Understanding RF Circuits with Multisim 10

A Workbook with 21 Circuit Experiments

By Tracy Shields

Notice to the Reader

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by activities described herein and to avoid all potential hazards. By following the instructions contained herein, the reader willingly accepts all risks in connection with such instructions.

For more information contact:
Electronics Workbench Corporation
111 Peter Street, Suite 801
Toronto, Ontario, Canada M5V 2H1

Phone: (416) 977-5550 Fax: (416) 977-1818

Preface

This workbook contains radio frequency (RF) applications to be used with Multisim. Twenty-one specially designed electronic communication experiments are provided for use with this computer controlled environment. These experiments are meant to supplement the theory students normally obtain in lectures. This workbook is not meant to replace traditional laboratory methodology but rather to augment students' hands-on learning experiences by applying features which are inconvenient to implement within the strict limitations of many RF communication laboratories. Within Multisim's environment, unlimited quantities of parts and multiple instruments are available to overcome the traditional obstacles typically encountered by students, technicians, technologists, engineers and others involved in electronic circuit design. Unlike the impact of the restrictions of a singular physical laboratory on an inquiring mind, implementation of the experimental approach to circuit analysis is simple when using Multisim. This simplicity encourages further investigation. The experiments provided in this workbook are flexible when implemented within Multisim's environment allowing the transition from theory to lab in an easy and satisfying manner. For the student learner, asking questions and obtaining answers becomes non-intimidating.

With the use of Multisim, in-depth experimentation is encouraged. Every component in each experiment in this workbook may easily be replaced or have its value changed and the resulting simulation predicted and observed. Designs may be quickly tested and altered in order to determine the effectiveness of the configuration. The design may then be transferred to physical components. Troubleshooting of the physical circuit is minimized.

I have seen many enhancements and features added to Multisim in the years that I have been using the program. Some of the newer features include an Instrument probe which allows the user to take quick measurements around the circuit without placing instruments on the workspace, a multiple choice and true/false choice question/answer feature which can easily be set up and submitted to teachers via e-mail and wizard features that design filters, 555 timers, BJT, and Opamp circuitry according to menu-driven specifications. Breadboarding features as well as Tektronix and Agilent instruments that closely simulate real instruments in their presentation and operation are among the new features of Multisim. An audio file is accessed through the description box of SpecIntro.ms10.

A fundamental, seamless liaison exists between National Instruments LabVIEW and Multisim creating two simulated worlds that connect directly to the physical laboratory. The simulated/physical liaison provides a versatile platform that promotes a new way of thinking about electronic circuit design and troubleshooting. In an environment which minimizes frustration and intimidation, troubleshooting concepts tend to follow naturally as a consequence of having fun. As well, learning course material through the utilization of both the physical and the simulated worlds, results in an alternative to understanding of topic areas through the mathematics that make them possible. LabVIEW instruments are used in the LabViewHarmonicComposition.ms10 lab.

I have written each experiment in modular format so that the facilitator may select those experiments which highlight and provide tangible meaning to existing course material. A textual background introduction to each topic area is provided. Relevant tables and areas for the recording of data and sketching of associated waveforms are supplied in the data section of each experiment.

I have provided the fundamentals of RF communications necessary to comprehend further communication topic areas. Some of these topic areas are covered in this workbook. The fundamentals that are discussed herein include filters, oscillators, amplitude modulation and detection along with principles of frequency modulation.

I have designed the experiments around student familiarization with usage of specialized instruments such as the Bode Plotter, Spectrum Analyzer and Network Analyzer. The familiarity of each of these instruments that is gained through running these RF simulations can be shifted to actual laboratory equipment. With the exception of the Wizard lab, a minimum of one working circuit file is provided with each RF experiment. Formulae are offered where applicable in each case. Many of the experiments are furnished with expected outcomes that the student-learner might use for data comparison. As well, I have provided an additional challenge for most experiments in order to encourage further in-depth circuit and topic area investigation. Solutions are provided.

About the Author

Tracy Shields began her 25 year career in electronics as a technician working in service departments of electronic suppliers. As well as completing an Electrical Engineering degree, Tracy has obtained an Electronics Computer Technician diploma and an Electronics Communication Technology diploma, providing her with a rich array of theoretical design and hands-on experience. Her interest in electronics has encouraged many others in the field through her ten years of full-time teaching in the Electronics Department at Seneca College, Toronto, Ontario, Canada. Tracy currently provides electronic circuit design for engineering companies and consulting services on a number of topics in the field of computer and communications technology.

Acknowledgement

Thank you Dennis, my husband, for the manner in which you inspire me and the patience that you exhibited during the writing and testing of this workbook.

For my children, Mary and Sarah.

Table of Contents

Experiment 1: Introduction to the Spectrum Analyzer

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of the Spectrum Analyzer as it applies to radio frequency (RF) communications. The Spectrum Analyzer is one of the most essential testing instruments in the field of communications. To this point, the oscilloscope has been relied upon to analyze AC signals and their relationships to one another. The oscilloscope allows measurements of phase variations between signals as well as period, amplitude and overall signal performance. The time versus amplitude scale, however, does not allow the specific components of a signal to be studied. The Spectrum Analyzer measures signals in the frequency domain, decomposing them into spectral lines at different frequencies. Ideally, a perfect undistorted sine wave is connected to the input of a Spectrum Analyzer. A single vertical line at its fundamental frequency of oscillation is observed. However, signal sources in the real world are not perfect. They always create some distortion which results in tiny spectral lines. These lines are multiples of the fundamental frequency and are low in amplitude.

Spectral analysis is encouraged in Multisim through use of the Spectrum Analyzer. Multiple duplicate instruments may be utilized for further convenience in the study and testing of AC signals.

Parts

AC Voltage Source

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Power in watts

$P = V_{rms}{}^2 / R_L$ Equation 1-1

Procedure

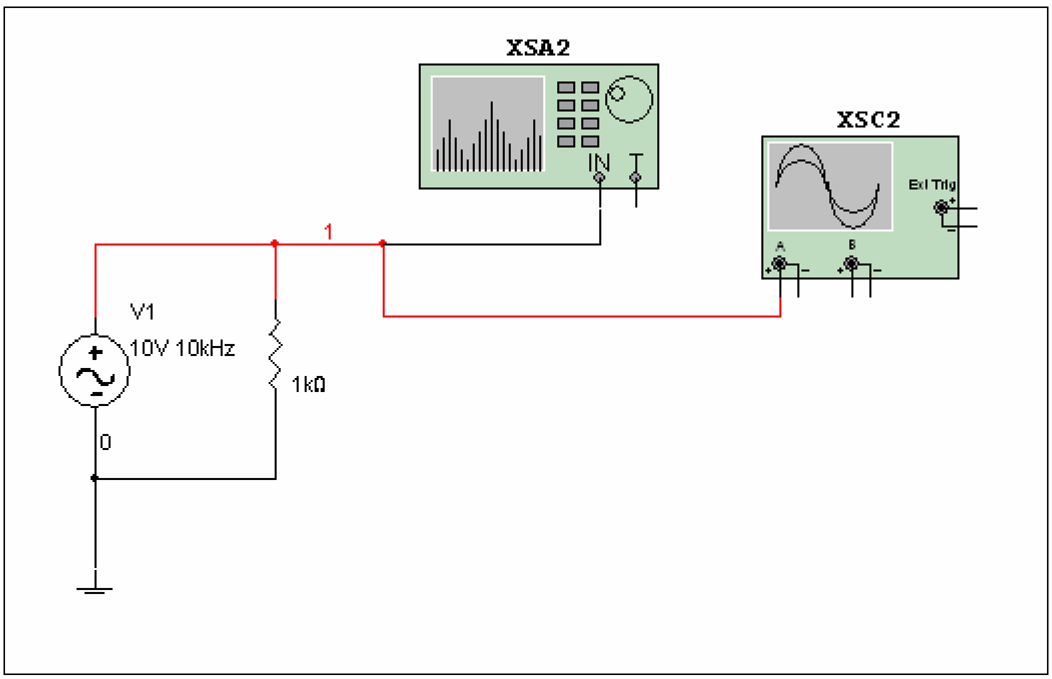


Figure 1-2

1. Connect the circuit illustrated in Figure 1-2.
2. Double-click on the AC Voltage Source. Select the Value tab and set Voltage (Pk) = 10V, Voltage Offset = 0V and Frequency = 1 kHz. Click OK.
3. Double-click on the Oscilloscope. Set the time division to 1 ms/Div and the Amplitude to 5 V/Div. Start the simulation and observe the 1 kHz frequency on the time versus amplitude scale. Stop the simulation.
4. Double-click on the Spectrum Analyzer. Select Set Span under *Span Control* and Lin under *Amplitude*. Under *Frequency*, set the *Start* and *End* frequencies. Since we are interested in a frequency of 1 kHz, select *Start* = 0 Hz and *End* = 2 kHz.

This will define the frequency settings for the beginning and end of the window, thus centering the 1 kHz frequency. Click *Enter* to enter the values. This method of setting the Spectrum Analyzer parameters is called the Frequency Control method.

5. Start the simulation. Place the cursor over the vertical marker line and drag the line to the center of the peak of the spectrum shown. Observe the frequency value in the lower-left portion of the window change as the marker is moved. Record the frequency and voltage values in Table 1-1.Verify that the frequency and amplitude values correspond to the AC Voltage Source settings.
6. Stop the simulation. Double-click on the AC Voltage Source and select Voltage Amplitude = 10V and Frequency = 10 kHz.
7. Since the frequency of interest is now 10 kHz, set the *Center* frequency to 10 kHz. Set *Span* to 10 kHz. This will designate a total window span of 10 kHz. Click *Enter*. Notice that the *Start* and *End* frequencies are automatically calculated. This technique is called the Span method. Note that one of the two illustrated methods may be used, but not both at once.
8. Start the simulation and move the marker to the left side of the window then the right side of the window, noting that the frequencies at each end correspond to the *Start* and *End* frequency settings. In order to obtain the *Span* setting, subtract the *Start* frequency from the *End* frequency. Record your results in Table 1-2. Verify the span setting. Record LIN voltage and mW values as in Step 5, converting the voltage to power in mW using Equation 1-1. Complete Table 1-1.

