average Value and Power for a Periodic Signal:

Suppose, in the following, f(t) is a real, periodic signal with a fundamental period of T_0. Then, the average value of f(t) is

$$\langle f(t) \rangle = \frac{1}{T_0} \int_{t_x}^{t_x+T_0} f(t)dt \qquad \text{Volts (or Amps).}$$

Next, the average power of f(t) is

$$P_{ave} = \langle f^2(t) \rangle = \frac{1}{T_0} \int_{t_x}^{t_x+T_0} f^2(t)dt \qquad \text{Watts.}$$

In the above, t_x is arbitrary indicating that the integrations are over the total period, but may "start" anywhere.

Finally, the RMS (root mean square) value of f(t) is just the square root of P_{ave}. Note, that some times P_{ave} is referred to as the mean square value of f(t).

The average power formula given above assumes the signal is developed across a one-ohm resistor. For other resistance values a modification must be made to the number calculated above. If f(t) is a voltage (current) the P_{ave} must be divided (multiplied) by the actual resistance value to get the actual power in the signal.

ECE 311 Communication Engineering

Lab Assignment #2

The Spectra of a Periodic Signal

Objective:

The purpose of this experiment is to calculate the exponential Fourier series coefficients of a periodic signal and to use MATLAB to plot the results of this Fourier analysis. In the laboratory, you will generate this waveform and investigate its spectral properties using a spectrum analyzer.

Introduction:

Please read sections 2.8 and 2.9 of the text for a review of the exponential Fourier series. If you are unfamiliar with MATLAB you should consult the MATLAB manual. In the Appendix A, there is a MATLAB example showing the plotting of Fourier series coefficients and the corresponding continuous functions, both approximate and "real".

Pre-lab:

Consider the periodic signal, $f(t)$, shown in the next figure. Where applicable, in the following tasks, use MATLAB to do the numerical calculations and plots.

f(t) (T$_0$ = 2 π)
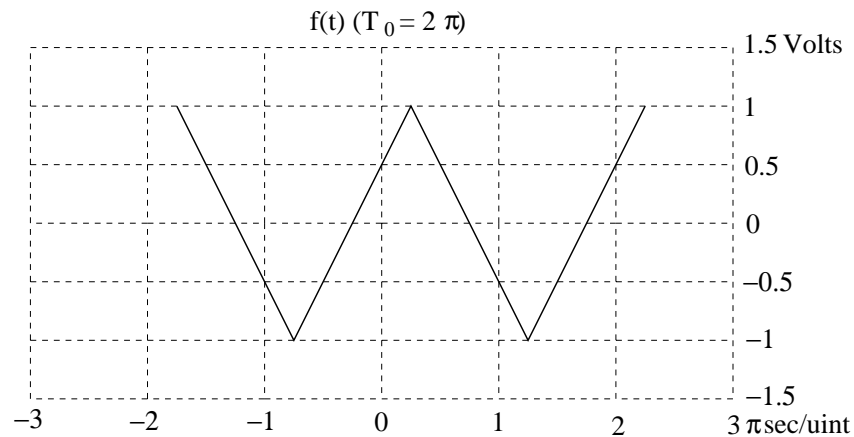
1) Find an expression for its exponential Fourier series coefficients, D_n.

2) Plot the *magnitude*, $|D_n|$ (in volts) and *phase*, $\angle D_n$ (in degrees) of the first thirteen coefficients ($n = -6, -5, \cdots, 6$) versus *frequency*, ω (in rad/sec).

3) Using Parseval's Theorem, find the (average) power in the signal's first thirteen coefficients. Express your answers in Watts and dBm.

Some warnings and comments: First, to find an expression for the D_n's of $g(t)$ you do NOT have to do an integration. Note, that $g(t)$ is just a time shifted (delayed) symmetric triangle wave. The D_n's for the symmetrical triangle wave are given in Appendix B. Next, the powers calculated using Parseval's Theorem assume a terminating resistance of 1 Ω (ohm). When you compare these powers to the measured powers from the laboratory, you must adjust them (the calculated D_n's) because the spectrum analyzer assumes a 50 Ω termination.

Task:

Use the function generator to produce a triangular wave with unit amplitude (peak-to-peak 2 volts) and a frequency of 1 MHz (or the maximum frequency allowed by the function generator for this type of signal). Verify that you have the correct waveform by displaying it on the oscilloscope. Then, use the spectrum analyzer to examine the magnitude of this signal's frequency spectra. Measure and record the power (in dBm) of the fundamental ($n = 1$) and the next five ($n = 2, 3, 4, 5, 6$) harmonic components.

Data Analysis:

At each frequency component calculate the relative power (in dB) using the power at the fundamental frequency ($n = 1$) as the reference. With powers measured in dBm the calculation of the relative power is quite straightforward as follows:

$$10 \log \left| \frac{P}{P_{\text{ref}}} \right| = 10 \log \left| \frac{P}{1\text{mW}} \right| - 10 \log \left| \frac{P_{\text{ref}}}{1\text{mW}} \right| \quad ,$$

or in words,

Relative power (in dB) = Power of the component of interest (in dBm) $-$ Power of the reference (in dBm).

Report:

For the pre-lab portion of your report, you should submit the following:

1) The "derivation" of the expression for D_n .

2) Plots (for $n = -6, -1, \cdots, 6$) of

   (a) $|D_n|$ vs. ω.
   (b) $\angle D_n$ vs. ω.

3) Two plots of the average power using Parseval's Theorem for the first thirteen ($n = -6, -1, \cdots, 6$) one in Watts and dBm.

For the laboratory portion of your report, produce a table for the data gleaned from measurements and those you calculated in the pre-lab. The table should have the following columns:

| n	Calculated* Power in dBm	Calculated* Power relative to $n = 1$ (in dB)	Measured Power in dBm	Measured Power relative to $n = 1$ (in dB)
⋮	⋮	⋮	⋮	⋮

* Including the adjustment for 50 Ω termination.

In your report, answer the following questions:

1) How close are the relative power columns – calculated and measured?

2) Does the power spectrum depend on the fundamental frequency? Explain!

3) What is the meaning of "n?" Why are there negative "n's?"

4) Verify that the *amplitude* of the Fourier components for this waveform vary as $(1/n^2)$.

Appendix A to Lab #2

Calculating and Plotting Exponential Fourier Series Coefficients

Consider the periodic signal, $g(t)$, shown below. The following tasks are performed using MATLAB to do the numerical calculations and plots.

g(t)
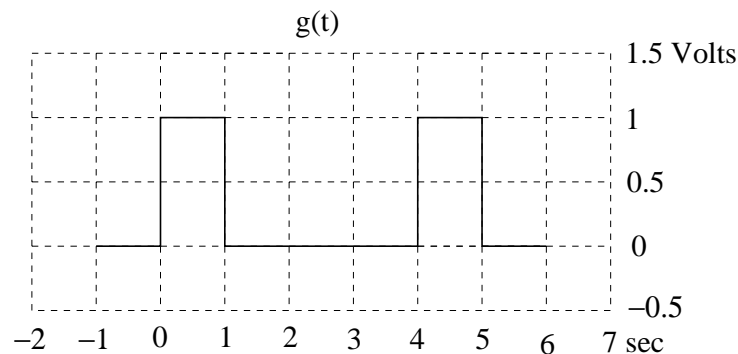
1) We want to evaluate the Fourier series coefficients using

$$D_n = \frac{1}{4} e^{-jn\frac{\pi}{4}} \text{sinc}\left(n\frac{\pi}{4}\right).$$

Plot the *magnitude*, $|D_n|$ (in volts) and *phase*, $\angle D_n$ (in degrees) of the first twenty-one coefficients ($n = -10, -9, \cdots, 10$) versus *frequency* (in rad/sec).

2) Plot two periods of $g(t)$, directly–i.e., by creating a vector of samples of $g(t)$ and plotting that vector.

3) Plot an *approximation* to $g(t)$ using these first twenty-one terms of the exponential Fourier series.

An annotated copy of the MATLAB code used to generate the attached plots in Figures 3 and 4.

n = [-10:10]; % Sets up the vector of integer indices. The semicolon suppressed output to the screen.

help exp

EXP Exponential.

EXP(X) is the exponential of the elements of X, e to the X.

For complex Z=X+i*Y, EXP(Z) = EXP(X)*(COS(Y)+i*SIN(Y)).

See also LOG, LOG10, EXPM, EXPINT.

z = n*(pi/4);

Dn = 0.25*exp(-i*z) .* sinc1(z); % The symbol " .* " means element-by-element multiply

magDn = abs(Dn); % Finds the magnitude of Fourier series coefficients

argDn = angle(Dn)*(180/pi); % Finds the phase (in degrees) of FS Coefficients

w=0.5*n;

stem(w,magDn),xlabel('Frequency in rad/sec (units of pi)'),grid

title('Magnitude of the Exponential Fourier Series Coefficients')

stem(w,argDn),xlabel('frequency in radians/sec (units of pi)'),grid

ylabel('degrees')

title('Phase of the Exponential Fourier Series Coefficients')

The following code is used to find the sum given by

$$g(t) \cong \sum_{n=-10}^{n=+10} D_n e^{jn\omega t}.$$

See Fig. 5. Note that explicit sum need not be calculated because we can use the properties of matrix multiplication to achieve the same result.

n = [-10:10];

nwo = n*(pi/2); % Defines the eleven frequencies of the sum

t = [0:0.01:8]; % Defines the time sampling points

BIG = nwo' * t; % Creates a "big" matrix so we can use matrix multiplication to do the sum

g = Dn * exp(i*BIG); % Here's where the sum is done

plot(t,real(g)),grid,xlabel('Seconds')

title('Approximation to g(t) using the first ten components of the Fourier series')

The following code is used to generate two periods of the "real" function g(t). See Fig. 6.

It uses a unit step function, $u(t)$, which is created in a homemade MATLAB function file. A copy is shown below. The ability to use the unit step function to write piecewise functions proves extremely efficacious when programming in MATLAB! Learn how to use it if you have not done so in the past.

gt=(u(t) - u(t-1)) + (u(t-4) - u(t-5)) + u(t-8);

plot(t,gt)

grid,xlabel('seconds')

title('The "real" g(t)')

axis([0 8 -0.2 1.2]);

% This changes the plot y-axis so it is the same as the previous plot of approximation to g(t).

Operations on the vector magDn can give power in Watts or dBm. However, care must be taken in interpreting the result of taking the logarithm since the magDn contains zeros. (Note, here a one-ohm termination has been assumed.)

Pave = magDn . 2 % Squares the elements of the vector of F.S. coefficient magnitudes

PdBm = 10*log10(Pave/1e-3)

The above powers have not been plotted here.

help log10

LOG10 Common (base 10) logarithm.

LOG10(X) is the base 10 logarithm of the elements of X.

Complex results are produced if X is not positive.

See also LOG, LOG2, EXP, LOGM.

The following MATLAB code generates a unit step function and must be place in a file "u.m".

```
function y=u(x)

y = 0.5 + 0.5*sign(x);
```