Expected Outcome

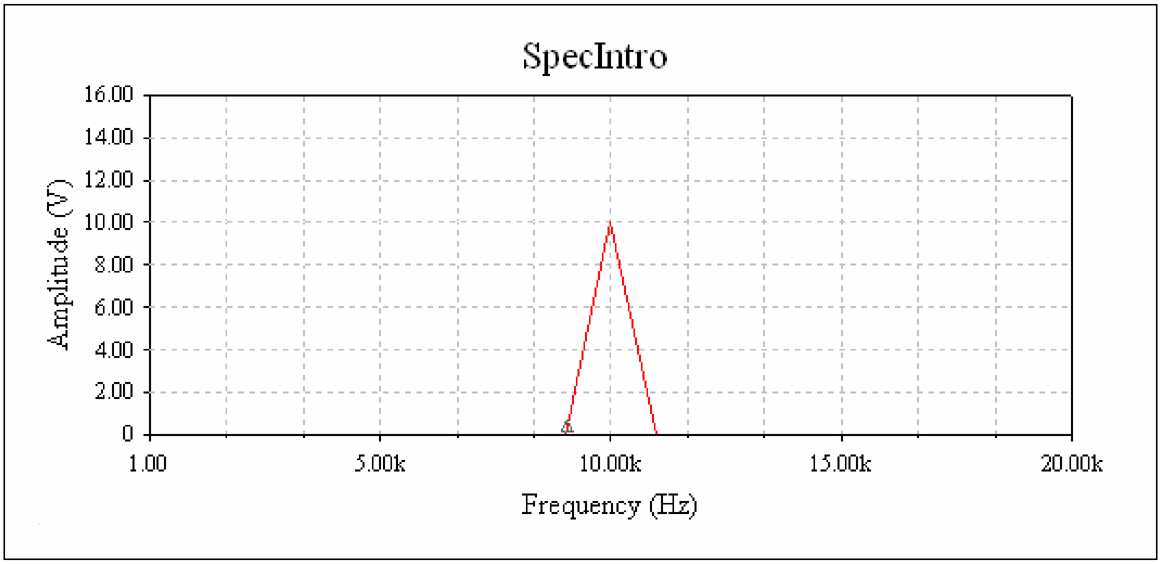


Figure 1-3 Frequency Spectrum of a 1 kHz Signal

Data for Experiment 1

Frequency (measured)	**LIN Voltage (V) (measured)**	**Power (mW) (calculated)**	**Frequency (Hz) (expected)**	**LIN Voltage (V) (expected)**
1 kHz				
10 kHz				

Table 1-1

Frequency (Hz)	**Start (Hz)**	**End (Hz)**	**Span (Hz) (measured)**	**Span (Hz) (expected)**
10 kHz				

Table 1-2

Additional Challenge

1. Set the amplitude of the AC Voltage Source in Figure 1-2 to 1 V and the frequency to 10 kHz.
2. Set *Span* = 10 kHz and *Center Frequency* = 8 kHz. Click *Enter*.
3. Verify that the *Start* and *End* frequencies are correct for the *span* and *center* frequencies selected.

Experiment 2: Spectral Analysis of Signal Harmonics

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of signal harmonics using the Spectrum Analyzer and Wattmeter. Pure sinusoidal waveforms represent themselves as a single vertical spectral line in the frequency versus amplitude domain. When distortion is introduced, harmonics appear at multiples of the fundamental or frequency of oscillation. Mathematically, the addition of power in mW of each harmonic to that of the fundamental frequency results in the reconstruction of the total waveform.

Other waveforms such as square waves, triangle waves and sawtooth waves contain an infinite number of harmonics which, when added together, furnish the shape, amplitude and frequency of the signal. Whatever the overall signal characteristics, spectral analysis is essential in the study of its individual components. Spectral Analysis is used in Experiment 9 to dissect an amplitude modulated signal.

A sawtooth wave is made up of an infinite number of periodic sinusoidal frequencies which are whole multiples of the fundamental frequency. As their order increases, their amplitude and power decreases. The Fourier series for a sawtooth wave is:

$$E_n = \frac{2A}{\pi}(\sin \omega t - \frac{1}{2}\sin 2\omega t + \frac{1}{3}\sin 3\omega t - \frac{1}{4}\sin 4\omega t + \ldots..)$$

For our rms harmonic voltage level calculations we will use the approximation:

$$E_n = \frac{2\,Amplitude}{n\pi\sqrt{3}}$$

The fundamental frequency is:

$$f_0 = \frac{1}{period\ \tau}$$

Although f_0 might represent the fundamental frequency of a square wave without the addition of its odd harmonics, f_0 is simply a sinusoidal wave. The second harmonic has a sinusoidal frequency of $3f_0$. The third harmonic has a sinusoidal frequency of $5f_0$. As each harmonic is added, the waveform looks more like a square wave. A perfect square wave is made up of an infinite number of odd harmonics.

For example, if the fundamental frequency of a square wave is 1 kHz, $3f_0$ = 3 kHz, $5f_0$ = 5 kHz and $7f_0$ = 7 kHz. The Fourier series for a square wave is:

$$E_n = \frac{4A}{\pi}(\cos \omega t - \frac{1}{3}\cos 3\omega t + \frac{1}{5}\cos 5\omega t - \frac{1}{7}\cos 7\omega t + \ldots..)$$

For our rms harmonic voltage level calculations we will use the approximation:

$$E_n = \frac{4\,Amplitude}{n\pi\sqrt{2}}$$

Since the amplitude of higher order harmonics is significantly smaller than that of the fundamental frequency, harmonics are generally only calculated to the 5th order.

Parts

Resistor: 1 kΩ

Test Equipment

- Function Generator
- Oscilloscope
- Wattmeter
- Spectrum Analyzer

Formulae

Power in watts

$P = V_{rms}{}^2 / R_L$ Equation 2-1

Voltage in rms of nth order harmonic of a square wave

$$E_n = \frac{4\,Amplitude}{n\pi\sqrt{2}}$$

Equation 2-2

$rms = amplitude$ Equation 2-3

Voltage in rms of nth order harmonic of a sawtooth wave

$$E_n = \frac{2\,Amplitude}{n\pi\sqrt{3}}$$

Equation 2-4

$$rms = \frac{amplitude}{\sqrt{3}}$$

Equation 2-5

Procedure

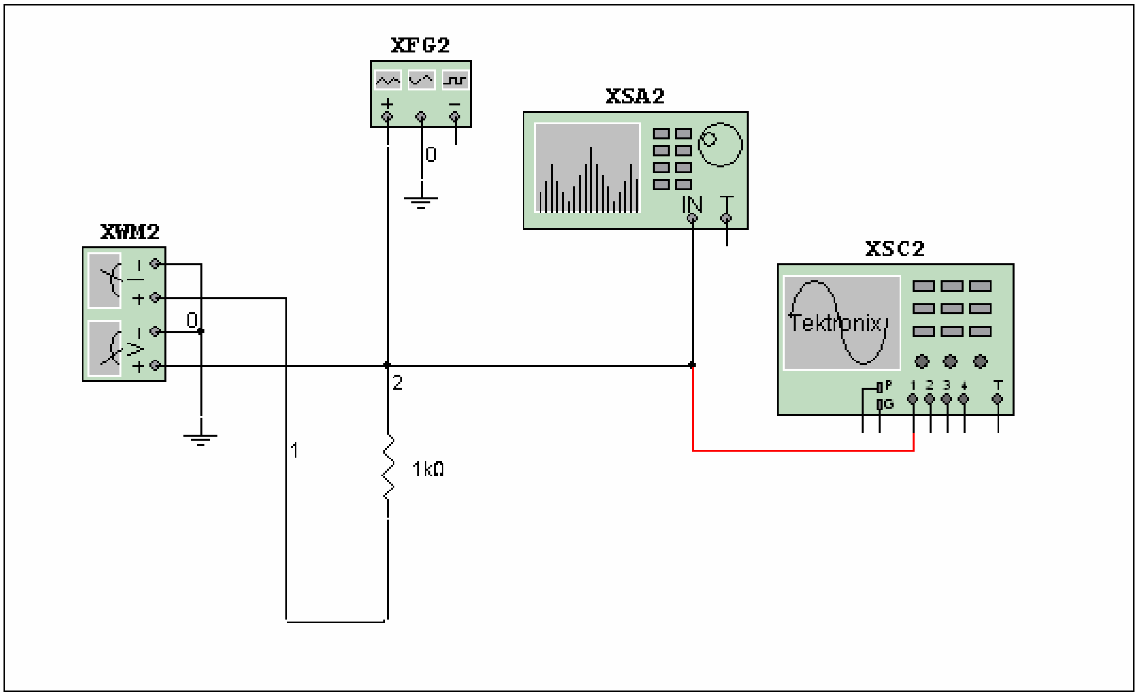


Figure 2-1

1. Connect the circuit illustrated in Figure 2-1. Connect the Function Generator, Oscilloscope, Wattmeter and Spectrum Analyzer as shown. When connecting the Wattmeter, note that the side of the meter marked with V is connected in parallel with the load and the right side is connected in series with the load.
2. Double-click the Oscilloscope to view its display. Set the time base to 50 µs/Div and Channel 1 to 10 V/Div. Select Auto triggering and DC coupling.
3. Double-click the Function Generator to view its display. Choose Frequency = 10 kHz, Duty Cycle = 50%, Amplitude = 10 V and Offset = 0. Choose Square wave.
4. Double-click on the Spectrum Analyzer to view its display. Select *Set Span*. Choose *Start* = 10k, *End* = 100k and *Amplitude* = Lin. Click *Enter*.
5. Start the simulation.
6. Double-click the Oscilloscope and verify that the amplitude is 10 V.
7. Double-click the Spectrum Analyzer. Move the vertical marker to the left side of the window and measure the frequency and amplitude of the fundamental 10 kHz signal. These values will be shown at the bottom left of the window. Calculate the associated power in mW, using equation 2-1.
8. Calculate the expected rms voltage using equation 2-3. Calculate the expected power. Repeat measurements and calculations for the 3rd and 5th harmonics. Add the power in mW of the fundamental, 3rd and 5th harmonics to calculate the total power. Verify your results by double-clicking on the Wattmeter. Record your results in Table 2-1.
9. Double-click on the Function Generator and select the triangle waveform.

10. Observe the display on the Oscilloscope window.
11. Double-click the Spectrum Analyzer and observe the display. Describe the spectrum including the location of the harmonics.
12. Double-click on the Function Generator once more and select a duty cycle of 80%. Observe the display on the Oscilloscope window noting the change in the waveform.
13. Double-click on the Spectrum Analyzer and observe the spectrum of a sawtooth waveform.
14. Complete Table 2-2 for the sawtooth waveform.