The following MATLAB code generates a sinc function and must be place in a file "sinc1.m".

```
function y=sinc1(x)

k=length(x);

for i=1:k

if x(i) == 0

y(i) = 1;

else

y(i) = sin(x(i))/x(i);

end

end
```
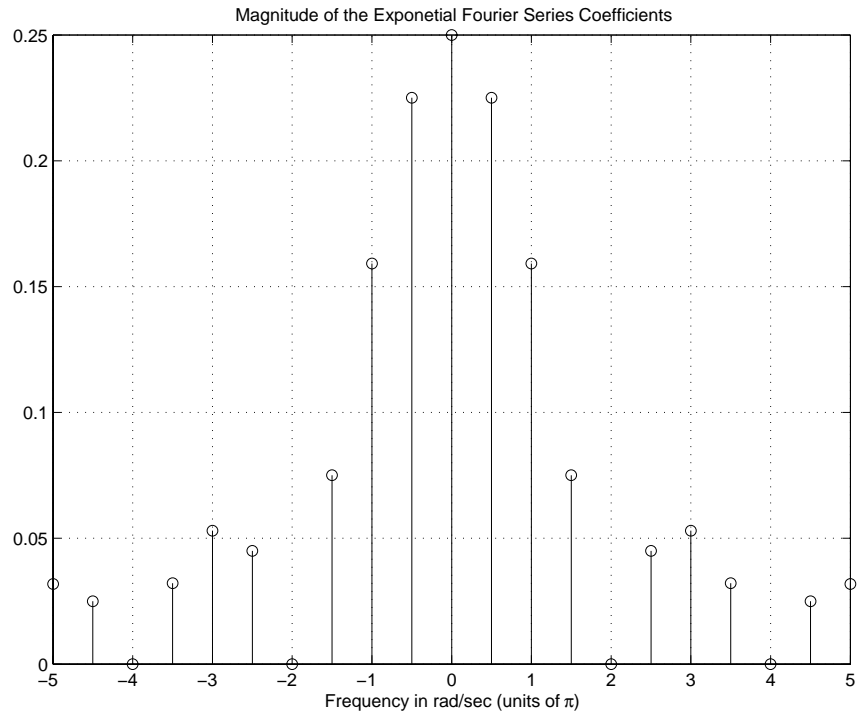
Fig. 3: Magnitude $|D_n|$
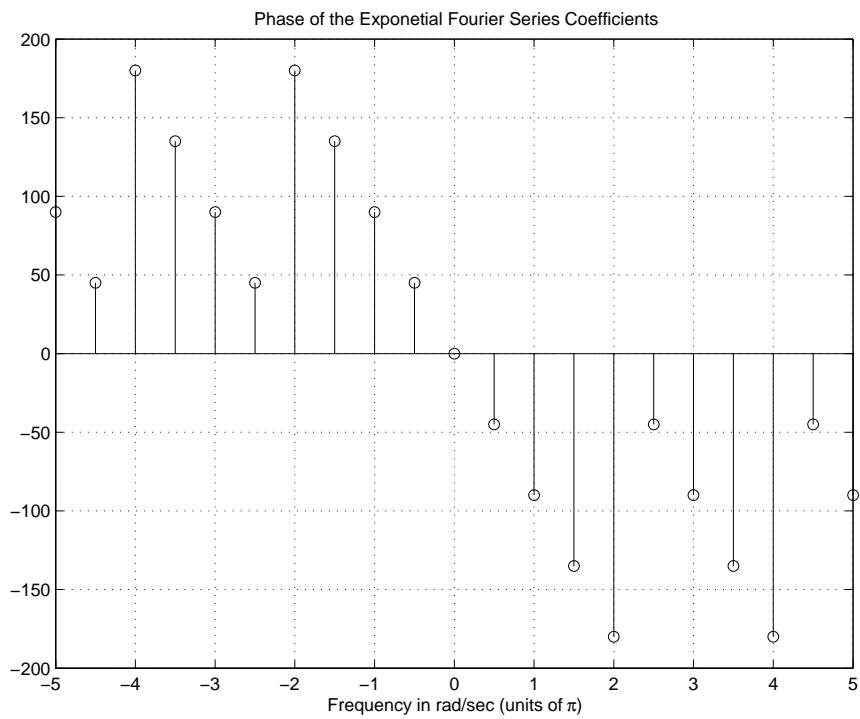
Fig. 4: Phase $\angle D_n$
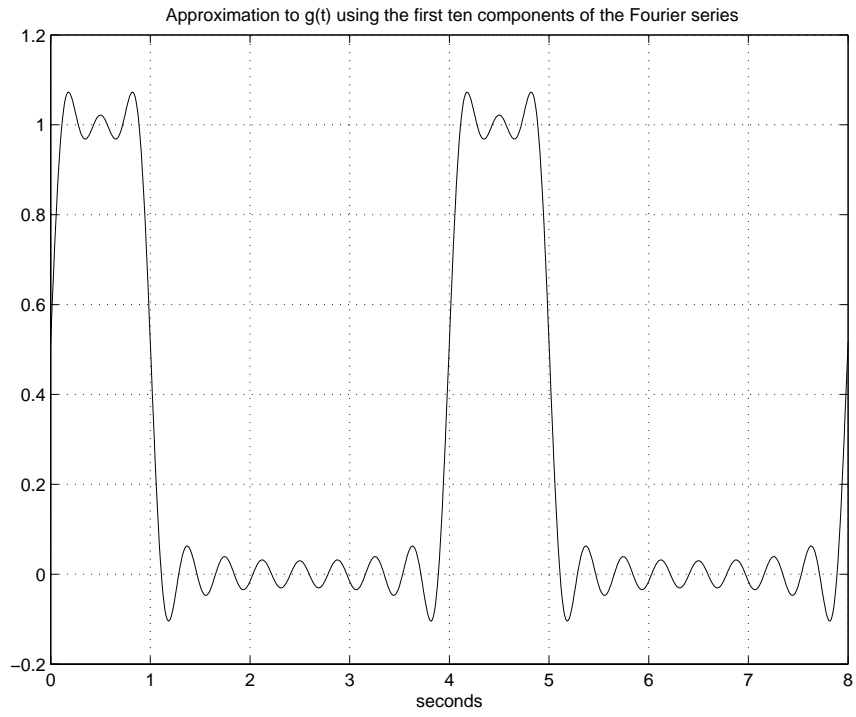
Fig. 5: Approximation of $g(t)$
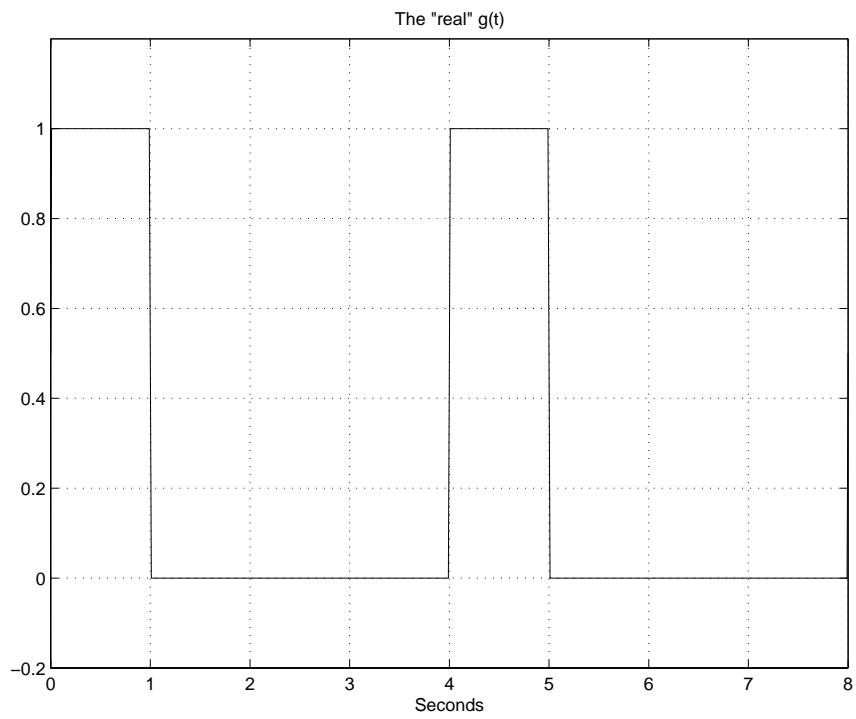
Fig. 6: Real "$g(t)$"

Appendix B to Lab #2

Coefficients of Exponential Fourier Series for Some Often-Seen Periodic Signals

Symmetric Triangle Wave

Period = T_0
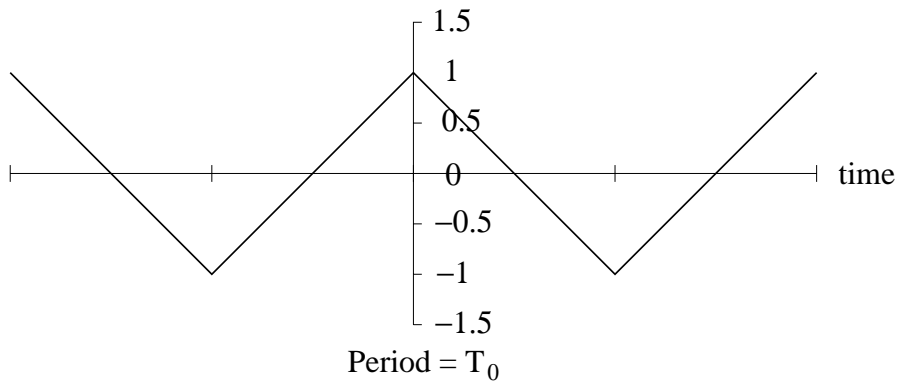
$$D_n = \begin{cases} \text{sinc}^2(n\pi/2), & n \neq 0 \\ 0, & n = 0 \end{cases}$$

Rectangular Pulse Train
Pulse Width = τ

Period = T_0
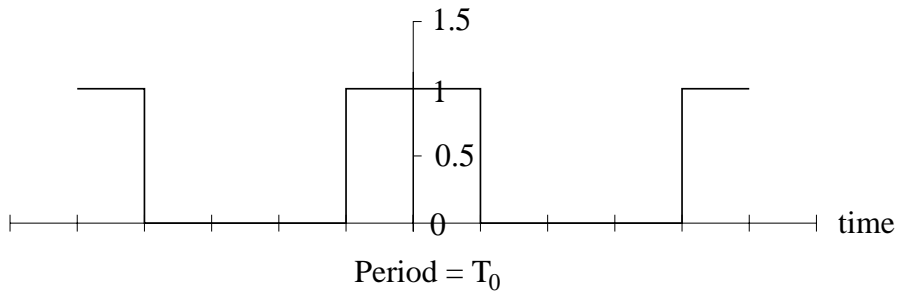
$$D_n = \frac{\tau}{T_0}\text{sinc}\left(n\pi\frac{\tau}{T_0}\right)$$

Symmetric Sawtooth

Period = T_0

$$D_n = \begin{cases} \frac{j\,(-1)^n}{n\pi}, & n \neq 0 \\ 0, & n = 0 \end{cases}$$

ECE 311 Communication Engineering

Lab Assignment #3

Fourier Series Coefficients using the FFT

Objective:

The purpose of this exercise is to introduce you to the FFT (Fast Fourier Transform) as a method of finding the (exponential) Fourier series coefficients of a periodic function. You will use MATLAB to calculate the coefficients using both the FFT algorithm and the theoretical formula. In the lab, you will generate the waveform and use the oscilloscope's FFT module to look at the signal's spectra.

You will learn the following:

1) How to interpret the output of the FFT, the spacing, the range, and the location of the positive and negative parts of the frequency components; and

2) The effects of the sampling rate and number of samples on the output.
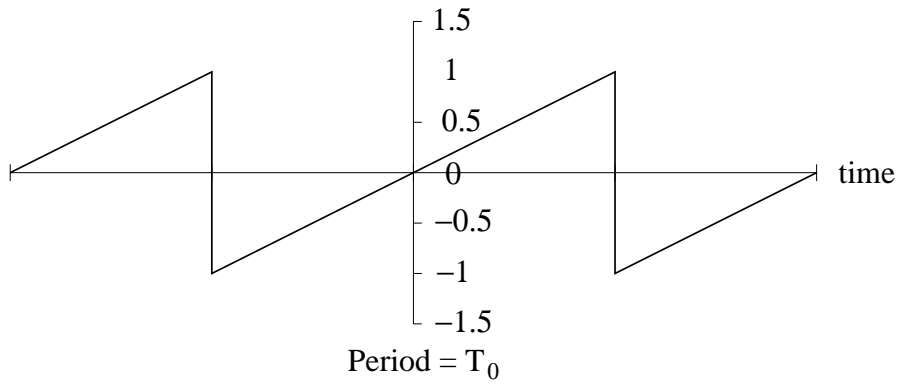
Introduction:

Read textbook Sections 2.10 and 3.9 (especially Computer Examples C3.1 and C3.2). A useful review of FFT theory as it relates to the Agilent (Hewlett-Packard) oscilloscope module you will use in the lab can be found at http://www.educatorscorner.com/experiments/pdfs/exp53a.pdf.

Pre-Lab:

Use MATLAB's FFT and FFTSHIFT functions to find the (exponential) Fourier series for the signal given below. Use two periods of the signal with a total of 128 sample points.

g(t)

$T_0 = 1$ sec

$$D_n = \begin{cases} \frac{j\,(-1)^n}{n\pi}, & n \neq 0 \\ 0, & n = 0 \end{cases}$$

1
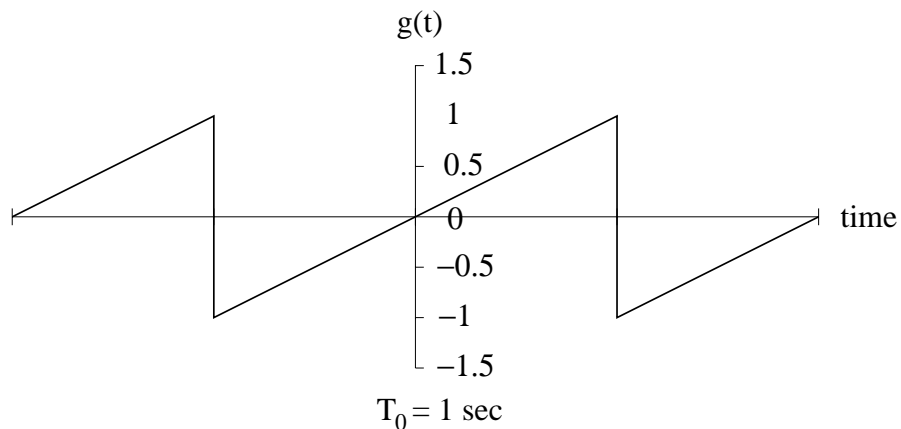
Plot the magnitude (in volts) of Fourier series coefficients vs. frequency (in Hertz) using the following guidelines.

a) Arrange the plot such that the DC component ($n = 0$) is at the center. (This involves using the MATLAB function FFTSHIFT.)

b) Plot the terms for $n = -5, -1, \cdots, 5$ only.

c) The plot should be vs. frequency in Hertz. (See the comment that follows.)

d) Calculate the magnitude (in volts) of the theoretically calculated (referred to in the following as "actual") coefficients given by the formula. Do this for the terms $n = -5, -4, \cdots, 5$. Plot these magnitudes using the same axis as Part (c).

e) In addition to the plots, record the FFT and "actual" amplitudes (in volts) at the frequency components for $n = 1, 2, 3, 4, 5$. Tabulate your data as shown the report requirement.

Comment on the Frequency Axis for the Plots:

This is the most confusing vectors to create because of how the FFT "scrambles" its output. But, without proof, it can be shown by a hard working student, that the frequency vector against which the shifted FFT, created by FFTSHIFT, should be plotted, is given as follows:

$$\mathbf{f} = \Delta f \left\{ \mathbf{n} - 1 - \frac{N}{2} \right\}$$

$\mathbf{f}$ = vector of frequencies

Δf = the frequency increment

$\mathbf{n}$ = a vector of integers from 1 to N

N = the number of samples in the vector used for the FFT.

Thus, the MATLAB code to generate the initial plot of Fourier series coefficient magnitudes vs. frequency is

plot($\mathbf{f}$, abs(FX))
where FX is the output vector from the FFTSHIFT MATLAB function.

This plot will look a little "cramped" and you will want to "zoom-in" on the middle elements of the vector. For help on doing this see the MATLAB manual and try typing "help axis."

Task:

In the lab, generate the periodic waveform shown above except make the fundamental frequency 100 kHz. Display it on channel 1 of the oscilloscope using a 50 Ω termination. Use the FFT module on the oscilloscope to view and measure the amplitude spectra of the periodic waveform. Measure the amplitudes and frequencies for $n = 1, 2, 3, 4, 5$ components. Record this data.

The FFT function appears in a soft key menu under the $\pm$ (math) button. There are two math function choices under this button, F1 and F2. The FFT is available with the F2 choice. (Turn it ON, press MENU, and choose FFT under OPERATION.) Once FFT is chosen, to get a better looking frequency plot, turn OFF the channel one, time-domain display by pressing the "*1*" button a couple of times.

The FFT menu has a number of soft keys. Use the default settings for most of the options, but make sure you use the FLATTOP choice in the WINDOWS soft key for amplitude measurements and the HANNING choice for frequency measurements.

The frequency axis span is adjusted using the *time/division* knob. Since only amplitudes are being displayed and you only have to measure the positive frequency components, you can set DC (0 Hz) to the left of the screen.

The amplitude axis is logarithmic with units in dBV with 1 volt RMS as the reference. Pressing the *cursor* button activates the cursors with amplitudes displayed in dBV. Pressing the FIND PEAKS soft key automatically puts the cursors at the two largest spectral lines on the screen. Cursors can be selected using the soft keys and manually moved using the *cursor* knob.

Data Analysis:

Convert the lab data from dBV to volts. First, convert the data to RMS voltages (V_{RMS}) by inverting the definition of dBV, i.e., dBV = 20 log (V_{RMS}). Second, adjust the RMS voltage to peak voltages (V_P) by multiplying by 1.414 (why?). Finally, this peak value must be divided by a factor of two. This factor of two is necessary because, when the oscilloscope FFT module presents the data at a given frequency, it sums the magnitude of the coefficients at both negative and positive frequencies. The number resulting from this procedure will be referred to as the "measured" value for the coefficient.

Next, using the "actual" amplitude as the reference, find the percentage error for each component's amplitude "computed" using MATLAB's FFT and "measured" using the oscilloscope's FFT module. Put your results in the appropriate row and column in the table below. Recall that

Percentage error $= 100 \times |$(test value – reference value)/reference value$|$.

Report:

For your report, submit the two graphs from your pre-lab and your analyzed data tabulated as shown below. As always, your report should start with a cover sheet and end with a copy of the raw data you took in the laboratory.

n	"Actual" Amplitudes (Volts)	"Computed" FFT Amplitudes (Volts)	Percentage Error for MATLAB	"Measured" FFT Amplitudes (Volts)	Percentage Error for Measured
⋮	⋮	⋮	⋮	⋮	⋮

Due date: Next lab session to your TA.

ECE 311 Communication Engineering

Lab Assignment #4
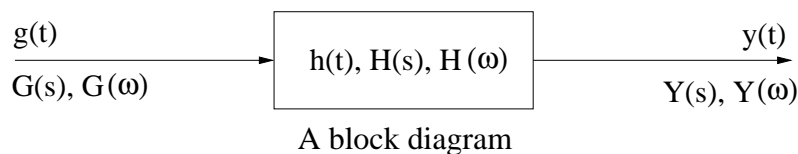
Investigation of a Simple Filter

Objective:

The purpose of this experiment is to investigate the frequency response of a simple linear system (a filter) and to use it to predict the system's output for a periodic input signal. We note that in this experiment a deterministic, periodic signal is used to do this investigation in order to facilitate the use of laboratory equipment to make measurements. In communication systems the information that is filtered is random and non-deterministic.

You should calculate a theoretically expected frequency response for the circuit given for this experiment. Using this result, you can then predict the effect this filter has on the spectrum of the input. A computer simulation should verify the results of this prediction.

In the laboratory, you should build and test the circuit. In particular, you want to verify the results of the calculation and simulation, by looking at the ratio of the output Fourier series coefficients to the input Fourier series coefficients.

Introduction:

The block diagram for a single-input, single-output linear system is shown below. If the system is asymptotically stable and we are interested in the operation of the system in the steady-state, the system's frequency response, H(ω), shows how the system modifies the input signal's spectrum. In this case, we call the system a filter. For this experiment, review the appropriate material on linear systems and frequency response from your linear system's course (ECE 310).

$$
\begin{array}{ccc}
\dfrac{g(t)}{G(s),\, G(\omega)} & \boxed{\; h(t),\, H(s),\, H(\omega) \;} & \dfrac{y(t)}{Y(s),\, Y(\omega)}
\end{array}
$$

A block diagram

For linear systems, we have

$$ H(s) = \frac{Y(s)}{G(s)}, \quad H(\omega) = H(s)|_{s=j\omega}. $$

Note, that in this experiment the frequency response will be evaluated at the discrete frequencies of the input signal's spectrum. For a linear system with a periodic input, the input and output set of frequencies are the same. The only thing that a linear system "does" is to change the magnitude and phase of the spectra at those frequencies.

In appendix A, a simple filter is analyzed. H(ω) is calculate, the MATLAB code used to generate plots of H(ω) is given, and the issue of scaling is discussed. In appendix B, the filter of appendix A is simulated using PSpice and its frequency response is found.

Theoretical Calculation

Find an expression for H(ω), the circuit's (shown in the figure) frequency response. Use MAT-LAB to plot its magnitude $|\text{H}(\omega)|$ (in dB) and phase angle $\angle H(\omega)$ (in degrees) for the fundamental frequency (f_0) and the first seven harmonics ($f = nf_0; n = 1, 2, \cdots, 8$). Remember that "$f$" is in Hertz (cycles/second) and "ω" is in radians/second and that $\omega = 2\pi f$.

FILTER

Vout

R1
50

Vin
AC=1
Phase=0

Function

Generator

L1
1.2uH

C1
5.6n

R2
50

Spectrum

Analyzer
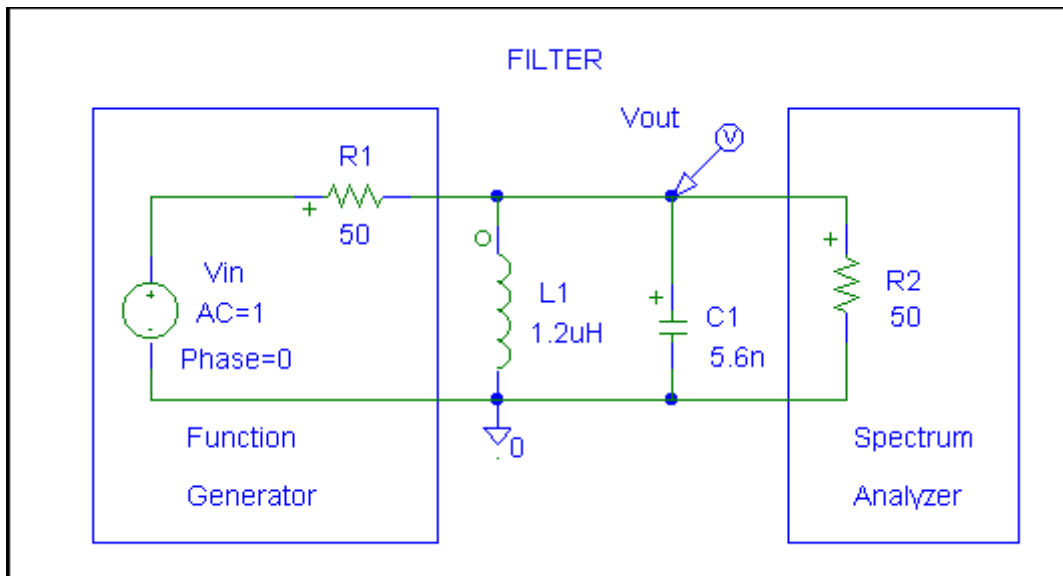
A circuit

Simulation

Simulate the circuit for this experiment using Pspice. You can use Pspice to find the circuit's frequency response, and therefore check the results of your theoretical calculation. This is done by using the "AC Sweep" option under the "Analysis" menu.

Tasks:

Build the circuit, excite the circuit with the input signal shown below and do the following measurements.

Use the spectrum analyzer to measure the magnitude (in dBm) of the Fourier series coefficients of the signal without the filter inserted in the circuit. Make this measurement for the fundamental frequency and the first seven harmonics ($f = nf_0; n = 1, 2, \cdots, 8$). Next, insert the filter into the circuit and repeat the measurements for the same frequencies used without the filter. Record your measurements.

The rectangular pulse train input signal:

    DC offset = 0 volts
    Pulse Amplitude = 5 volts (peak-to-peak)
    Duty Cycle = 20%
    Period = 1 microsecond

Generic rectangular pulse train
pulse width $= \tau$

Period = T
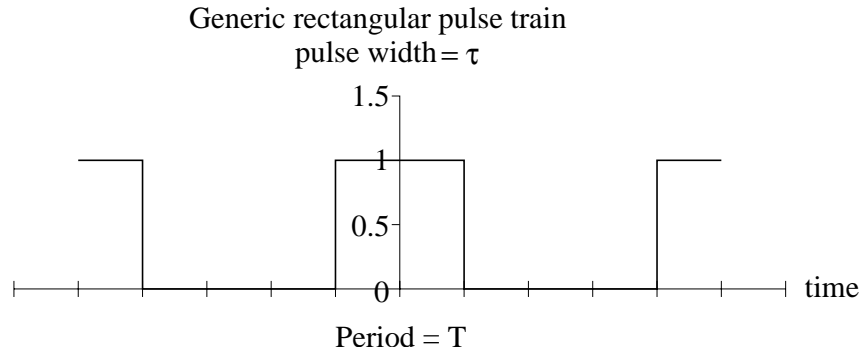
Analyzing the Data:

Next, for the pair of data points at each frequency subtract measurement taken with the filter in the circuit from that taken without the filter. This gives you the magnitude of the circuit's frequency response, its attenuation, at that frequency (the fundamental and integer multiples of it). This should be compared with those attenuations obtained from the calculations and simulation.

Tabulate your data as follows:

n	frequency $(n\omega_0)$	Measured input (dBm)	output (dBm)	$|H(n\omega_0)|$ (dBm)	Calculated $|H(n\omega_0)|$ (dB)	Simulated $|H(n\omega_0)|$ (dB)
1						
2						
3						
4						
5						
6						
7						
8						

<u>Report:</u>

First, each group should submit the work they did on the theoretical calculations and simulations. For the theoretical calculations this should include (i) your calculation of $H(\omega)$, and (ii) your MATLAB plots of $|H(\omega)|$ and $\angle H(\omega)$. For the simulation, include plots of $|H(\omega)|$ and $\angle H(\omega)$ found using the "AC sweep" analysis. The frequency axes for all plots should be in Hertz.

Then, each group should submit the measurements they took in the laboratory. This should include the table shown above and a few words commenting on "how close were your calculations." Is there a systematic difference between the measured results and those obtained from MATLAB and PSpice? Explain.

ECE-311

Appendix A to Experiment #4

Frequency Response of a Simple RC Filter

Suppose we want to find the frequency response of the filter shown in Figure 1.
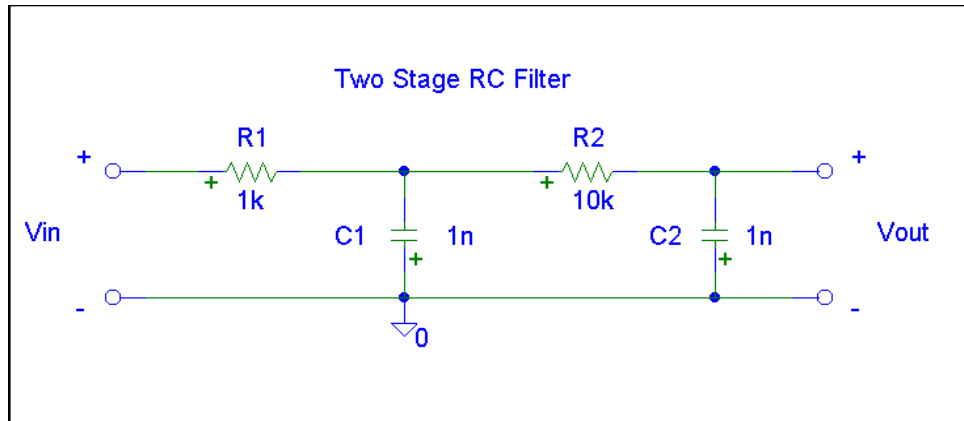
Fig. 1: Schematic diagram of a simple lowpass filter

The system's (circuit's) s-domain transfer function, $H(s)$, can be found by doing nodal analysis, assuming $V_{in}(s)$ is known, solving for $V_{out}(s)$ in terms of $V_{in}(s)$, and taking their ratio to get $H(s)$. The transfer function is

$$H(s) = \frac{1/R_1 R_2 C_1 C_2}{s^2 + \left(\frac{1}{R_2 C_2} + \frac{1}{R_1 C_1} + \frac{1}{R_2 C_1} \right) s + \frac{1}{R_1 R_2 C_1 C_2}} \quad \text{Volts/Volt.}$$

Upon substituting the given capacitor and resistor values and letting $s = j\omega$, circuit's frequency response is found (after collecting numerator and denominator into real and imaginary parts) to be

$$H(\omega) = \frac{0.1 \times 10^{12}}{[0.1 \times 10^{12} - \omega^2] + j [1.2 \times 10^6 \omega]} \quad \text{Volts/Volt.}$$

Now, suppose we want to plot the magnitude $|H(\omega)|$ and phase $\angle H(\omega)$ of the frequency response from DC to 1 MHz (2π Megaradians/second.) You could do this as it stands, but to simplify the calculation and diminish errors in the numerical computation, you should scale this function by 10^6 radians/second. In essence, we are making a change to a new variable, ω_x, which has a unit measure of 1 Megaradians/second. This means that $\omega = 10^6 \omega_x$, i.e., one unit of ω_x is 10^6 units of ω. Making this change of variable and simplifying gives a scaled the frequency response

$$H(\omega) = \frac{0.1}{[0.1 - \omega^2] + j [1.2\,\omega]} \quad \text{Volts/Volt.}$$