Expected Outcome

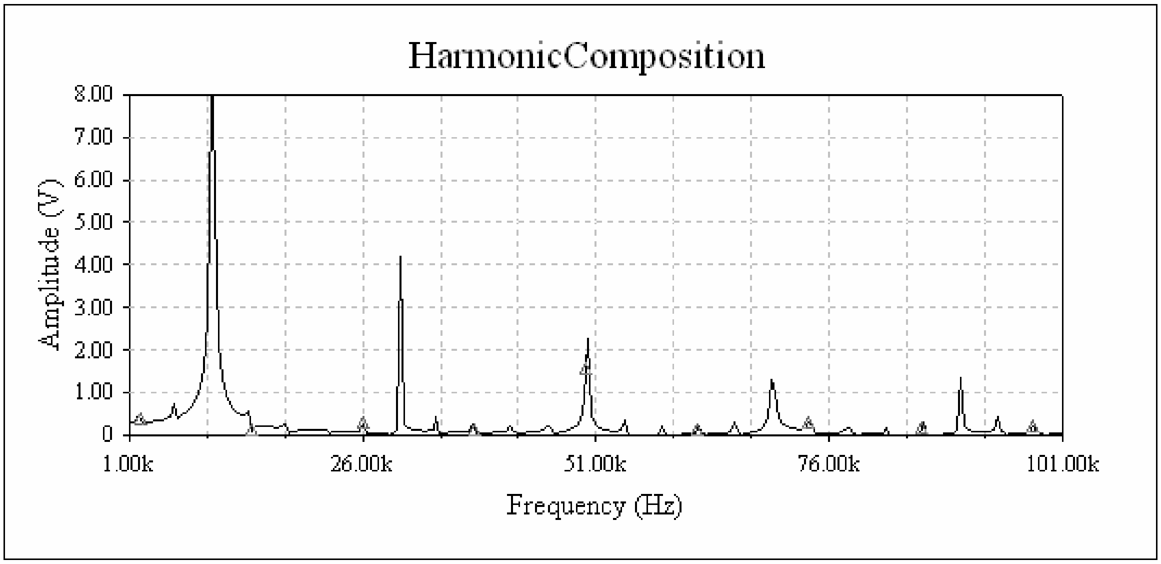


Figure 2-2 Frequency Spectrum of a 10 kHz Square wave

Data for Experiment 2

	Measured Frequency (Hz)	**Measured rms Voltage (V)**	**Measured Power (mW)**	**Expected rms Voltage (V)**	**Expected Power (mW)**
f_0					
f_3					
f_5					

Table 2-1 Square Wave Results

Total Power in mW (measured) _________________

Total Power in mW from wattmeter ______________

	Measured Frequency (Hz)	**Measured rms Voltage (V)**	**Measured Power (mW)**	**Expected rms Voltage (V)**	**Expected Power (mW)**
f_0					
f_2					
f_3					

Table 2-2 Sawtooth Wave Results

Total Power in mW (measured) _________________

Total Power in mW from wattmeter ______________

Additional Challenge

Double-click on the Function Generator. Select triangle wave at a frequency of 1 kHz, an amplitude of 5 V and a duty cycle of 50%. Repeat steps 2 to 5 for measured parameters.

Experiment 2a: LabVIEW Harmonic Composition

Purpose and Discussion

The purpose of this experiment is to demonstrate the characteristics of signal harmonics using National Instruments LabVIEW instruments. As discussed in Experiment 2, the addition of power in mW of each harmonic to that of the fundamental frequency will result in the total reconstruction of the waveform. A pure sinusoidal wave will be represented as one spectral line at the fundamental frequency while square waves, triangle and sawtooth waveforms are made up of an infinite number of harmonics. Generally, only 5th order harmonics are added together as harmonics beyond this order do not add a significant dB level to the overall wave.

Parts

Resistor: 1 kΩ

Test Equipment

- LabVIEW Signal Generator
- LabVIEW Signal Analyzer

Procedure

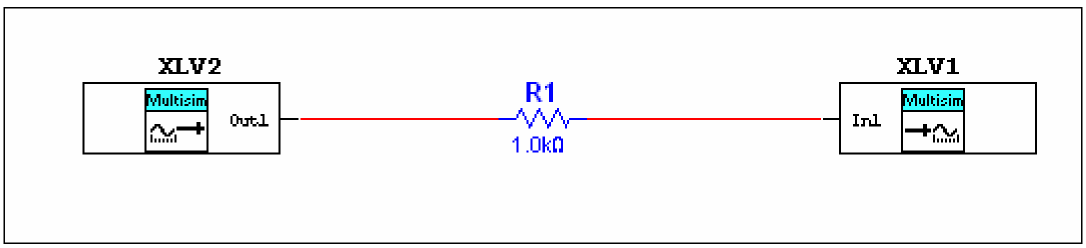


Figure 2a-1

1. Connect the circuit illustrated in Figure 2a-1, selecting the LabVIEW Signal Generator and the LabVIEW Signal Analyzer from the Instruments toolbar.
2. Double-click the LabVIEW Signal Generator to view its display. Select Squarewave and a frequency of 500 Hz.
3. Double-click the LabVIEW Signal Analyzer to view its display. Select *time domain signa*l. Run the simulation and observe the signal as it is displayed on the LabVIEW Signal Analyzer. Select a*uto power spectrum* as the Analysis Type on the Analyzer. Run the simulation and press stop when you have a clear output signal. Observe the dB level of the first five harmonics and record your results.
4. Double-click the LabVIEW Signal Generator to view its display. Select *Trianglewave* and a frequency of 500 Hz.

5. Double-click the LabVIEW Signal Analyzer to view its display. Select *time domain signal*. Run the simulation and observe the signal as it is displayed on the LabView Analyzer. Select a*uto power spectrum* on the Analyzer. Run the simulation and stop it when you have a clear output signal. Observe the dB level of the first five harmonics and record your results.
6. Double-click the LabVIEW Signal Generator to view its display. Select *Sawtoothwave* and a frequency of 500 Hz.
7. Double-click the LabVIEW Signal Analyzer to view its display. Select *time domain signal*. Run the simulation and observe the signal as it is displayed on the LabView Analyzer. Select a*uto power spectrum* on the Analyzer. Run the simulation and stop it when you have a clear output signal. Observe the dB level of the first five harmonics and record your results.

Filters

Filters are used to pass specified frequencies while rejecting others. The extent that a frequency is passed or rejected in the frequency response of the filter is measured by its amplitude. A filter consideration of specific interest is the point on the amplitude versus frequency curve where the amplitude has decreased by 3dB from its maximum voltage or current. This point is referred to as the 3dB cutoff or half-power point and can be observed directly with a bode plotter. This cutoff frequency represents 0.707 of the maximum output ($20 \log 0.707 = 3$ dB) voltage or current and is equivalent to half the maximum power [$10 \log (0.707)^2 = 3$ dB]. The bandwidth or pass-band of a bandpass filter is usually defined by the frequencies between the upper and lower 3 dB points.

Finally, the frequency response of any filter is determined by how fast the curve drops off past the center frequency. This is commonly referred to in decibels/decade (10 times the frequency) or decibels/octave (double the frequency). A single pole filter is characterized by a slope of 20 dB/decade or 6 dB/octave. Second order filters or two-pole filters feature slopes that approach 40 dB/decade or 12 dB/octave. The number of poles found within a given filter is a determining factor in the number of active elements contained within that filter. Sharper roll-off characteristics are provided with higher order filters allowing undesired frequencies to be subjected to greater attenuation. The passive bandpass filter and the passive bandstop circuit are both filters of particular interest in RF communications and are studied in this section. A low pass filter is utilized in the AM envelope detector of Experiment 12.

Experiment 3: The Passive Band-Stop Circuit with Load

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a passive band-stop circuit. The passive band-stop circuit consists of a parallel LC network and requires a load in order to utilize its effectiveness in practice. Tuned circuits are heavily used in receivers as they amplify certain frequencies at resonance. Hence, the term “tuned circuit” is interchangeable with the term “resonant circuit”. An LC band-stop circuit distinguishes itself from a series LC circuit by its parallel arrangement. It features a sharp increase in net impedance at the resonant or center frequency. This results in zero total current at the resonant frequency since equaling produces equal inductive and capacitive current branches.

At the resonant frequency, the circuit exhibits a phase angle of zero. At frequencies above resonance, the line current of a band-stop circuit increases while the impedance decreases. The total line current leads the applied voltage at frequencies greater than resonance and lags the applied voltage below resonance.

Laplace transform analysis is used to designate the number of poles in a given filter. The Laplace transform function for Figure 3-1 is given by the following equation. This equation relies on the assumption that r_1 is resistance inherent to the inductor.

$$\frac{vo}{vi} = \frac{s^2 + \frac{1}{LC}}{s^2 + s\frac{1}{LC} + \frac{1}{LC}}$$

The cutoff frequency represents 0.707 of the maximum output impedance. As in the series bandpass filter, the bandwidth of a LC band-stop circuit is defined by the frequencies between the upper and lower 3 dB points. For the first order filter in this experiment the slope should approach 40 dB per decade around the frequency of interest.

Parts

AC Voltage Source
Resistors: virtual 10 Ω (2)
Inductor: virtual 200 µH
Capacitor: virtual 220 pF

Test Equipment

- Oscilloscope
- Bode Plotter

Formulae

Center Frequency

$$f_C = \frac{1}{2\pi\sqrt{LC}}$$

Equation 3-1

Impedances

$$X_L = 2\pi f_C L$$

Equation 3-2

$$X_c = \frac{1}{2\pi f_C}$$

Equation 3-3

Decibels

$$dB = 20 \log V$$

Equation 3-4

Procedure

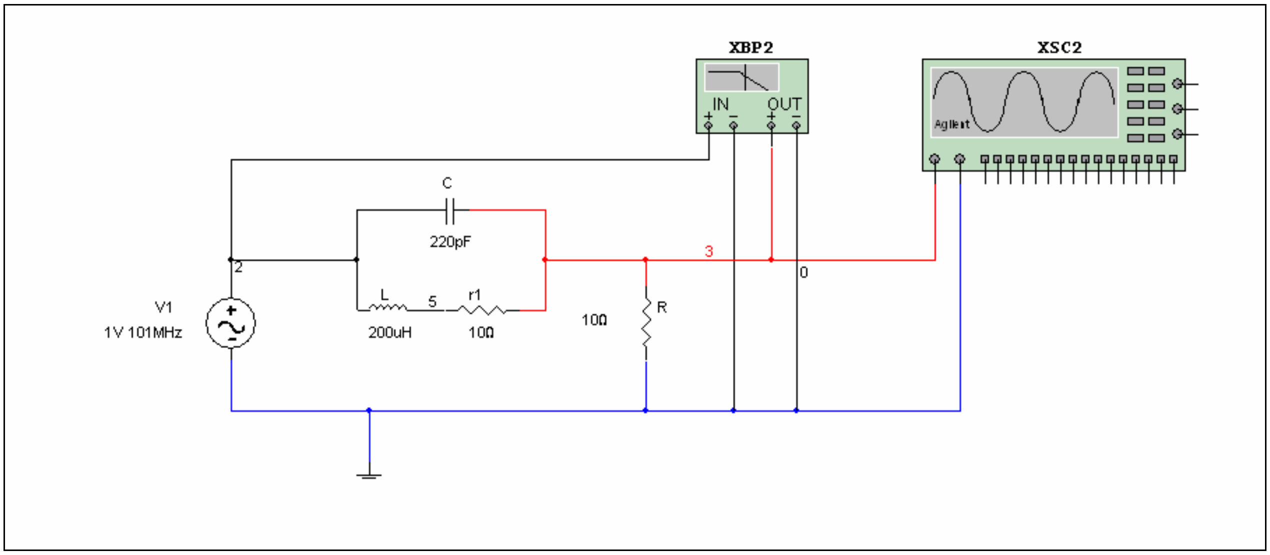


Figure 3-1

1. Connect the circuit components illustrated in Figure 3-1.
2. Calculate the resonant frequency of the band-stop circuit and note this value in Table 3-1.