5

It is this function that is used in the following MATLAB code to generate the plots of Figures 2 through 4.

```
w = 0:0.05:2*pi; %Sets up a vector of frequencies.
NUM = 0.1; %Here's the numerator function.
DEN = (0.1 - w . 2) + j*1.2*w; %Here's the denominator.
Hw =NUM ./ DEN; %These are complex numbers.


plot(w,abs(Hw)),grid %Produces Figure 2
title('Magnitude of Frequency Response'),xlabel('megaradians/sec')
ylabel('Volts/Volt')
magdB = 20*log10(abs(Hw)); %Converts to decibels
plot(w,magdB),grid %Produces Figure 3
title('Magnitude of Frequency Response'),xlabel('megaradians/sec')
ylabel('decibel')


phase = angle(Hw)*180/pi; %Changes radians to degrees
plot(w,phase),grid %Produces Figure 4
title('Phase of Frequency Response'),xlabel('megaradians/sec')
ylabel('degrees')
```
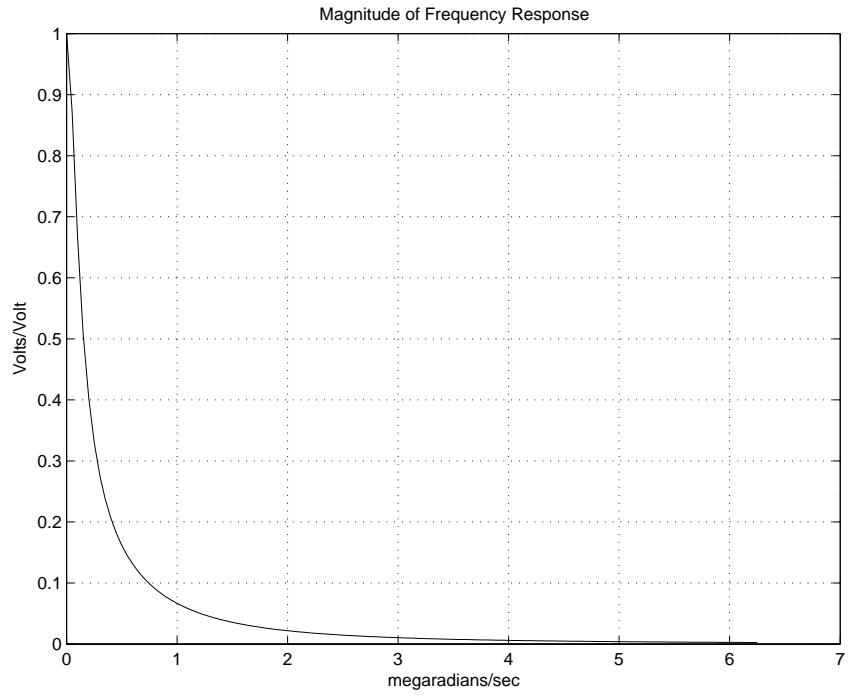
Fig. 2: Magnitude of the filter's frequency response (in Volts/Volt)
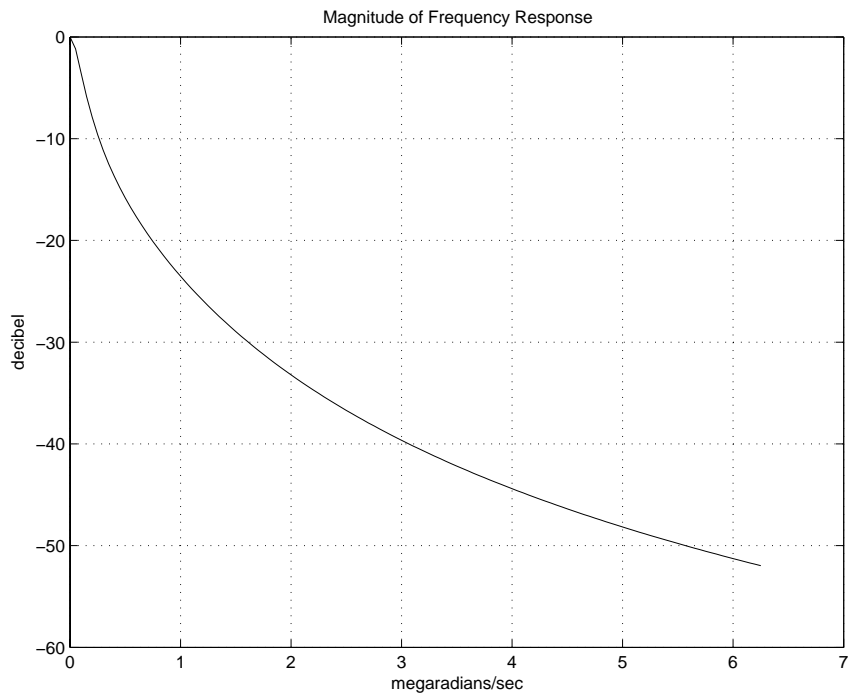
Figure 3: Magnitude of the filter's frequency response (in dB)
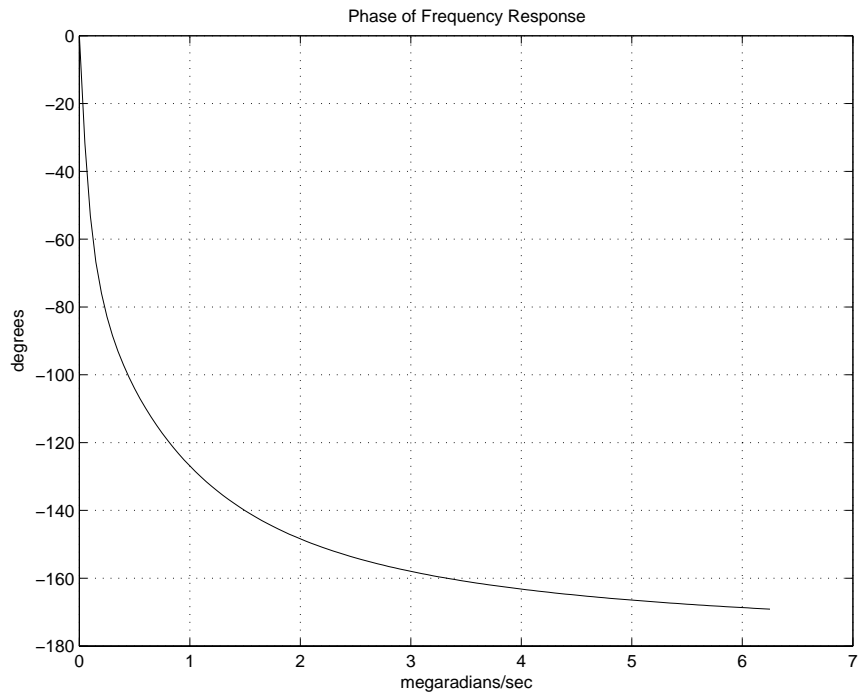
Figure 4: Phase of the filter's frequency response

ECE-311

Appendix B to Experiment #4

Information on Doing PSpice Simulations

Installing and Running PSpice on the ACCC Machines:

Version 8 of PSpice is available on the ACCC PC's, but it usually has to be installed on the server you are using. (Sometimes someone else on the same server has already installed PSpice, but this is NOT common. If it is installed, the installing program will tell you.)

To install PSpice use the following menu path to get to the installing program: Start → Programs → ACCC Software Installer → ACCC Network Installer. Click on ACCC Network Installer "menu item." Find PSpice 8 and highlight it. Click on the "install" button.

To run PSpice use the following menu path to get the schematic capture front-end of PSpice: Start → Programs → Class Apps → PSpice → Schematics. Click on the "Schematics" menu item.

Simulation of the Simple Filter of Appendix A to Obtain Its Frequency Response:

It is assumed that you have had some exposure to PSpice. For those who need a review, a good introduction and reference from which to learn is the book *Schematic Capture Using MicroSim PSpice for Windows 95/98/NT* by Marc E. Herniter (Prentice-Hall).

What follows are copies of various PSpice windows used to do the analysis for and generate plots of $H(f) - |H(f)|$ and $\angle H(f)$, the magnitude and phase of the frequency response. PSpice uses Hertz as the frequency variable; therefore, a certain amount of interpretation is needed when the output is viewed. The output is viewed using the PSpice's graphical output, *Probe*.

The following circuit was "built" using PSpice's schematic capture front-end, *Schematics*. If you are not familiar with creating circuit diagrams using *Schematics,* seek the help of your teaching assistant for some initial guidance or refer to the book referenced above.
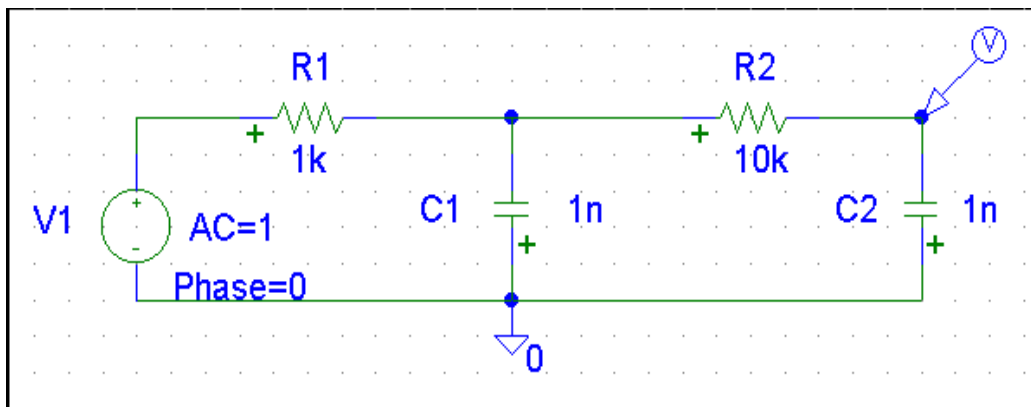
The voltage marker at the output node tells *Probe* to automatically display this node voltage after the simulation is finished. The analog ground (AGND) is needed for PSpice to do the analysis.

In order to do a frequency response simulation, you must use "VAC" as the voltage source. Note, that if you set the amplitude of the voltage source equal to one and its phase equal to zero, the plot of a particular node voltage as a function of frequency will be the frequency response (in volts/volt vs. Hertz) of the circuit, considering the voltage source as the input and the node as the output. Next, you must enable the "AC Sweep" from the Analysis → Setup menu and choose the frequency range and intervals.
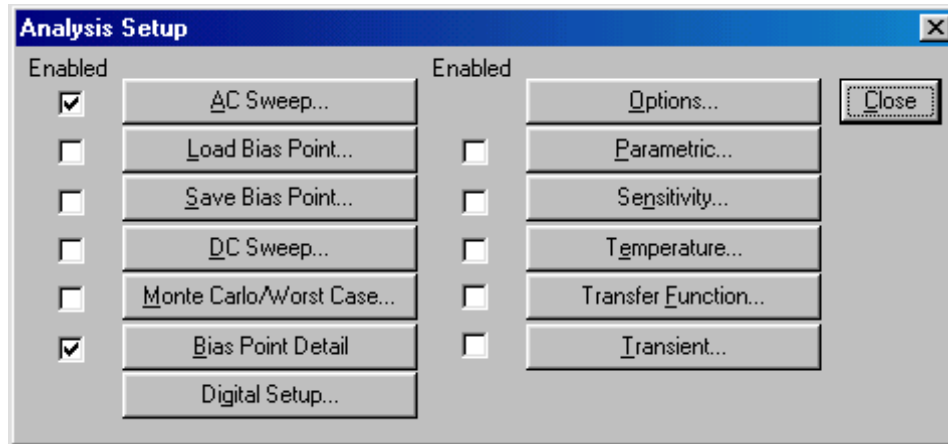
The following dialog box is activated with "AC Sweep" button, above. You can choose the type of sweep, the start frequency, the stop frequency, and the number of data points to be calculated. Frequencies are in Hertz. Usually, for a narrow band of frequencies, a linear sweep is used. For a broad band of frequencies, a decade sweep is used. A certain amount of "playing around" is encouraged in order to make your output "look pretty." If your output looks jagged, you have not chosen enough points.
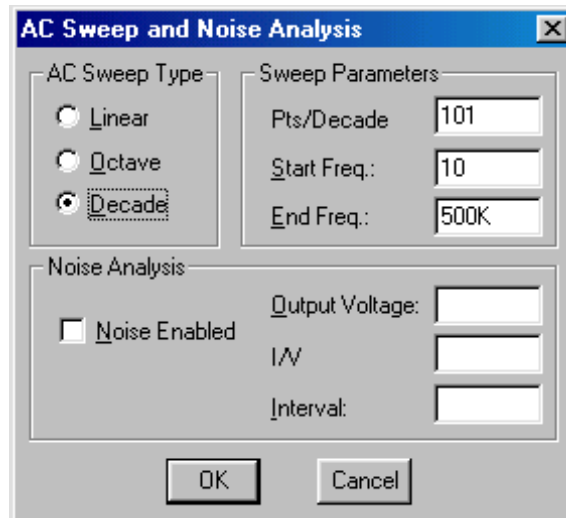
Once you have correctly "built" the circuit, with the proper AC input source and with the output node identified with a voltage marker, and once you have enabled the "AC Sweep" and chosen a frequency range, you are ready to do the simulation. This is activated by clicking on the menu path: Analysis → Simulate.

The result of the simulation is plotted by *Probe* and is shown below. Because we chose the input voltage amplitude to be one volt, this trace is the magnitude of the frequency response.
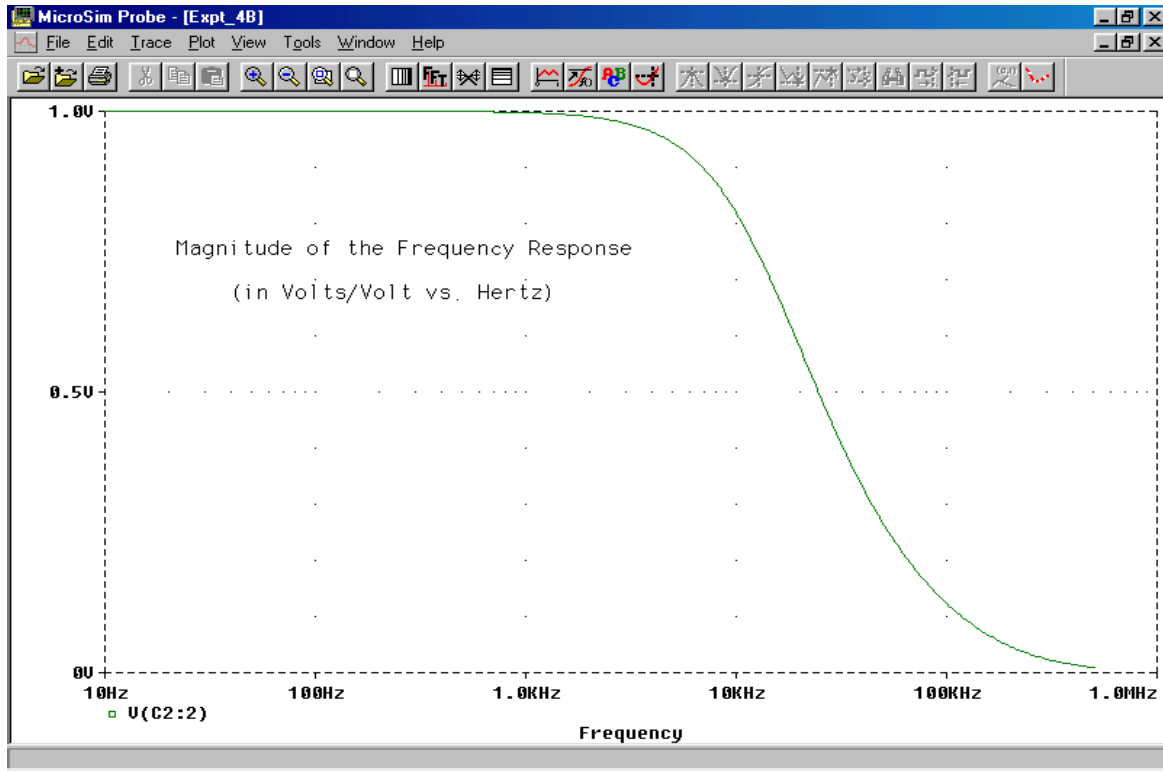
To get the plot in dB you double click on output voltage variable label, V(C2:2), located in the lower left-hand corner of the *Probe* window. When the "Modify Trace" window appears, in the Trace Expression box change the label to Vdb(C2:2). This window and the resulting plot change are shown on the next two figures.

To get the phase plot you double click on output voltage variable label, V(C2:2), located in the lower left-hand corner of the *Probe* window. When the "Modify Trace" window appears, in the "Trace Expression" box change the label to Vp(C2:2). This window and the resulting plot change are shown after next two figures.

The plots on the next two pages also show cursors, which can be used to identify the coordinates of points on the trace. The cursors are activated by the "Cursor Toggle" button on the toolbar. The A1 cursor is controlled by the left mouse button and the A2 by the right.

Modify Trace

Simulation Output Variables

`*`

| Frequency |
| I(C1) |
| I(C2) |
| I(R1) |
| I(R2) |
| I(V1) |
| V(0) |
| V(C1:1) |
| V(C1:2) |
| V(C2:1) |
| V(C2:2) |
| V(R1:1) |
| V(R1:2) |
| V(R2:1) |
| V(R2:2) |
| V(V1:+) |
| V(V1:-) |
| V($N_0001) |
| V($N_0002) |
| V($N_0003) |
| V1(R1) |
| V1(R2) |
| V1(V1) |
| V2(C1) |

☑ **A**nalog
☐ **D**igital
☑ **V**oltages
☑ **C**urrents
☐ **N**oise (V²/Hz)
☑ **A**lias **N**ames
☐ **S**ubcircuit Nodes

27 variables listed

Full List

Functions or Macros

Analog Operators and Functions ▼

| # |
| () |
| * |
+
/
@
ABS( )
ARCTAN( )
ATAN( )
AVG( )
AVGX(,)
COS( )
D( )
DB( )
ENVMAX(,)
ENVMIN(,)
EXP( )
G( )
IMG( )
LOG( )
LOG10( )
M( )
MAX( )

Trace Expression: Vdb(C2:2)

[**O**K] [Cancel] [**H**elp]

MicroSim Probe - [Expt_4B]

File Edit Trace Plot View Tools Window Help

Magnitude of the Frequency Response

(in Decibels vs. Hertz)

```
Probe Cursor
A1 =   10.158K,   -1.7807
A2 = 100.000K,  -18.165
dif= -89.842K,   16.384
```

Frequency
(10Hz, 100Hz, 1.0KHz, 10KHz, 100KHz, 1.0MHz)

▫ Vdb(C2:2)

For Help, press F1

12

Modify Trace

Simulation Output Variables

`*`

| Frequency |
| I(C1) |
| I(C2) |
| I(R1) |
| I(R2) |
| I(V1) |
| V(0) |
| V(C1:1) |
| V(C1:2) |
| V(C2:1) |
| V(C2:2) |
| V(R1:1) |
| V(R1:2) |
| V(R2:1) |
| V(R2:2) |
| V(V1:+) |
| V(V1:-) |
| V($N_0001) |
| V($N_0002) |
| V($N_0003) |
| V1(R1) |
| V1(R2) |
| V1(V1) |
| V2(C1) |

☑ **A**nalog
☐ **D**igital
☑ **V**oltages
☑ **C**urrents
☐ **N**oise (V²/Hz)
☑ Alias **N**ames
☐ **S**ubcircuit Nodes

27 variables listed

Full List

Functions or Macros

Analog Operators and Functions ▼

| # |
| () |
| * |
+
/
@
ABS( )
ARCTAN( )
ATAN( )
AVG( )
AVGX(,)
COS( )
D( )
DB( )
ENVMAX(,)
ENVMIN(,)
EXP( )
G( )
IMG( )
LOG( )
LOG10( )
M( )
MAX( )

Trace Expression: Vp(C2:2)   [OK] [Cancel] [Help]

MicroSim Probe - [Expt_4B]

File   Edit   Trace   Plot   View   Tools   Window   Help

Phase of the Frequency Response
(in Degrees vs. Hertz)

Probe Cursor
A1 =   10.000K,   -38.131
A2 = 101.623K,   -111.880
dif= -91.623K,   73.749

□ Vp(C2:2)

Frequency

For Help, press F1

Due date: Next lab session to your TA.

13

ECE 311 Communication Engineering

Lab Assignment #5

Examination of DSB-LC (Broadcast AM) Spectra

Objective:

In this project, you will investigate the broadcast form of amplitude modulation (AM), double sideband large carrier (DSB-LC). To do this you will use MATLAB and Pspice, with their FFT capabilities, to investigate the expected time-domain waveform and the expected frequency domain spectra of single-tone modulated AM signals. In the laboratory, you will use the function generator to produce these AM signals and to examine their time domain signals and its spectra.

Introduction:

Read Sections 4.1 through 4.3 in the text for the theory underlying DSB-LC (AM) modulation. The settings needed to produce these AM signals in the laboratory are discussed in the following sections of the function generator's *User's Guide*: Chapter 2, pp. 41–43; Chapter 3, pp. 71–75; Chapter 7, pp. 283–288.

Review the material from experiment #3 on interpreting the FFT. An appendix is attached that discusses the PSpice simulation of AM signals.

Pre-Lab:

Part I

Using MATLAB, plot the time domain representation and the magnitude of its Fourier transform, obtained using the FFT routine, for the two traditional AM signals (DSB-LC), shown below,

$$\varphi_{am}(t) = [1 + \mu \cos(8\pi t)]2\cos(64\pi t), \quad \mu = \{0.2, 0.8\}.$$

The horizontal scale for the FFT output should be in Hertz. Answer the following questions for both signals.

a) Use the time-domain AM signal to find "μ," its modulation index. Is it right?

b) Compare the FFT output to what is theoretically expected.

   i) What were the frequencies of the carrier and sidebands?
   ii) What is the total power in the signal (in dBm)?
   iii) What is the power in the carrier and sidebands (dBm)?
   iv) What are the relative powers of the sidebands with respect to the carrier (in dB)?

Note, that the spectra of these signals will be "delta functions", your choice of N, T_s , and T_o must be made carefully so as to avoid the "picket fence" effect associated with the DFT.

Part II

Using Pspice, simulate the DSB-LC signal, shown below,

$$\varphi_{am}(t) = [1 + \mu \cos(2\pi f_m t)] 2 \cos(2\pi f_c t), \quad \mu = \{0.2, 0.8\}$$

where $f_m = 20$ kHz and $f_c = 1$ MHz.

Plot the time domain output and its FFT-generated magnitude spectra.

a) Use the time-domain AM signal to find "μ," its modulation index. Is it right?

  i) Compare the FFT output to what is theoretically expected.

  ii) What were the frequencies of the carrier and sidebands?

  iii) What is the total power in the signal (in dBm)?

  iv) What is the power in the carrier and sidebands (dBm)?

  v) What are the relative powers of the sidebands with respect to the carrier (in dB)?

Task:

Use the function generator to generate two single-tone modulated AM waveforms with the specifications given below. Use the "Front-Panel Operation" outlined in the *User's Guide* referenced above. For each AM signal, do the following.

a) Examine the output using the oscilloscope. Sketch the waveform and use this information to find the index of modulation. Is it what you expected?

b) Next use the spectrum analyzer to find the power in the signal – the carrier and the sidebands. Compare these to the calculated values.

  i) Are the frequencies what you expected?

  ii) Are the powers what you expected?

  iii) Are the relative values – sidebands with respect to the carrier what you expected?

Signal Specifications:

    Carrier: Frequency = 1 MHz, Amplitude = 4 volts (p-p)
    Modulating Tone: Sinusoid at 20 kHz
    Index (Depth) of Modulation: 20% and 80%

Data Analysis:

Determine the percentage error for the two indices of modulation you calculated from the time domain AM signals that you displayed in each of the three methods you used – MATLAB, PSpice, and Function Generator – to generate the waveform.

Determine the efficiency, n, of the AM transmission for the two indices of modulation you used for this experiment. Calculate a theoretically expected efficiency and the actual efficiency from the sideband and carrier power data you obtained in the lab or from the simulations. The efficiency formulas are as follows:

$$\eta_{\text{actual}} = \frac{P_S}{P_S + P_C} 100\%, \quad \eta_{\text{theoretical}} = \frac{\mu^2}{2 + \mu^2} 100\%$$

where P_c = carrier power in Watts and P_s = total sideband power in Watts.

Report:

In your report, answer the questions concerning the comparison of the calculated and measured values of indices of modulation, powers of the carrier and sidebands, and the frequencies of the carrier and sidebands.

Summarize your analyzed data in following table:

		Expected	MATLAB	PSpice	Laboratory Measurements
	P_s/P_c (in dB) *				
$\mu = 0.2$	η				
	% error in μ				
	P_s/P_c (in dB)				
$\mu = 0.8$	η				
	% error in μ				

* Here P_s is the power in one sideband.

Appendix A to Experiment #5

Simulating Linear Modulation in PSpice
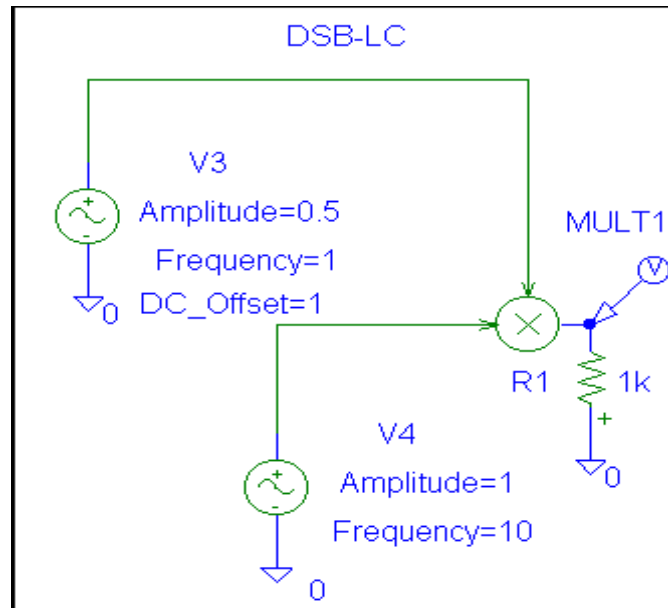
The above schematic can be "built" in PSpice. It uses a function block, MULT, which just does the time domain multiplication of two input signals. MULT can be found in the ABM library of PSpice. The voltage sources are VSIN with the attributes assigned as indicated above. The circuit needs a load resistor and an analog ground to work.

"Transient" simulation is done (with Print Step = 20m and Final Time = 2). This is found under the "Analysis Setup" menu. The output, taken across the 1 kΩ resistor, is automatically displayed in *Probe*. Its FFT may be obtained by clicking on the "FFT Icon" on the *Probe* toolbar.

The mathematical expression for the output voltage is as follows:

$$V_{out}(t) = [1 + 0.5\sin(2\pi t)]\sin(20\pi t)$$

$$V_{out}(t) = 1\sin(20\pi t) + 0.25\cos(18\pi t) + 0.25\sin(22\pi t).$$

Time-domain output is shown in the next figure.
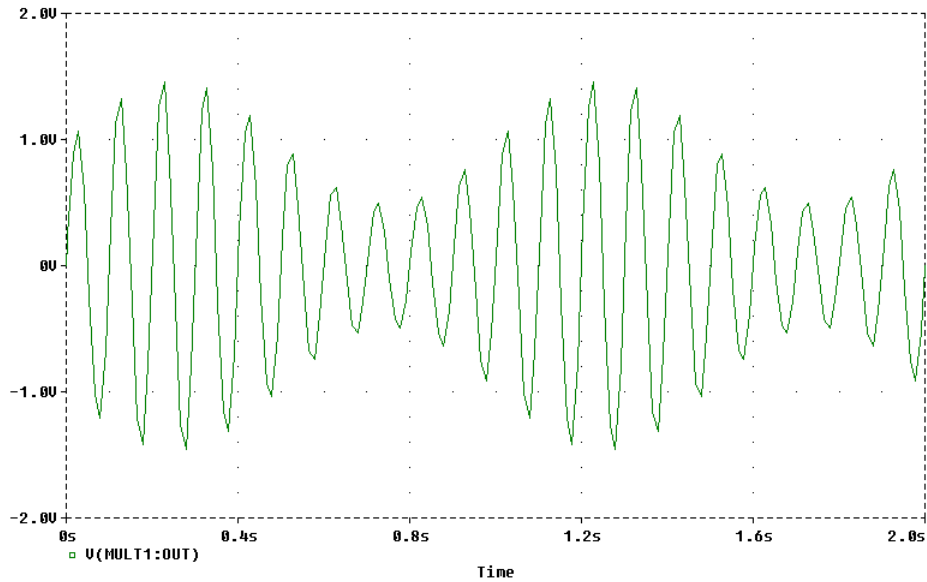
The FFT of the above signal is shown next (Here we have "zoomed-in" on the important part of the spectra).
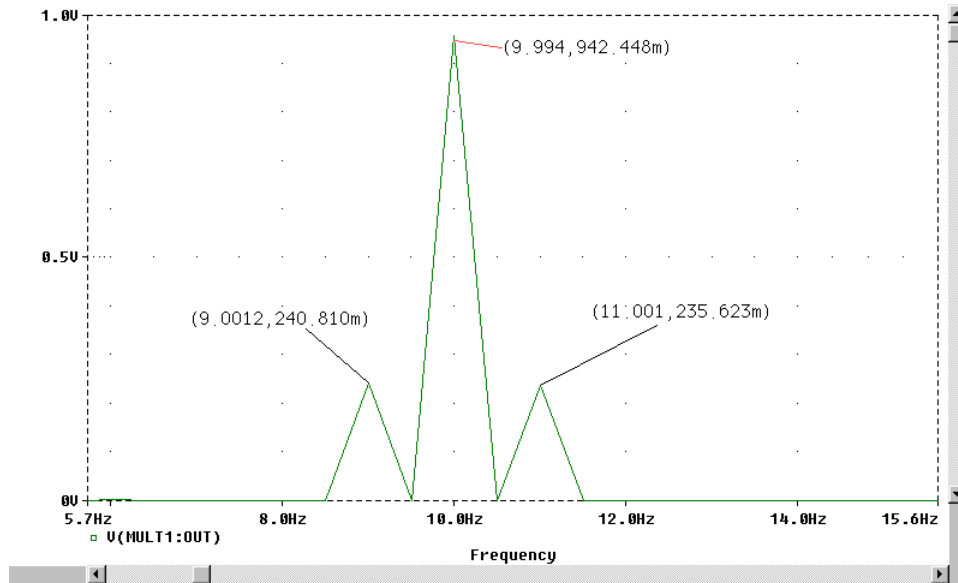
Compare the magnitude of the FFT coefficients and the last $V_{\text{out}}(t)$ expression given above. Note that like the oscilloscope's FFT module, the PSpice version calculates the magnitude of the compact trigonometric Fourier Series coefficients. No phase information is given.