3. Double-click the AC voltage source and enter the calculated resonant frequency.
4. Double-click the Oscilloscope to view its display. Set the time base to 10 ns/Div and Channel 1 to 500 mV/Div as indicated at the top of the display window.
5. Start the simulation and measure the frequency of oscillation at the output. Note the associated amplitude in Table 3-1.
6. Refer to Table 3-1 and enter the AC Voltage Source frequency = each frequency listed and the Amplitude = 1. Measure and note the associated amplitude at each frequency given. Calculate the associated dB value using equation 3-4. You will have to run the simulator for each measurement. Draw a sketch of amplitude versus frequency for your data. Comment on your data.
7. Double-click the Bode Plotter and choose Magnitude, LOG, F = 0 dB, 1 GHz, I = -200 dB, 1 mHz.
8. Restart the simulation and estimate the bandwidth of the filter by dragging the red marker to the 3dB points as indicated by the frequency and dB values shown on the lower right section of the Bode Plotter. Verify that your sketch corresponds to the Bode Plotter display.

Expected Outcome

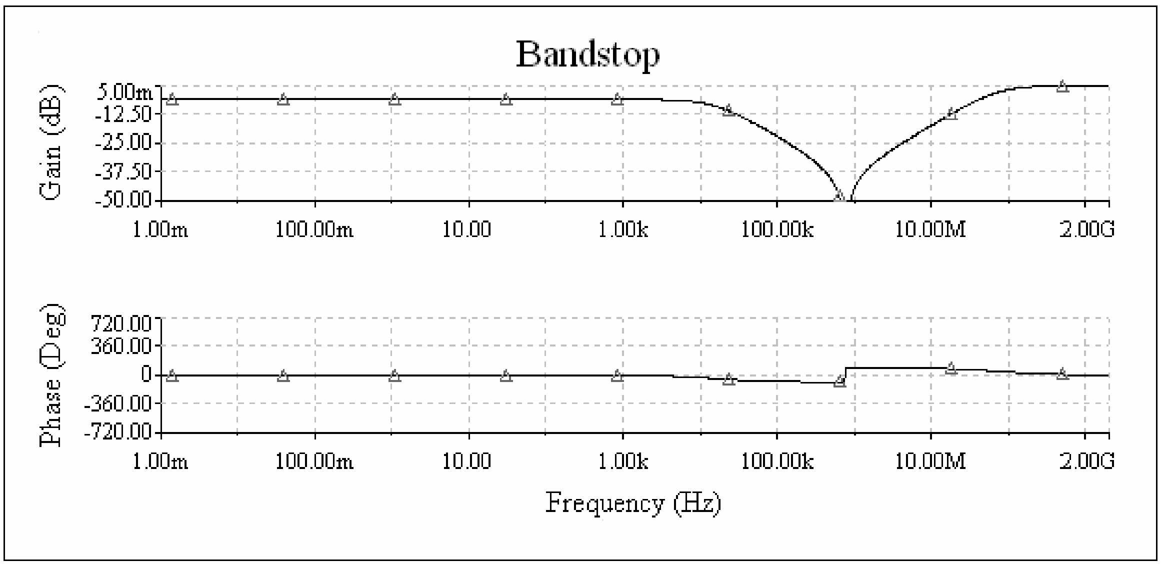


Figure 3-2 Bode plot of band-stop circuit

Data for Experiment 3

Frequency	Amplitude (mV)	Decibel Gain (dB)
7.6 kHz		
76 kHz		
f_c = ________		
760 kHz		
7.6 MHz		
76 MHz		

Table 3-1

Additional Challenge

For Figure 3-1, calculate C so that the circuit resonates at f_c = 1010 kHz. Set up a table similar to Table 3-1 using frequencies of 10.1 kHz, 101 kHz, 1010 kHz, 10.1 MHz and 101 MHz. Replace existing simulated component values by double-clicking on the component of interest. Run the simulation and comment on your data.

Experiment 4: The Passive Band-Pass Filter

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a passive band-pass filter. Band-pass filters reject all input signal frequencies outside a specified given range called the bandwidth, while passing all frequencies within this range. The frequency at which the output voltage is a maximum is known as the resonant or center frequency. In a passive band-pass circuit configuration, a low LC ratio provides a wide band-pass while a high LC ratio produces a more narrow response. The bandwidth of an LC series circuit is defined by the frequencies between the upper and lower 3 dB points. In the band-pass filter shown in Figure 4-1, the roll-off characteristics apply to both sides of the center frequency.

Laplace transform analysis is used to designate the number of poles in a given filter. For the second order filter in this experiment the slope should approach 40 dB per decade around the frequency of interest. The Laplace transform transfer function for Figure 4-1 is given by the following equation:

$$\frac{vo}{vi} = \frac{R}{L}\left(\frac{s}{s^2 + s\frac{R}{L} + \frac{1}{LC}}\right)$$

The varying of the values of R, L or C will result in changes of the location of the poles. Modifying resistor R will change the bandwidth but not the resonant frequency. Varying capacitor C_2 will change the resonant frequency but not the bandwidth. Changing the value of inductor L will alter both the resonant frequency and the bandwidth.

Parts

AC Voltage Source
Resistors: 1.1 Ω, 1 Ω
Inductor: 33 µH
Capacitor: 2.4 nF

Test Equipment

- Oscilloscope
- Bode Plotter

Formulae

Bandwidth

$$BW = \frac{R}{2\pi L}$$

Equation 4-1

Quality Factor

$$Q = \frac{f_c}{BW} = \frac{\omega L}{R} = \frac{1}{R}\sqrt{\frac{L}{C}}$$

Equation 4-2

Center Frequency

$$\omega = \frac{1}{\sqrt{LC}}$$

Equation 4-3

Decibels

$$dB = 20 \log V$$

Equation 4-4

Procedure

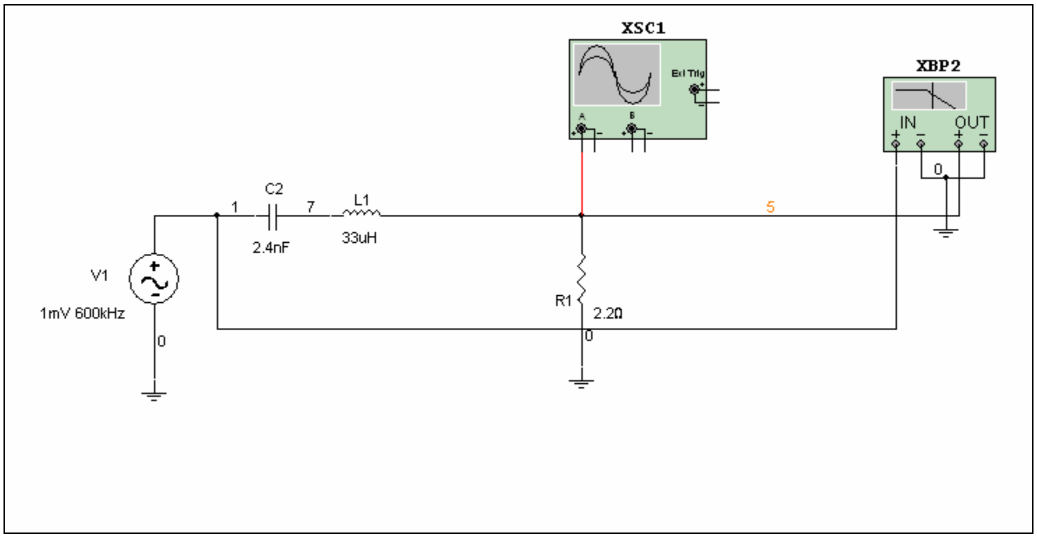


Figure 4-1

1. Connect the circuit components illustrated in Figure 4-1.
2. Calculate the resonant frequency of the bandpass circuit and note this value in Table 4-1.
3. Double-click the AC voltage source and enter Frequency = calculated value.
4. Double-click the Oscilloscope to view its display. Set the time base to 5 µs/Div and Channel A to 200 µV/Div. Select Auto triggering and DC coupling.
5. Start the simulation and measure the frequency of oscillation at the output. Note the associated amplitude in Table 4-2.
6. Refer to Table 4-2 and select the AC Voltage Source frequency = each frequency listed and the Amplitude = 1. Measure and note the associated amplitude at each frequency given. Calculate the associated dB value using equation 4-5. You must run the simulator for each measurement. Draw a sketch of amplitude versus frequency for your data. Comment on your data.
7. Double-click the Bode Plotter and choose Magnitude, LOG, F = 5 dB, 1.3 MHz, I = -60dB, 200 kHz.
8. Restart the simulation and estimate the bandwidth of the filter by dragging the red marker to the 3dB points as indicated by the frequency and dB values shown on the lower right section of the Bode Plotter. Verify that your sketch corresponds to the Bode Plotter display.
9. Compare the bandwidth with theoretical calculations and complete Table 4-1.