Due date: Next lab session to your TA.

ECE-311
Experiment #7
The Detection of AM Signals

Objective:

The purpose of this experiment is to investigate the two major forms of AM demodulation, coherent (synchronous) detection and non-coherent detection. You will generate a single-tone modulated AM signal using the function generator and use these two methods to recover the modulating tone.

These two methods use different circuits which you will be asked to design and build. The coherent detection method uses the balanced modulator circuit from the last laboratory experiment and a simple RC low-pass filter that you will design. The non-coherent detection method uses an envelope detector circuit, which you will design.

Introduction:

Read that part of section 4.3 in the text, which discusses AM demodulation. Focus your attention on the design of an envelope detector detailed in Example 4.6. Problem 4.3-1 shows the block diagram for a synchronous AM demodulator. If you have not, as of yet, done this problem, do it and compare your solution to the answer book's.

Pre-Lab:

For this experiment, you must design a envelope detector for non-coherent detection method. Also, you must design the circuit for the low-pass filter block of the synchronous demodulator. Do this prior to lab. For this design you should find actual component values (numbers), not just the formulas for them. Note, that you are trying to detect an AM signal modulated ($\mu = 0.5$) with a 2 KHz tone.

Simulate both your designs using PSpice.

Task:

Part One (Synchronous Detection)
Build and test the synchronous detection circuit shown below. There are two inputs for the balanced modulator, the AM signal and the local oscillator signal. Their specification is given below. Use one function generator to produce the AM signal and the other to produce the local oscillator signal. In order for synchronous detection to work properly these two generators must be *synchronized*! As shown in figure 2, this can be done via a connection on the back of the two function generators. Your teaching assistant will show you how to do this.

Connect the output of the demodulator to the oscilloscope (AC coupling with high input impedance). Did you get the 2 kHz modulating sinusoid at the output?

AM signal: $f_C = 1$ MHz, $f_m = 2$ KHz (sinusoid) $\mu = 0.5$ $V_{RMS} < 300$ mV

Local Oscillator: $f_{LO} = 1$ MHz $V_{RMS} < 60$ mV

Figure 1 – Synchronous Detection

Vlo

5V

0.1uF

| 8 | 7 | 6 | 5 |

SA612AN

| 1 | 2 | 3 | 4 |

0.1uF

0.1uF

Vin

R

Vout

C
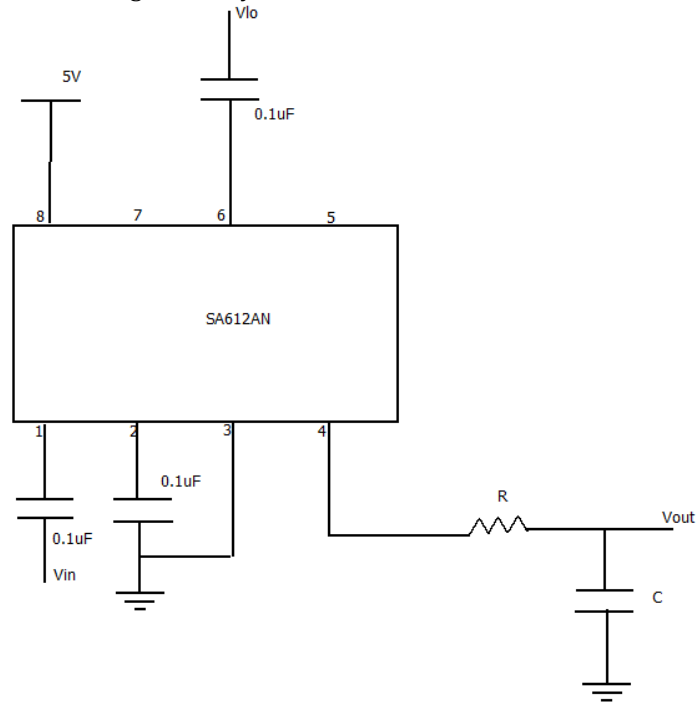
Figure 2 – Phase Locking Function Generators

Reference Out

Function Generator #1

Sync   Out

External Ref. In

Function Generator #2

Out

$V_S(t)$

$V_{LO}(t)$

Oscilloscope

External Trigger
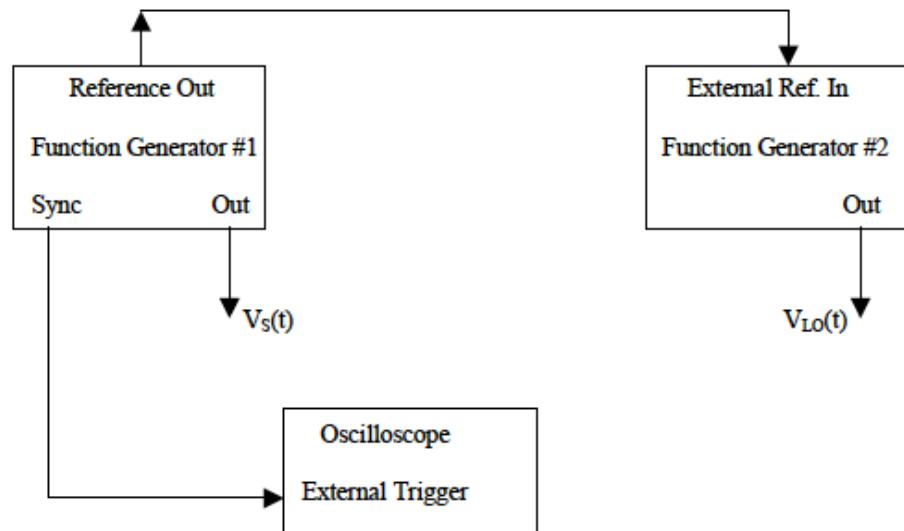
Part Two (Envelope Detector)

Build and test the envelope detector shown in figure 3. In the pre-lab, you were to determine the values
(i.e. numbers) for R and C. Here you don't need the synchronized local oscillator. The only input needed
is the AM signal. Use the same specifications as *Part One*, EXCEPT adjust the amplitude of the AM signal
so that its "envelope valley voltage" is *greater than* the diode's "cut-in" voltage (0.7 volts). Did you get
the 2 kHz modulating sinusoid at the output?

Figure 3 – Envelope Detector
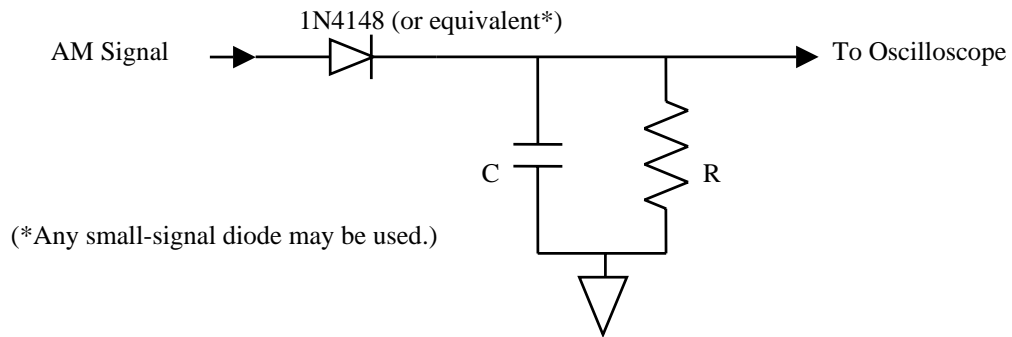
Both Parts

For both parts, have your teaching assistant verify that your detectors "worked." There may be a certain
amount of "tweaking" (adjusting component values) needed to make the detectors operational.

Data Analysis:

Report:

In your report, include the following:
  • The design component values, from the pre-lab calculations, for both detectors,
  • The simulations used to "test" your designs, and
  • The actual design values you used.
Also, include a brief discussion of the advantages and disadvantages of both methods of detection.

ECE-311
Experiment #8

Examination of a Single-Tone FM Signal's Spectra

Objective:

The purpose of this experiment is to examine the spectra of a single-tone FM signal.

In the pre-lab portion of this exercise you will use MATLAB and PSpice to simulate a single-tone FM signals and to use the internal FFT algorithms to examine the spectra of these signals. Three different modulation indices, β, are used giving examples of narrowband and wideband FM, along with an intermediate case.

In the laboratory portion of this exercise you will generate a single-tone FM signal using the function generator, examine its frequency domain properties using the spectrum analyzer, and compare your data with those calculated in the pre-lab.

Introduction:

Read sections 5.1 through 5.3 in the text to find the theoretically expected spectra for these signals. Appendix A contains a discussion of the theory of single-tone FM and Bessel functions. Read the appendix B to learn how to simulate a single-tone frequency modulated signal using Pspice. Also, read the information from the function generator's *User's Guide* (Chapter 3) to learn how to produce a single-tone FM signal and to adjust its carrier frequency, its modulating frequency, and its frequency deviation.

Pre-lab:

Simulate the single-tone modulated signal, $\varphi_{FM}(t)$, given below for the three listed values of β, the FM index of modulation. Examine the spectra using the built-in FFT routines of MATLAB and PSpice. Remember that MATLAB calculates the "D_n's" whereas PSpice determines the "C_n's."

$$\varphi_{FM}(t) = A \cos [\ \omega_c t \ + \ \beta \sin \omega_m t \ ] \qquad\qquad \beta = \{0.1, \ 1, \ 10\}$$
$$A = 1$$

Use the following values for the carrier and modulating frequencies for the two different types of simulations

	Frequencies in Hertz	
	Carrier	Modulating Tone
MATLAB	32	2
PSpice	1.0 M	5 K

For each index of modulation, determine the power in the sidebands from the output from the two simulation methods. Next, find the theoretically expected sideband powers. (Here you need to use a table of Bessel functions and MATLAB's built-in function "besselj". Type "help besselj" for more information.) In all cases, express the powers in dBm. (Note, that in MATLAB you can do the power calculations in a relatively straightforward manner once you have the FFT output. Unfortunately, it is not so easy with PSpice where you have to do it "by hand" once you have obtained the FFT in *Probe*.)

Task:

Use the function generator to generate a single-tone modulated FM waveform with the specifications given below. Use the spectrum analyzer to find the power in the signal — the carrier and the sidebands — (in dBm).

Signal Specifications:

Carrier:	Frequency = 1 MHz	Amplitude = 2 volts (p-p)
Modulating Tone:	Sinusoid at 5 kHz	
Index of Modulation:	$\beta = \{0.1, 1, 10\}$	

(Note that the function generator does NOT have a direct adjustment to set "β." You must use "peak frequency deviation." setting.)

Data Analysis:

Normalize your data using the total power available in an FM signal as a reference. Recall that the total power in an FM signal is constant ($= A^2/2R$). For the signals in the pre-lab, R = 1 and P_{TOTAL} = 0.5 W (27 dBm). Thus, for these pre-lab signals, the normalized power (in dB) is the calculated power (in dBm) minus 27. For the signals measured in the laboratory, R = 50 and P_{TOTAL} = 10 mW (10 dBm). Thus, for these laboratory signals, the normalized power (in dB) is the measured power (in dBm) minus 10.

Choose those sidebands with powers that are greater than or equal to 1% of the total power in the FM signal. Use these sidebands to determine the signal's bandwidth. Note, that those sidebands with relative powers greater than –20 dB meet the 1% criteria for inclusion in the bandwidth calculation. (Verify this statement for yourself. It could be a good test question!!)

Report:

On a separate sheet of paper for each index of modulation, tabulate your above calculations using the following template.

$\beta =$		Relative Powers (in dB)			
n	Frequency (Hz)	Theory	PSpice	MATLAB	Measured

•
•
•

BW =	BW =	BW =	BW =	BW =
(Theory)	(PSpice)	(MATLAB)	(Measured)	(Carson's Rule)

Here the bandwidth (BW) you state should be normalized to the modulating frequency – i.e.,
$$BW = \text{"real" bandwidth} \div f_m$$

Also, accompanying each sheet include another sheet with plots of the MATLAB and PSpice FFT outputs, zoomed-in as appropriate.

Bessel Functions and Single-Tone FM

A single-tone FM signal may expanded in a Fourier series as follows:

$$\varphi_{FM}(t) = A \cos [\omega_c t + \beta \sin \omega_m t]$$

$$\varphi_{FM}(t) = A \sum_{n=-\infty}^{n=+\infty} J_n(\beta) \cos[(\omega_c + n\omega_m)t]$$

where the $J_n(x)$'s are a discrete infinite set, indexed by the integer "n," of real-valued functions of a real variable. These are called the Bessel Functions of the First Kind. Bessel functions occur in a number of applications in mathematical physics. Formulas for evaluating the functions have been developed and can be found in any standard mathematical handbook. A useful tabulation of a number of Bessel Function values at various values of the argument is attached. This tabulation can be used for negative indices by using the following property of the Bessel Functions:

$$J_n(x) = (-1)^n J_{-n}(x) \quad .$$

In reference to the single-tone FM signal, it has been represented as an infinite number of sinusoids centered at the carrier frequency (f_c), with a frequency spacing determined by the modulating frequency (f_m). The index "n" determines the frequency of the sideband, $f_{sideband} = f_c + nf_m$ (n = 0 is the carrier). The amplitude of the signal component at that frequency is determined by the nth order Bessel Function value where the argument is the signal's <u>index of modulation</u>, β. Thus, we may interpret the $|AJ_n(\beta)|$'s as the C_n's of the compact trigonometric form of the Fourier series for $\varphi_{FM}(t)$.

As an example, suppose a unit amplitude 100 kHz (f_c) sinusoid is frequency modulated with a 5 kHz (f_m) tone and an index of modulation of 8. (Note, this signal is simulated using PSpice in Appendix B.) What is the power in the signal at 80 kHz?