Expected Outcome

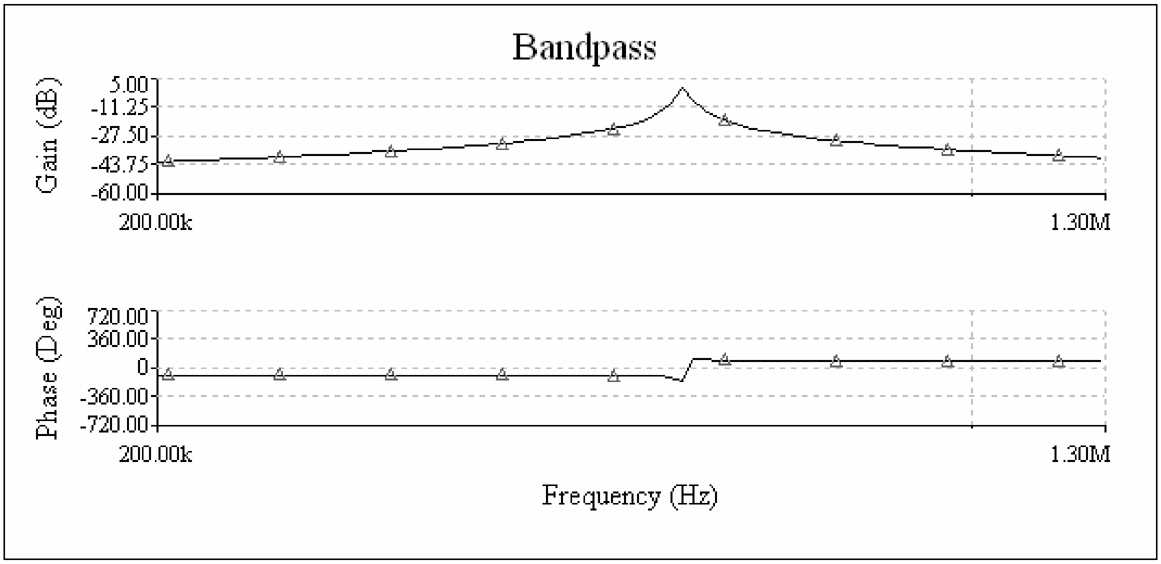


Figure 4-2 Bode Plot of Band Pass Filter

Data for Experiment 4

	Measured Value	**Calculated Value**
BW		
f_c		
Q		

Table 4-1

Frequency	**Amplitude (V)**	**Decibel Gain (dB)**
f_c = ________		
600 Hz		
6 kHz		
60 kHz		
600 kHz		
6 MHz		
60 MHz		
600 MHz		

Table 4-2

Additional Challenge

Using the formulae provided, re-design component values for the circuit of Figure 4-1 in order to achieve fc = 455 kHz. Replace existing simulated component values by double-clicking on the component of interest. Run the simulation and compare the output data with expected theoretical values.

Experiment 5: Low Pass and High Pass Filter Generation

Purpose and Discussion

The purpose of this experiment is to demonstrate the characteristics and operation of the many variables involved when generating Low and High Pass Filters utilizing the Wizard feature of Multisim. Both active and passive circuit characteristics will be examined. Both Butterworth and Chebychev circuits will be generated by the Wizard feature. Low pass filters reject all frequencies above the pass frequency. High pass filters reject all frequencies below the pass frequency. The -3dB or corner frequency is the frequency where the amplitude of the signal has been attenuated by 0.707 or -3dB.

Test Equipment

- Bode Plotter
- AC Voltage Source

Formulae

$$Corner\ frequency = 0.707Vin$$

Equation 5-1

Procedure

1. Select Tools/Circuit Wizards/Filter Wizard from the main menu. Select Low Pass Filter, approximately 9 kHz low pass frequency, approximately 19 kHz Stop frequency, Butterworth Type, Active Topology. When you select Active, you are choosing to build your filter from Operational amplifiers. When you select Passive, you are choosing to build your filter from passive components such as resistors, inductors and capacitors.
2. Click Verify. If the calculation is successful, click Build Circuit. The circuit will be automatically generated for you. Click on the workspace where you would like it to be situated. Of what order is this circuit? Which filter selection would you change to build a higher order circuit?
3. Select an AC source from the top of the Power source components bar. Double-click on it to change the frequency to approximately 500 Hz and the voltage to 12 volts.
4. Select a Bode Plotter from the Instrument panel to the right of the workspace. The IN connections of the Bode Plotter should be placed across the input and the OUT connections should be placed across the output. Run the simulation. You should see a typical low pass filter output as shown in Figure 5-1.
5. Select the right red marker and drag it until you reach the -3dB point as shown at the bottom of the window. Calculated the number of dB per decade.

6. Repeat, selecting High Pass Filter approximately 3 kHz high pass frequency, approximately 3.75 kHz stop frequency, Chebyshev Type and Passive Topology. Which filter selection would you change to build a lower order circuit? Restart the simulation, observing the high pass filter output.
7. Change the Bode Plotter display to PHASE and measure the phase shift (leading or lagging) at the frequencies determined above.
   a. Phase shift when output is 90% = __________________degrees.

   b. Phase shift when output is 70.7% = ________________degrees.

   c. Phase shift when output is 10% = __________________degrees.

Expected Outcome

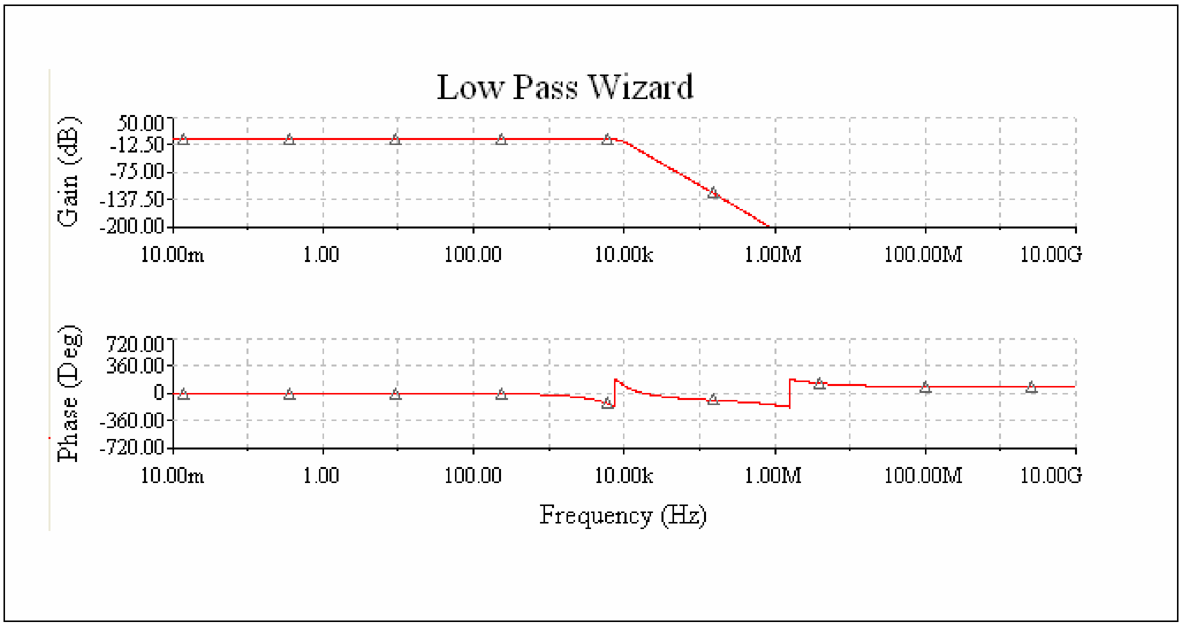


Figure 5-1

Oscillators

Oscillators are used as AC signal sources. They convert DC power supplied by a DC power supply, into an AC signal. Oscillators generate both sinusoidal and non-sinusoidal continuous output signal waveforms maintaining a desired frequency within design limits. Undesired oscillations are known to occur in circuits that are not designed to oscillate. Thus, an overall understanding of the basic principles of oscillators can aid in their prevention.

Oscillators are essential in the development of any communication system. High frequency carrier signals which are discussed in the AM communication section of this book are generated by oscillators. Low frequency modulating test signals also make use of oscillation designs. Oscillators produce pulse generators, sawtooth generators and timing clocks. Several different sinusoidal oscillators will be discussed, some of which are named after their designers.

Oscillators require a gain of one from input to output and the phase around the feedback loop must, in all cases, equal zero. The above requirements satisfy the Barkhausen criteria. The input power supplied by the DC source is also required for self sustaining oscillations. Oscillators must not only deliver power to the load, but they themselves consume power inherently. Oscillators must also provide positive feedback and gain. Gain must be provided in order to make up for circuit attenuations. Small start-up voltages are supplied to the oscillator input terminals which are amplified and are required to launch the oscillations. In practice, these voltages are provided by surges resulting when the DC supplies are first turned on. In Multisim, these initial small starting voltages are simulated.

Experiment 6: The Phase Shift Oscillator

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a phase shift oscillator. Figure 6-1 uses a third order high-pass RC network feedback loop for its particular oscillator circuit design. As is the case with all oscillators, the Barkhausen criteria specifying a required 360 degree phase shift from input to output and a total gain of one must be adhered to in the design of a phase shift oscillator. In Figure 5-1, the inverting op amp provides a phase shift of 180 degrees. The RC network must provide an additional 180 degrees for a total phase shift of zero degrees. Each section provides approximately 60 degrees of this requirement. The filter portion consisting of the RC network introduces an attenuation that the op-amp must match in gain in order to achieve an overall gain of one.

The minimum gain required of the op-amp so that it sustains oscillations is 29. Keeping the gain as close to 29 as possible will prevent the peaks of the waveform from being driven into the non-linear region. This will minimize clipping of the sinusoidal output.

Parts

DC Supplies: +10 V, -10 V
Opamp: 741-DIV
Resistors: 10 kΩ (3), 1 MΩ potentiometer
Capacitor: 10 nF (3)

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Frequency of Oscillation

$$f_C = \frac{1}{2\pi RC\sqrt{6}}$$

Equation 6-1

$$\frac{R_F}{R} = 29$$ in order to sustain oscillations

Equation 6-2

Procedure

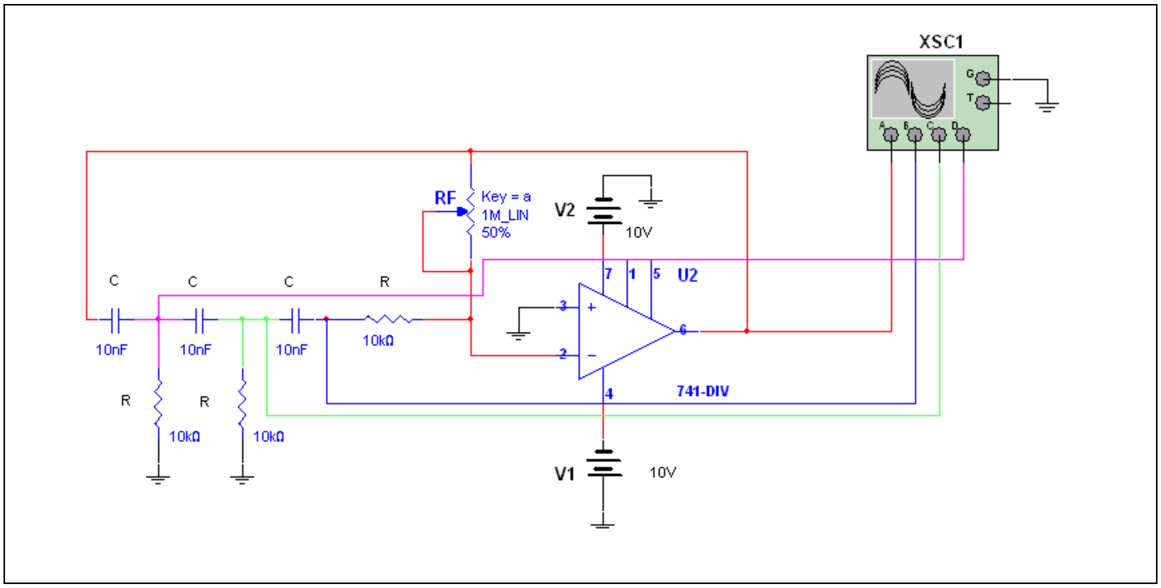


Figure 6-1

1. Connect the circuit components illustrated in Figure 6-1.
2. Double-click the oscilloscope to view its display. Set the time base to 2 ms/Div, Channel A to 2V/Div and Channel B to 200mV/Div.
3. Select Simulate/Interactive Simulation Settings, and select Set to Zero for Initial Conditions.
4. Start the simulation and measure the frequency of oscillation.
5. Stop the simulation and place a Spectrum Analyzer on the workspace and connect its input to the output lead of the oscillator.
6. Double-click to open the Spectrum Analyzer window.
7. Press *Set Span*. Set *Start* = 0 kHz, *End* = 1 kHz, *Amplitude* = LIN and *Range* = 2V/DIV. Click *Enter*.
8. Restart the simulation. When the oscillator has stabilized, drag the red marker to the position of the spectrum line observed. Note the frequency in the lower left corner of the Spectrum Analyzer window.
   f_c = ____________________.
9. Adjust the potentiometer to the point where oscillation just begins. Measure the value of the potentiometer resistance at this point and complete the table below.
10. Open the Oscilloscope window. Measure and note the phase shift at the oscilloscope inputs.

Data for Experiment 6

	Measured Value	Calculated Value
Frequency (Hz)		
Rf/R at point where oscillations begin		

Table 6-1

Expected Outcome

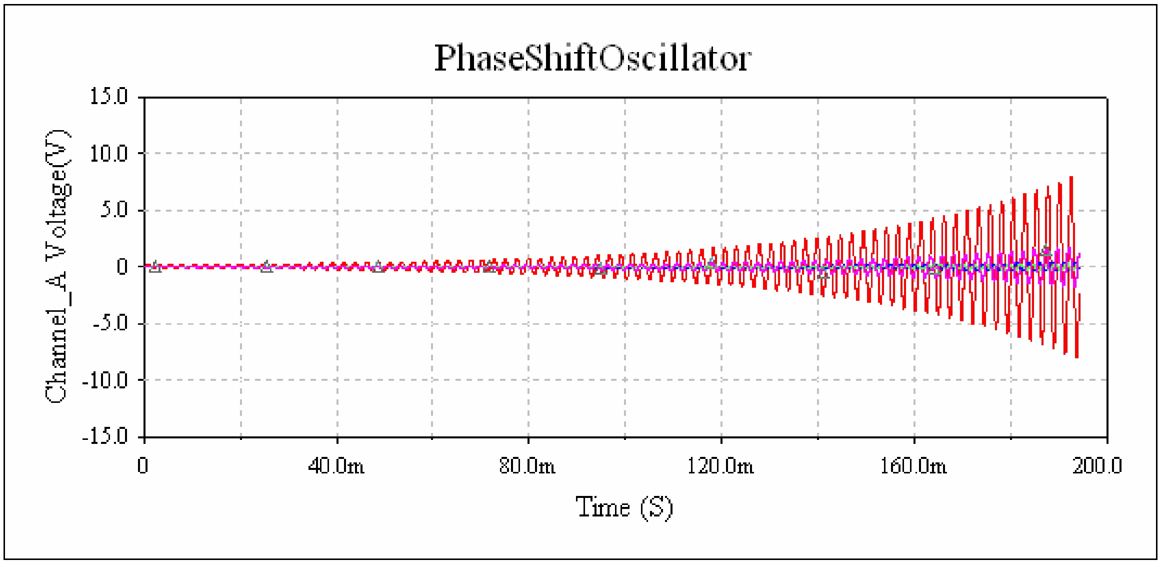


Figure 6-2 Oscilloscope Display of Initial PSO Oscillations

Additional Challenge

For the circuit of Figure 6-1 calculate the value of C required to provide a frequency of oscillation of 900 Hz. Replace existing simulated component values by double-clicking on the component of interest. Run the simulation and compare the output data with expected theoretical values.

Experiment 7: The Hartley Oscillator

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a Hartley oscillator. The Hartley oscillator is characterized by the inductive voltage divider made up of L_1 and L_2. This feedback voltage is used to sustain the oscillations. Once again, the parallel LC resonant circuit is responsible for the oscillator frequency.

As with other LC oscillators, the Barkhausen criteria must be met in order for oscillation to take place. Specifically the gain from input to output must be one and the net phase around the loop must be zero. In the design in Figure 7-1, the BJT must have a voltage gain greater than the ratio L_1 / L_2 in order to sustain oscillations. In other words, the gain of the BJT must make up for the attenuation created by the feedback fraction:

$$B = \frac{L_2}{L_1}. \quad \textit{Since} \quad AvB > 1, \quad \textit{then} \quad Av > \frac{1}{B} = \frac{L_1}{L_2}$$

Parts

DC 12 V Supply
Transistor: Ideal BJT
Resistors: 500 Ω, 10 kΩ
Inductor: virtual 0.5 mH, 2.5 mH
Capacitor: virtual 1 μF

Test Equipment

- Oscilloscope

Formulae

Frequency of Oscillation

$$f_C = \frac{1}{2\pi RC\sqrt{6}}$$

Equation 7-1

Gain

$$Av > \frac{L_1}{L_2}$$

Equation 7-2

Total Inductance

$$L = L_1 + L_2$$

Equation 7-3

Procedure

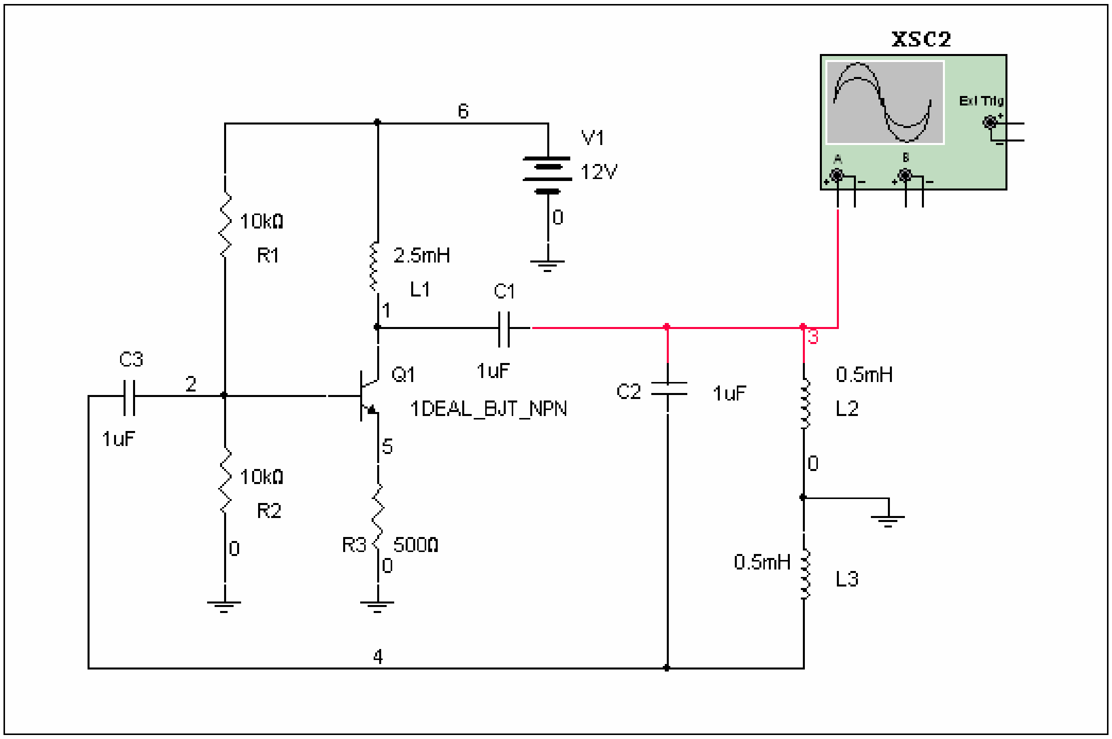


Figure 7-1

1. Connect the circuit components illustrated in Figure 7-1.
2. Double-click the Oscilloscope to view its display. Set the time base to 200 µs/Div and Channel A to 2V/Div. Select Auto triggering and DC coupling.
3. Select Simulate/Interactive Simulation Settings, and select Set to Zero for Initial Conditions.
4. Start the simulation. The oscillator may take a few seconds to stabilize. Measure the frequency of oscillation.
5. Compare with theoretical calculations.
   f_c = _______________*measured* = __________________ *calculated*
6. Stop the simulation and place a Spectrum Analyzer on the workspace.
7. Connect the output lead of the oscillator to the input of the Spectrum Analyzer. Double-click on the Spectrum Analyzer to open its window.
8. Press *Set Span*, set *Span* = 10 kHz, *Center* = 5 kHz and *Amplitude* = Lin and click *Enter*.
9. Restart the simulation. When the oscillator has stabilized, drag the red marker to the position of the spectrum line observed. Note the frequency in the lower left corner of the spectrum analyzer window.
   f_c = _______________

10. Calculate the gain of the circuit and verify that it is greater than 1/B.
11. Calculate the value of L_2 that is required to obtain oscillations of 3.5 kHz. Replace existing simulated component values by double-clicking on the component of interest. Run the simulation to verify your results.