Since $f_{sideband} = 80$ kHz and $f_{sideband} = f_c + nf_m$, then n = –4. The amplitude of the sinusoid at 80 kHz is
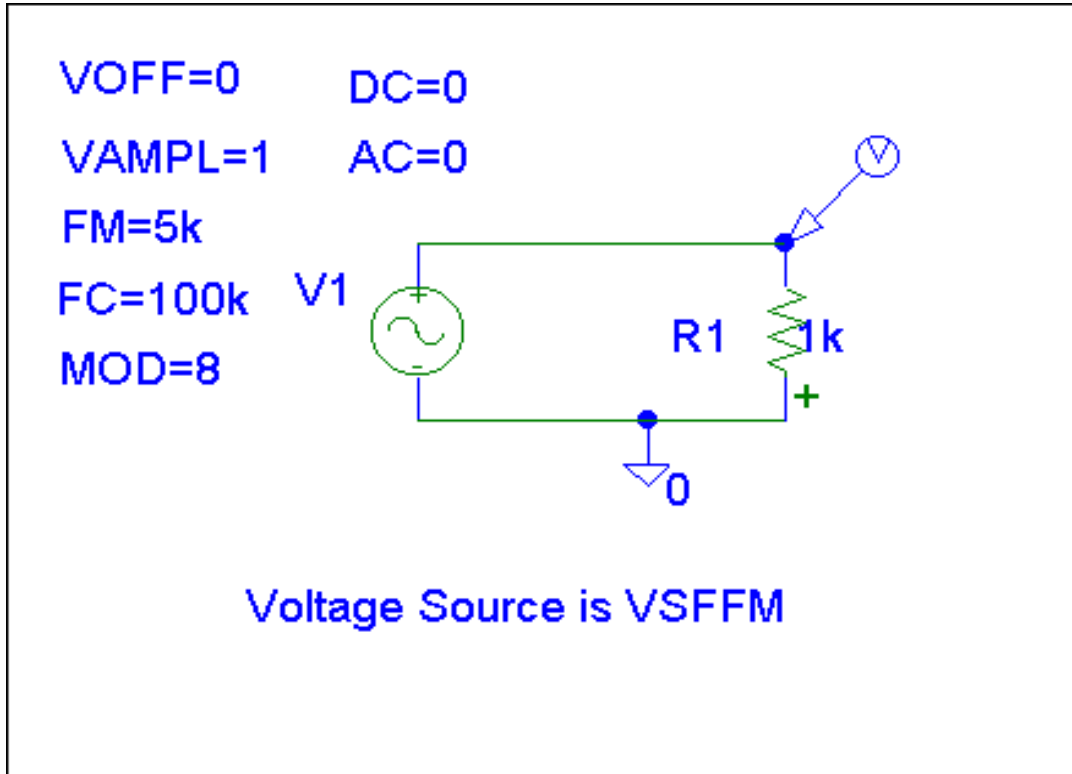
$$|1 \cdot J_{-4}(8)| = 0.110$$

This compares favorable with the spectra plot of Appendix B. To calculate the power associated with the signal at 80 kHz, we use $A^2/2$ where is "A" is the amplitude of the sinusoidal component of the signal at 80 kHz – i.e., 0.110. Thus, the power is 6.05 mW (7.82 dBm).

Bessel Functions of the First Kind, $J_n(x)$

x=	n =	0	1	2	3	4	5	6	7	8	9	10
0.0		1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.2		0.99	0.10	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.4		0.96	0.20	0.02	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.6		0.91	0.29	0.04	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.8		0.85	0.37	0.08	0.01	0.00	0.00	0.00	0.00	0.00	0.00	0.00
1.0		0.77	0.44	0.11	0.02	0.00	0.00	0.00	0.00	0.00	0.00	0.00
1.2		0.67	0.50	0.16	0.03	0.01	0.00	0.00	0.00	0.00	0.00	0.00
1.4		0.57	0.54	0.21	0.05	0.01	0.00	0.00	0.00	0.00	0.00	0.00
1.6		0.46	0.57	0.26	0.07	0.01	0.00	0.00	0.00	0.00	0.00	0.00
1.8		0.34	0.58	0.31	0.10	0.02	0.00	0.00	0.00	0.00	0.00	0.00
2.0		0.22	0.58	0.35	0.13	0.03	0.01	0.00	0.00	0.00	0.00	0.00
2.2		0.11	0.56	0.40	0.16	0.05	0.01	0.00	0.00	0.00	0.00	0.00
2.4		0.00	0.52	0.43	0.20	0.06	0.02	0.00	0.00	0.00	0.00	0.00
2.6		-0.10	0.47	0.46	0.24	0.08	0.02	0.01	0.00	0.00	0.00	0.00
2.8		-0.19	0.41	0.48	0.27	0.11	0.03	0.01	0.00	0.00	0.00	0.00
3.0		-0.26	0.34	0.49	0.31	0.13	0.04	0.01	0.00	0.00	0.00	0.00
3.2		-0.32	0.26	0.48	0.34	0.16	0.06	0.02	0.00	0.00	0.00	0.00
3.4		-0.36	0.18	0.47	0.37	0.19	0.07	0.02	0.01	0.00	0.00	0.00
3.6		-0.39	0.10	0.44	0.40	0.22	0.09	0.03	0.01	0.00	0.00	0.00
3.8		-0.40	0.01	0.41	0.42	0.25	0.11	0.04	0.01	0.00	0.00	0.00
4.0		-0.40	-0.07	0.36	0.43	0.28	0.13	0.05	0.02	0.00	0.00	0.00
4.2		-0.38	-0.14	0.31	0.43	0.31	0.16	0.06	0.02	0.01	0.00	0.00
4.4		-0.34	-0.20	0.25	0.43	0.34	0.18	0.08	0.03	0.01	0.00	0.00
4.6		-0.30	-0.26	0.18	0.42	0.36	0.21	0.09	0.03	0.01	0.00	0.00
4.8		-0.24	-0.30	0.12	0.40	0.38	0.23	0.11	0.04	0.01	0.00	0.00
5.0		-0.18	-0.33	0.05	0.36	0.39	0.26	0.13	0.05	0.02	0.01	0.00
5.2		-0.11	-0.34	-0.02	0.33	0.40	0.29	0.15	0.07	0.02	0.01	0.00
5.4		-0.04	-0.35	-0.09	0.28	0.40	0.31	0.18	0.08	0.03	0.01	0.00
5.6		0.03	-0.33	-0.15	0.23	0.39	0.33	0.20	0.09	0.04	0.01	0.00
5.8		0.09	-0.31	-0.20	0.17	0.38	0.35	0.22	0.11	0.05	0.02	0.01
6.0		0.15	-0.28	-0.24	0.11	0.36	0.36	0.25	0.13	0.06	0.02	0.01
6.2		0.20	-0.23	-0.28	0.05	0.33	0.37	0.27	0.15	0.07	0.03	0.01
6.4		0.24	-0.18	-0.30	-0.01	0.29	0.37	0.29	0.17	0.08	0.03	0.01
6.6		0.27	-0.12	-0.31	-0.06	0.25	0.37	0.31	0.19	0.10	0.04	0.01
6.8		0.29	-0.07	-0.31	-0.12	0.21	0.36	0.33	0.21	0.11	0.05	0.02
7.0		0.30	-0.00	-0.30	-0.17	0.16	0.35	0.34	0.23	0.13	0.06	0.02
7.2		0.30	0.05	-0.28	-0.21	0.11	0.33	0.35	0.25	0.15	0.07	0.03
7.4		0.28	0.11	-0.25	-0.24	0.05	0.30	0.35	0.27	0.16	0.08	0.04
7.6		0.25	0.16	-0.21	-0.27	-0.00	0.27	0.35	0.29	0.18	0.10	0.04
7.8		0.22	0.20	-0.16	-0.29	-0.06	0.23	0.35	0.31	0.20	0.11	0.05
8.0		0.17	0.23	-0.11	-0.29	-0.11	0.19	0.34	0.32	0.22	0.13	0.06
8.2		0.12	0.26	-0.06	-0.29	-0.15	0.14	0.32	0.33	0.24	0.14	0.07
8.4		0.07	0.27	-0.00	-0.27	-0.19	0.09	0.30	0.34	0.26	0.16	0.08
8.6		0.01	0.27	0.05	-0.25	-0.22	0.04	0.27	0.34	0.28	0.18	0.10
8.8		-0.04	0.26	0.10	-0.22	-0.25	-0.01	0.24	0.34	0.29	0.20	0.11
9.0		-0.09	0.25	0.14	-0.18	-0.27	-0.06	0.20	0.33	0.31	0.21	0.12
9.2		-0.14	0.22	0.18	-0.14	-0.27	-0.10	0.16	0.31	0.31	0.23	0.14
9.4		-0.18	0.18	0.22	-0.09	-0.27	-0.14	0.12	0.30	0.32	0.25	0.16
9.6		-0.21	0.14	0.24	-0.04	-0.26	-0.18	0.08	0.27	0.32	0.27	0.17
9.8		-0.23	0.09	0.25	0.01	-0.25	-0.21	0.03	0.25	0.32	0.28	0.19
10.0		-0.25	0.04	0.25	0.06	-0.22	-0.23	-0.01	0.22	0.32	0.29	0.21

Generating an FM signal in PSpice

VOFF=0   DC=0
VAMPL=1   AC=0
FM=5k
FC=100k V1
MOD=8

R1 1k

0

Voltage Source is VSFFM

The above circuit generates the signal given as follows:

$$v_1(t) = V_{off} + V_{ampl} \sin(2\pi f_c t + MOD \sin(2\pi f_m t)) \quad .$$

The plot of $v_1(t)$ is given below. Using the *Probe* FFT button (and zooming-in) gives the spectra plot shown. Some comments are in order.

First, the index of modulation (β) for this signal is eight (8) and its amplitude is one (1). This means that the amplitude of the FM signal's component at the carrier frequency (100 kHz) is given by

A $J_0(\beta)$ = 1 $J_0(8)$ = 0.17

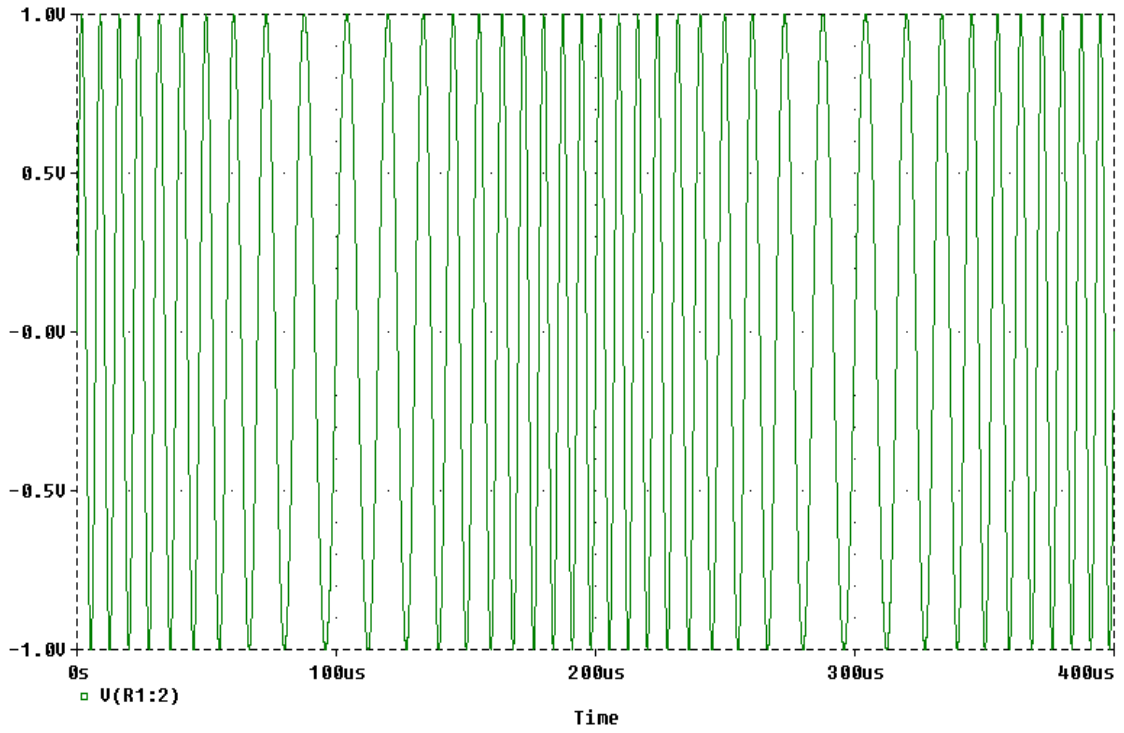where J_0 is the zeroth order Bessel Function of the First Kind.

Check out the FFT output. Its RIGHT ON!!

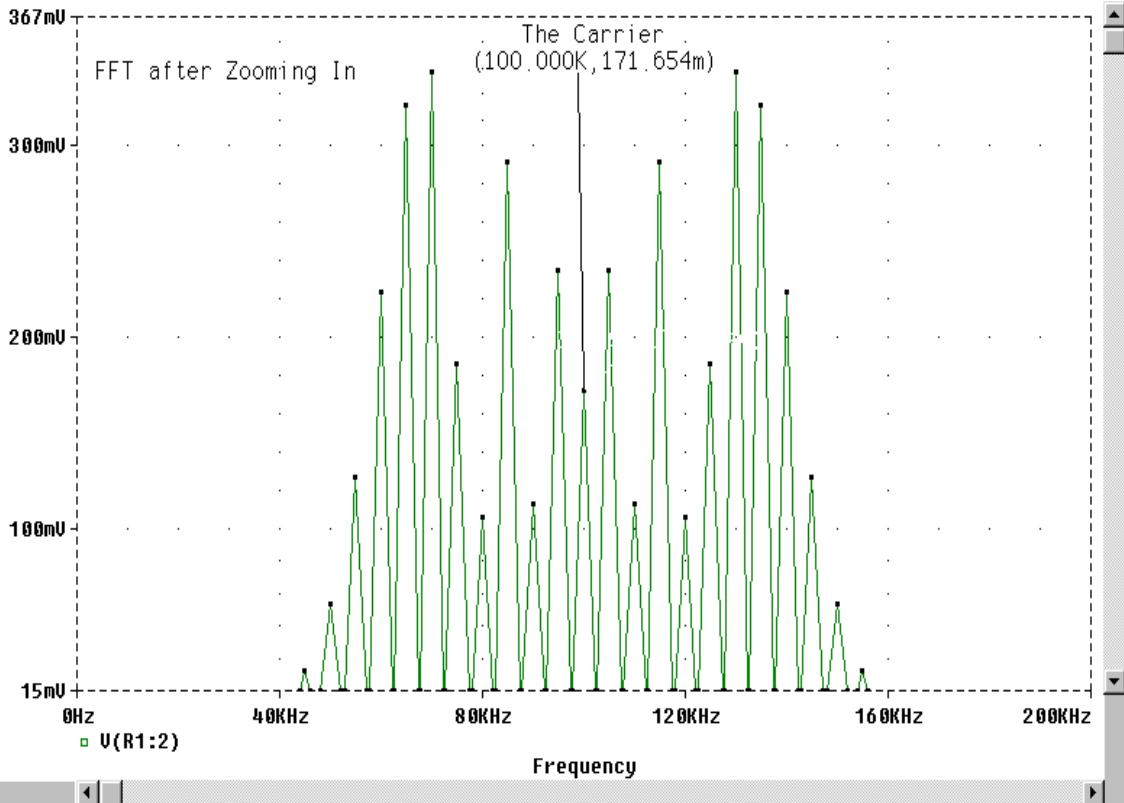*Second*, the bandwidth by Carson's rule is

BW = 2 f_m (β + 1) = 90 kHz.