Expected Outcome

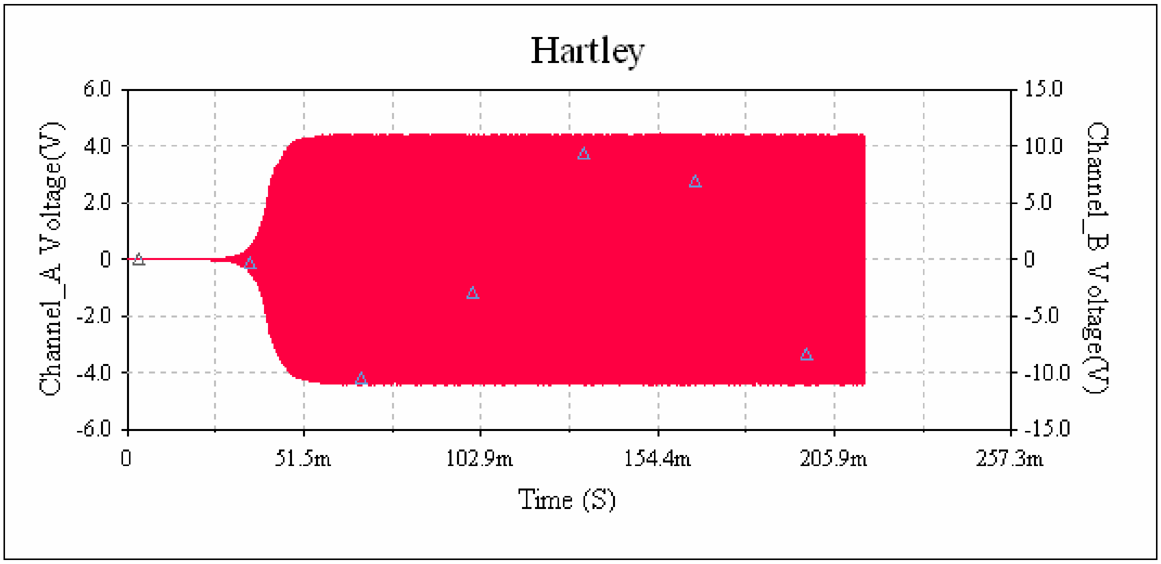


Figure 7-2 Oscilloscope Display of Initial Hartley Oscillations

Additional Challenge

Re-design the circuit of Figure 7-1 to lower the gain to 10. This lower gain will result in a more predictable oscillator. Replace existing simulated component values by double-clicking on the component of interest. Run the simulation and compare the output data with expected theoretical values.

Experiment 8: The Colpitts Oscillator

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a Colpitts oscillator. As is the case with other LC oscillators, the Colpitts oscillator is used for higher frequencies, typically between 1 and 500 MHz. It is characterized by the capacitive voltage divider made up of C_1 and C_2. This feedback voltage is used for the oscillations. Colpitts oscillators can be designed using BJTs, FETs or JFETs. In the design illustrated in Figure 8-1, the loading effect is greatly reduced as compared to a BJT design due to the high input impedance at the gate.

As with other LC oscillators, the Barkhausen criteria must be met in order for oscillation to take place. Specifically the gain from input to output must be one and the net phase around the loop must be zero. In the design illustrated in Figure 8-1, the JFET must exhibit an absolute value of open circuit voltage gain greater than or equal to the ratio C_1/C_2 in order to sustain oscillations. In other words, the gain of the JFET must make up for the attenuation created by the feedback fraction:

$$B = \frac{C_2}{C_1}.\ \ Since\ \ AvB = 1,\ \ then\ \ Av = \frac{1}{B} = \frac{C_1}{C_2}$$

In order to vary the frequency of oscillation, L_1 should be varied. If C_1 or C_2 were chosen instead, the feedback fraction B would be affected.

Parts

DC 12 V Supply
Transistor: Ideal N JFET
Resistors: 1 kΩ, 120 kΩ
Inductor: virtual 60 μH, 5.1 mH
Capacitor: virtual 22 pF, 180 pF

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Frequency of Oscillation

$$fc = \frac{1}{2\pi\sqrt{\frac{L_1C_1C_2}{C_1 + C_2}}}$$

Equation 8-1

Gain

$$Av = -g_m r_d$$

Equation 8-2

Condition for Oscillation

$$Av \geq \frac{C_2}{C_1}$$

Equation 8-3

Procedure

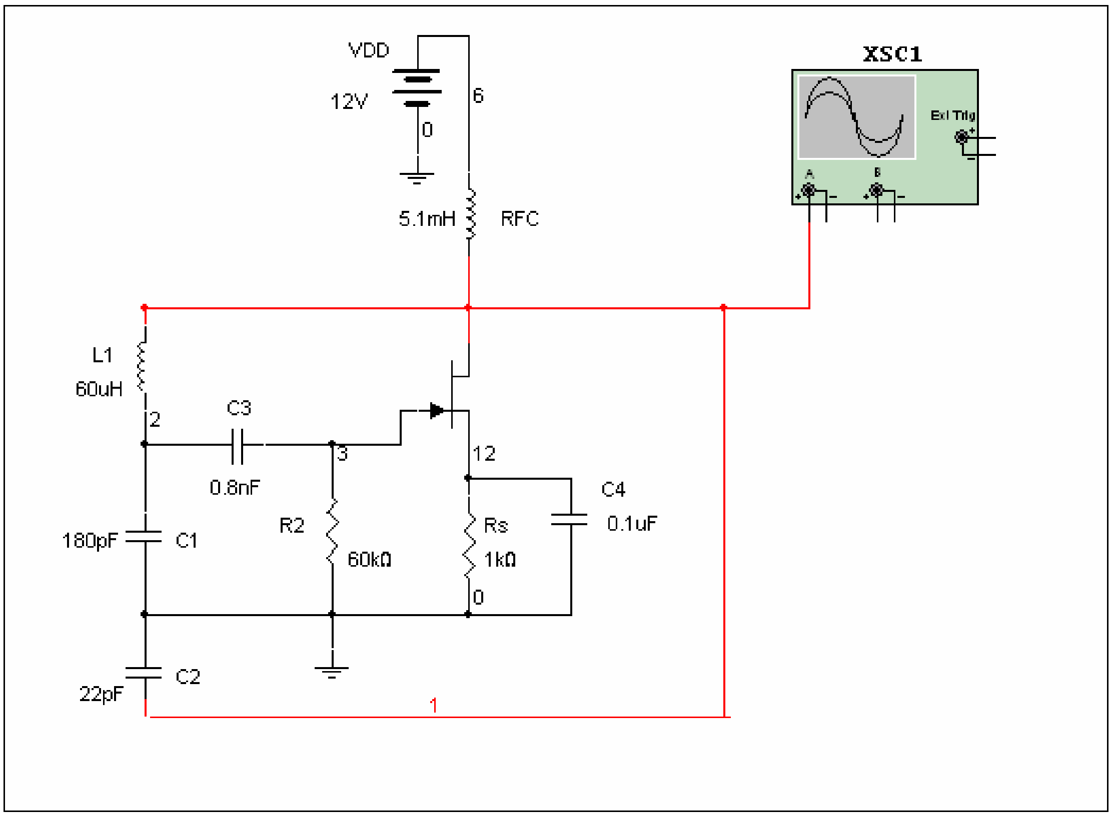


Figure 8-1

1. Connect the circuit components illustrated in Figure 8-1.
2. Double-click the oscilloscope to view its display. Set the time base to 200 ns/Div and Channel A to 10V/Div. Select Auto triggering and DC coupling.
3. Select Simulate/Interactive Simulation Settings, and select Set to Zero for Initial Conditions.
4. Start the simulation. When the oscillator has stabilized, measure the frequency of oscillation.

5. Compare with theoretical calculations:
   f_c = _______________*measured* = __________________ *calculated*
6. Stop the simulation and place a Spectrum Analyzer on the workspace.
7. Connect the output lead of the oscillator to the input of the Spectrum Analyzer.
8. Double-click to open the Spectrum Analyzer window.
9. Press *Set Span*. Set *Start* = 10 kHz, *End* = 10 MHz, *Amplitude* = Lin and *Range* = 2V/DIV. Press *Enter*.
10. Restart the simulation. When the oscillation has stabilized, drag the red marker to the position of the spectrum line observed. Note the frequency in the lower left corner of the Spectrum Analyzer window:
    f_c = _______________
11. Calculate L_1 necessary to achieve a frequency of oscillation of 8 MHz. Replace L_1 by double-clicking on it and selecting Replace. Run the simulation to verify your calculation.
12. Given that g_m = 1.6 *ms* and r_d = 12 kΩ, determine whether oscillations will be sustained.

Expected Outcome

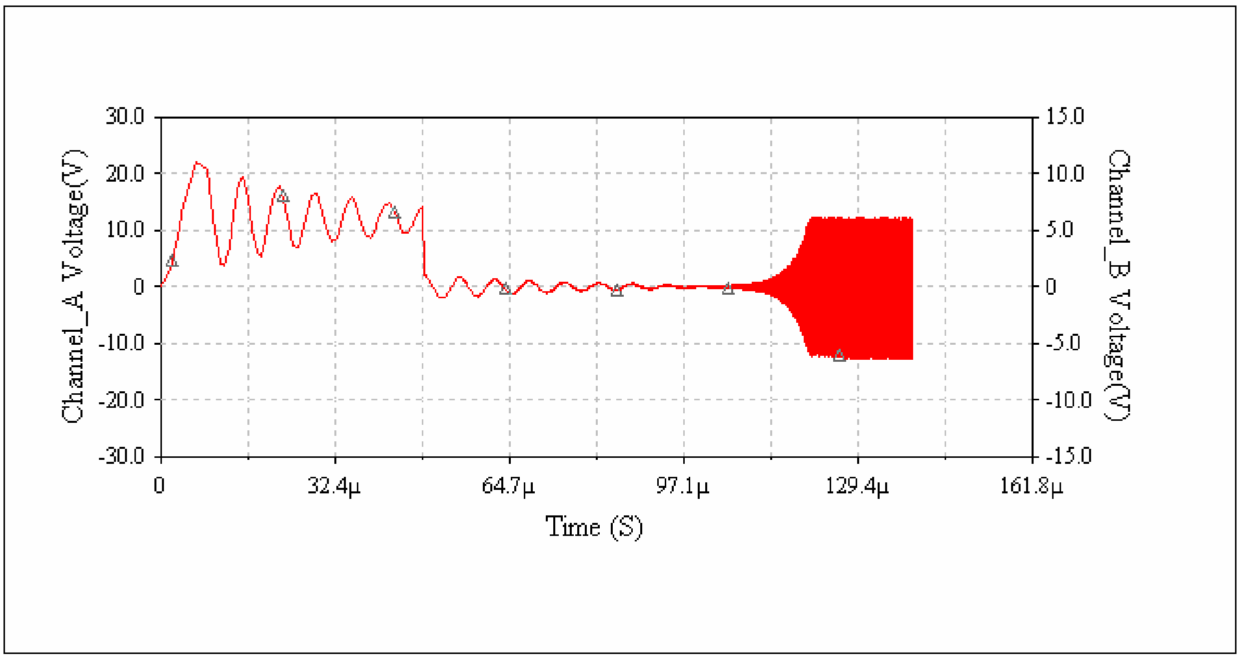


Figure 8-2 Oscilloscope Display of Initial Colpitts Oscillator Oscillations

Additional Challenge

Re-design the circuit of Figure 8-1 choosing values of C_1 and C_2 so that Avβ = 10 and the frequency of oscillation is approximately 3 MHz. Replace existing simulated component values by double-clicking on the component of interest. Run the simulation and compare the output data with expected theoretical values.