A "rough" estimate from the FFT plot gives about 100 kHz. "Close enough for government work."

TIME DOMAIN PLOT

1.0V

0.5V

-0.0V

-0.5V

-1.0V

0s   100us   200us   300us   400us

□ V(R1:2)

Time

SPECTRA INFORMATION

367mV

FFT after Zooming In   The Carrier
                                  (100.000K,171.654m)

300mV

200mV

100mV

15mV

0Hz   40KHz   80KHz   120KHz   160KHz   200KHz

□ V(R1:2)

Frequency

ECE 311 Communication Engineering

Experiment #9

Generation of an FM Signal using a VCO

Objective:

In this laboratory exercise you will generate a single-tone modulated FM signal using the voltage-controlled oscillator (VCO) in a phase-locked loop (PLL). You will examine the frequency domain properties of the signal using the spectrum analyzer and compare your results with those you expected to get.

In order to use the VCO as an FM generator it will be necessary to do some circuit assembly, testing, and characterization.

Introduction:

Read Section 5.3 in the textbook, especially that part on "Direct Generation."

The micropower PLL CD4046BC will be used for this experiment and the next. A data sheet for it can be downloaded at "http://www.fairchildsemi.com" from Fairchild Semiconductor and other URL's.

Task:

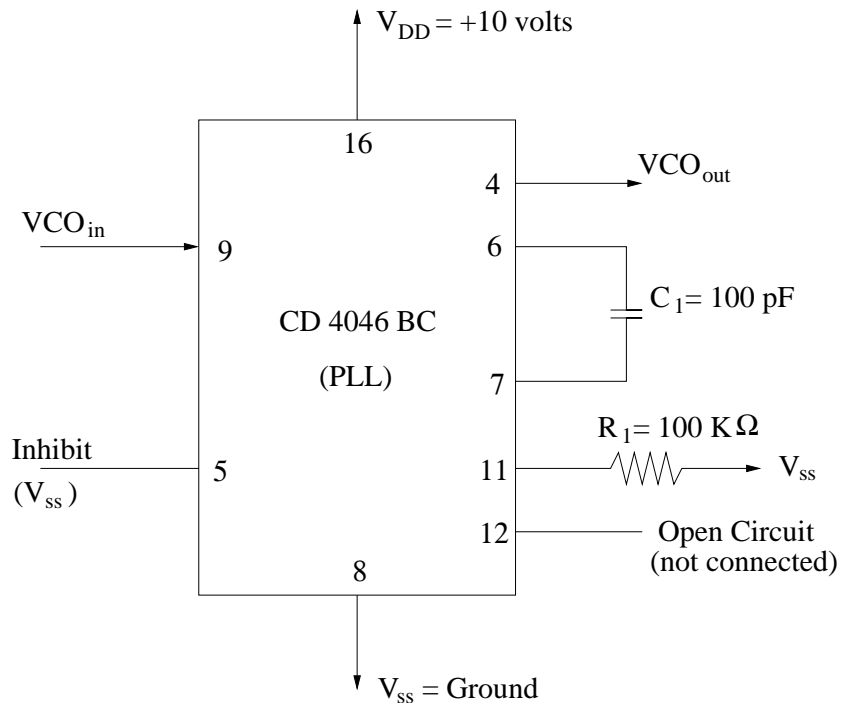Build the circuit shown next. This uses the VCO portion of the 4046 PLL.

Figure 1: Basic VCO Circuit

First, investigate it using the "test input" circuit that is shown in Figure 2. Find the frequency and sketch the waveform for the three VCO input voltages shown in the table below. From that information, determine the FM constant, k_f, for your modulator. See the data analysis section below for guidance in this calculation.
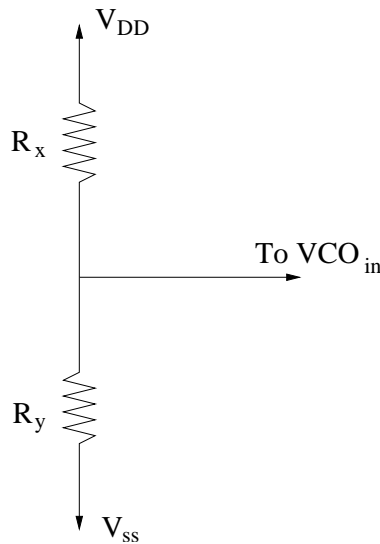

Figure 2: Test Input Circuit

VCO$_{in}$	Frequency
$2V_{DD}/3$	
$V_{DD}/2$	
$V_{DD}/3$	
Here it is assumed that V_{ss} = ground.	

Second, instead of the "test input" circuit, use, as the input, the function generator with the sinusoidal output listed as follows:

>   Frequency = 5 kHz (f_m = modulating frequency)
>
>   Amplitude = 2 volts (p-p)
>
>   D.C. Offset = maximum allowed by function generator

Examine the time-domain signal at the VCO output. It should look similar to the plot of Figure 3. Essentially, this is a rectangular waveform with a varying frequency, i.e., a frequency that is modulated. The maximum and minimum frequencies, $f_{\max}$ and $f_{\min}$, can be determined using the following formulas:

$$f_{\min} = \frac{1}{T_1}, \qquad f_{\max} = \frac{1}{T_2}.$$

Write an expression for the time domain output, assuming that the output waveform is sinusoidal like. What is the "β" for your signal? Examine the spectra using the spectrum analyzer. (Make the connection to the spectrum analyzer using a high impedance scope probe.) Sketch the

spectra and measure the power in the significant sidebands (powers greater than one percent of the total transmitted power). Record this data in the table shown in the "Report" section.
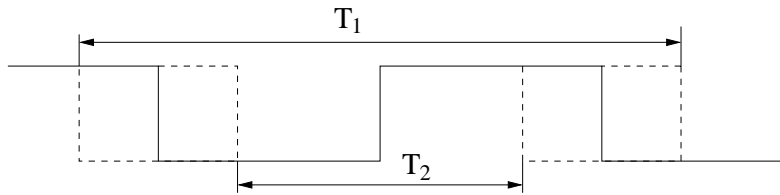

Figure 3: VCO Time-Domain Output

Data Analysis:

The FM constant, k_f, can be determined by plotting the VCO's output frequency vs. the VCO's input voltage. This should give (approximately) a straight line, its slope is k_f in hertz-per-volt. You will want to convert it to radians/second-per-volt in order to write the expression for the FM signal you generate. To find β, use $\beta =$ [peak modulating tone amplitude/modulating tone frequency (in Hz)] $\cdot k_f$. An alternative method is given by

$$\beta = \frac{f_{\max} - f_{\min}}{f_m}.$$

Report:

What are the two values of β that you have found? Using an average of these two β's as an estimate for the output signal's β, find the expected power spectral density (PSD), i.e., the power in the output signal at each sideband frequency, of the FM signal at the VCO output. This is the same calculation you did in lab #8. Enter your calculated power in (dBm) for each sideband frequency in the table below.

Sideband Frequencies	Expected Power (dBm)	Relative Expected Power (dB)*	Measured Power (dBm)	Relative Measured Power (dBm)†
⋮	⋮	⋮	⋮	⋮

* Use the expected power at the carrier frequency as the reference.

† Use the measured power at the carrier frequency as the reference.

Due date: Next lab session to your TA.

3

ECE-311
Experiment #11

Analog Sampling

Objective:

In this laboratory exercise you will build and test a "crude," but effective, analog sampling circuit. This circuit uses a CMOS (Complimentary Metal-Oxide-Semiconductor) analog switch to produce a pulse amplitude modulated (PAM) signal. You will use a single-tone sinusoid as the input signal and will sample it at five times the Nyquist rate. You will then examine the output signal on the oscilloscope and its spectrum on a spectrum analyzer.

Introduction:

Read section 6.1 of the text, especially the part entitled "practical sampling." A somewhat expanded discussion can be found in section 5.1 of *Signal Processing and Linear Systems* by B. P. Lathi, the ECE-310 text. In this lab you will be investigating a particular kind of "practical sampling" called "natural sampling."

The CD4016B CMOS quad bilateral switch will be used for this experiment. A data sheet for it can be downloaded at "http://www.fairchildsemi.com" from Fairchild Semiconductor and from other URL's. This is essentially a voltage-controlled switch. When the control voltage is HIGH the input and output terminals are connected together—i.e., a short circuit, and when the control voltage is LOW the input and output terminals are disconnected—i.e., an open circuit.

Pre-Lab:

Determine the theoretically expected spectrum by determining the Fourier Transform for the output, y(t), in the problem given below. Make sketches of the time domain output signal and its Fourier Transform.

$$y(t) = f(t) \bullet g(t)$$

$f(t) = \cos(\omega_x t),$ frequency = 10 kHz
$g(t)$ = 20% duty cycle rectangular wave, pulse height = 1, baseline = 0, frequency = 100 kHz

Task:

Construct the circuit (figure 1) and test set-up shown on the following pages. Perform the observations and measurements discussed below.

In order to view the natural sampling on the oscilloscope it is necessary to *synchronize* the input sinusoid, $v_{IN}(t)$, that you will be sampling, with the control signal, $v_{CONTROL}(t)$, that opens and closes the switch. First, using the proper triggering techniques (see figure 2), look at the output, $v_{OUT}(t)$, on the oscilloscope. Make a sketch of the waveform seen there.

Next use the spectrum analyzer to examine the output of the sampler. Since the output of the CMOS switch wants to "see" a high impedance, you should use the oscilloscope probe to connect to the spectrum analyzer. Measure the power (in dBm) of the components of the spectra at the sampling frequency and at integer multiples of that frequency up to six times the fundamental. Record your data.

     $v_{IN}(t)$ = 10 kHz sinusoid with one (1) volt peak-to-peak and a zero d.c. offset

     $v_{CONTROL}(t)$ = 100 kHz, 20% duty cycle rectangular-wave with five (5) volts peak-to-peak voltage and a zero d.c. offset

Data Analysis:

Because you are using a high impedance connection to the spectrum analyzer, the absolute power measurements will be small and not compare favorably with those expected from the theory. To make a comparison you need to normalize the data to a measured (and calculated) reference power. I suggest choosing the data measured (and calculated) at the sampling frequency for the reference.

> Normalized Data (dB) = Raw Data (dBm) – Reference Raw Data (dBm)

> (Here "raw data" refers to the measured or calculated power at the various frequencies. Whereas the "reference raw data" refers to the measured or calculated power at the one reference frequency.)

Report:

In your report, show the results of your theoretically expected calculation of the pre-lab. Carefully draw the expected oscilloscope waveform. Make a plot of the theoretically calculated output spectrum. Determine the predicted powers (in dBm) of the components in this spectrum at the sampling frequency and at integer multiples of that frequency up to six times the fundamental.

Tabulate, as shown below, the measured powers of the spectrum and the calculated powers of the spectrum. Compare their powers at the various frequencies. Remember, it is the relative values that are important when doing this comparison not the absolute values. Thus, hopefully, the "normalized" columns should be close.

	Frequency (Hz)	Expected (dBm)	Normalized Expected (dB)	Measured (dBm)	Normalized Measured (dB)
Sampling Frequency					
Twice Sampling					
Three times Sampling					
Four times Sampling					
Five times Sampling					
Six times Sampling					

Figure 1
"Crude" PAM Circuit

V_{DD} = +5 volts

$v_{IN}(t)$

1

50 Ω

14

CD 4016 B

2

$v_{OUT}(t)$

10 kΩ

5
6
12

$v_{CONTROL}(t)$

13

50 Ω

7

V_{SS} = − 5 volts

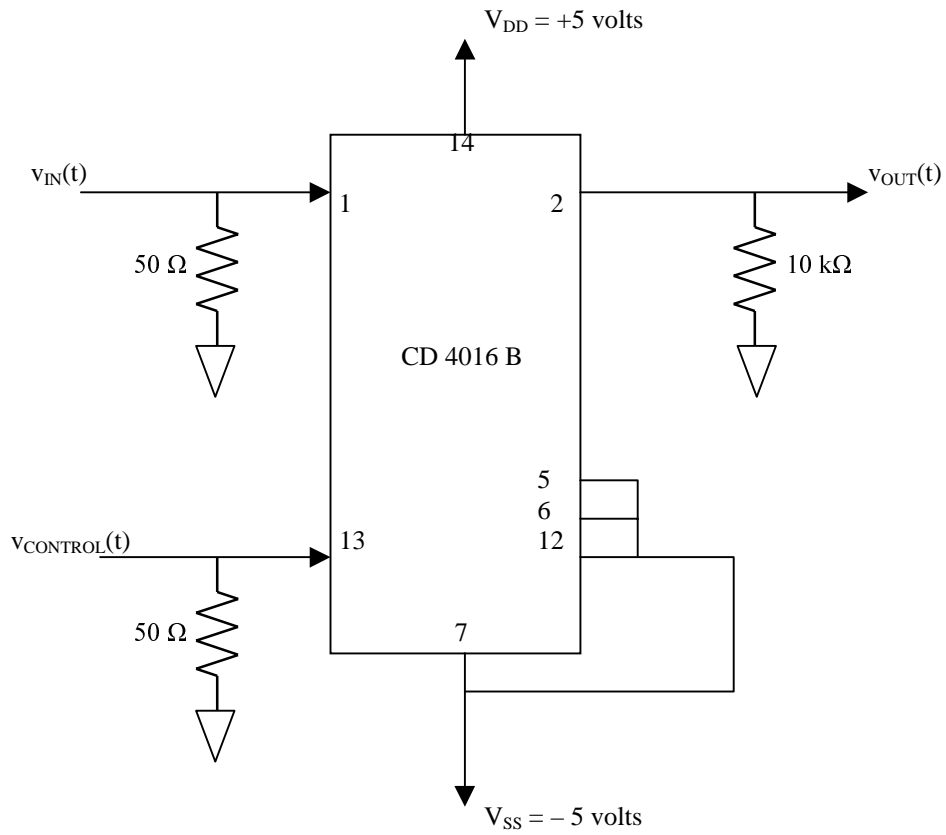==

Figure 2
Block Diagram for Synchronization of Function Generators

Reference Out

Function Generator #1

Sync   Out

External Ref. In

Function Generator #2

Out

$V_{IN}(t)$

$V_{CONTROL}(t)$

Oscilloscope

External Trigger