Experiment 9: The Clapp Oscillator

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a Clapp oscillator. The Clapp oscillator is much like a Colpitts oscillator with the capacitive voltage divider producing the feedback signal. The addition of a capacitor C in series with the inductor L_1 results in the difference in the two designs and is what makes the Clapp Oscillator unique. As with all oscillators, the Barkhausen criteria must be adhered to requiring a total gain of one and a phase shift of zero degrees from input to output.

Ignoring the transistor capacitive effect between the base and collector, the resonant frequency may be calculated using the equivalent capacitance:

$$C_{EQ} = \frac{1}{\frac{1}{C} + \frac{1}{C_1} + \frac{1}{C_2}}$$

But since C is typically much smaller than C_1 and C_2, the effects of C_1 and C_2 become negligible and:

$$f_r = \frac{1}{2\pi\sqrt{LC}}$$

As stated above, it is the addition of and the small value of C that creates the Clapp oscillator's unique characteristic of not being influenced by stray and transistor capacitances which would otherwise alter the values of C_1 and C_2. This results in a much more stable oscillator whose accuracy is dependable. The range of frequency of operation is limited in a Clapp oscillator but nevertheless, its reliability makes it a popular design. C_1 and C_2 may be adjusted for optimum feedback. The frequency of oscillation is altered through the adjustment of C.

Parts

DC 10 V Supply
Transistor: BJT 2N4401
Resistors: 20 kΩ, 3.9 kΩ, 1.2 kΩ
Inductor: 2.4 mH, 68 μH
Capacitor: 12 nF, 750 pF, 3.9 nF, 120 pF

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Frequency of Oscillation

$$fc = \frac{1}{2\pi\sqrt{LC}}$$

Equation 9-1

Procedure

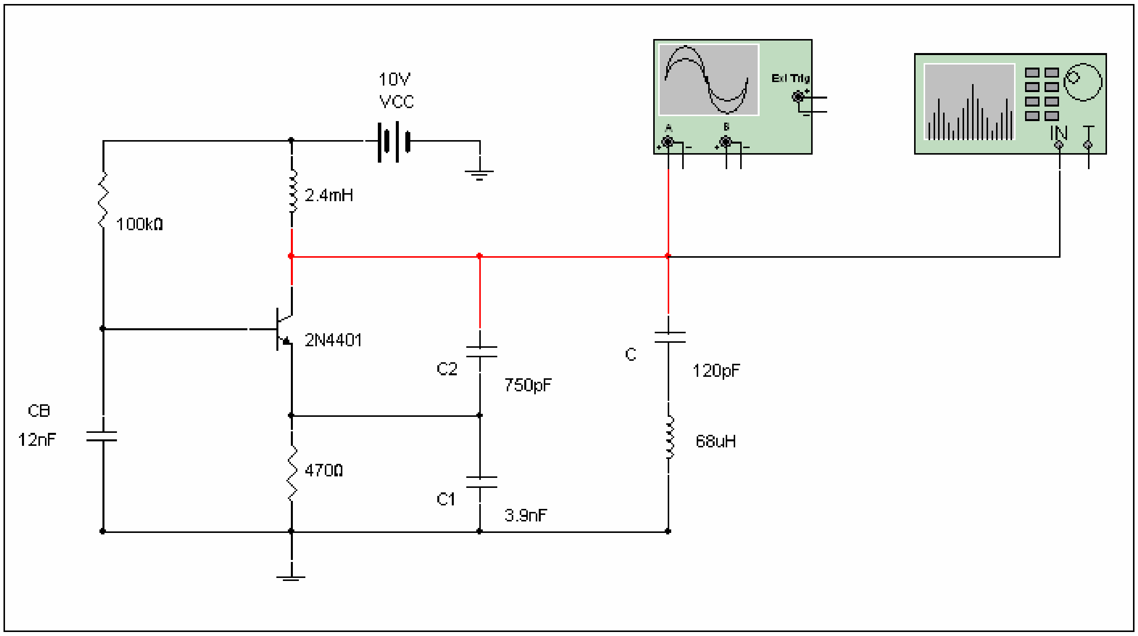


Figure 9-1

1. Connect the circuit components illustrated in Figure 9-1.
2. Double-click the Oscilloscope to view its display. Set the time base to 500 ms/Div and Channel A to 5 V/Div. Select Auto triggering and DC coupling. Set to AC coupling.
3. Select Simulate/Interactive Simulation Settings, and select Set to Zero for Initial Conditions. Check maximum time step and set to 3.6 e-008.
4. Start the simulation. The oscillator will require about 20 seconds to stabilize. Measure the oscillation frequency. Calculate the value of C necessary to achieve an oscillation frequency of 2 MHz. Change the value of C by double-clicking on it and run the simulation to verify your results.
5. Compare data with theoretical calculations and complete Table 9-1.
6. Stop the simulation and place a Spectrum Analyzer on the workspace.
7. Connect the output lead of the oscillator to the input of the Spectrum Analyzer.
8. Double-click to open the Spectrum Analyzer window.

9. Press *Set Span*. Set *Start* = 1 MHz, *End* = 4 MHz, *Amplitude* = LIN and *Range* = 1V/DIV. Press *Enter*.
10. Restart the simulation. When the oscillation has stabilized, drag the red marker to the position of the spectrum line observed. Note the frequency in the lower left corner of the Spectrum Analyzer window.
    f_c = _______________

Expected Outcome

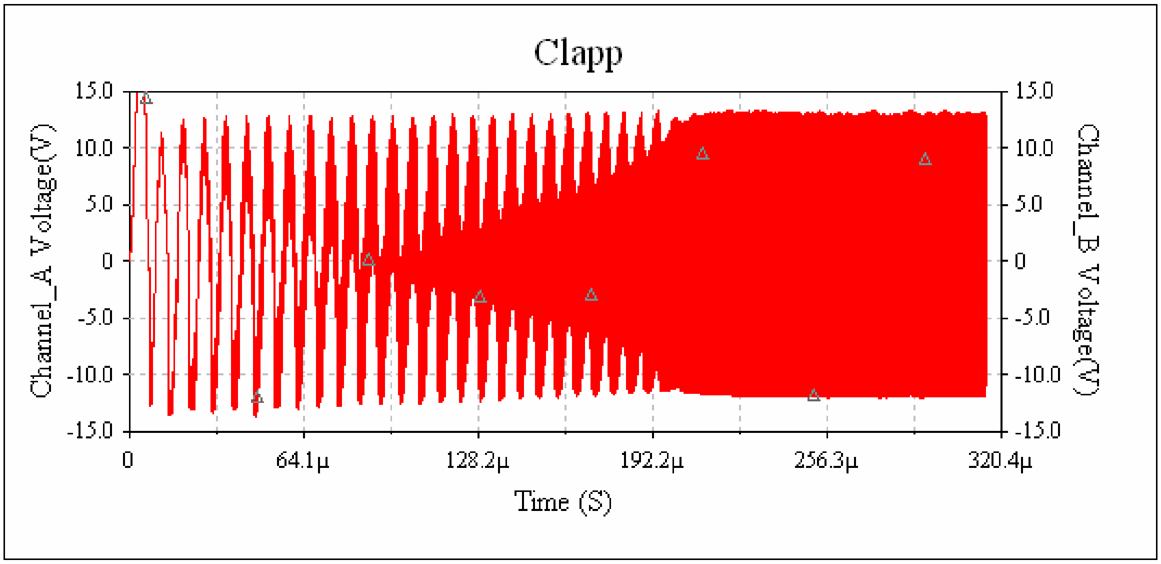


Figure 9-2 Oscilloscope Display of Initial Clapp Oscillations

Data for Experiment 9

	Measured Value	**Calculated Value**
f_c (step 2)		
f_c (step 3)		

Table 9-1

Additional Challenge

Replace C with a variable capacitor. Highlight C, right-click and choose delete. Select a variable capacitor from the parts bin and set a value of 120 pF. Highlighting, then pressing “a” or “A” will alter its capacitance ratio. Determine the upper frequency limit possible through the varying of C.

Experiment 10: Introduction to AM Communications

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of amplitude modulation. The desire for humans to communicate with each other is increasingly prevalent as we move into the global communication platform of the 21st century. In fact, the most astounding technological advances that took place in the 20th century were in the field of communications, with RF communications demanding its share of the marketplace.

Amplitude modulation (AM) communication is a method by which voice and music is combined or modulated with a high frequency radio signal and transmitted via a transmission medium before it is demodulated by a receiver. Amplitude modulation is the process which is responsible for the combining of the audio or message signal with the high frequency carrier before transmission takes place. Audio information requires a carrier frequency of some kind for two reasons. The range of hearing of the human ear is 20 - 20 kHz, relatively low in frequency. Information being transmitted in the audio range would require a far longer antenna than that of a high frequency carrier. This is due to the requirement that antenna length should be one half of the frequency wavelength. Since wavelength = c/f_c, a 2 kHz signal would require an antenna length of $c/2f_c = 3 \times 10^5 /2(2 \times 10^3) = 75$ km. which of course is not a realistic length. If this same 2 kHz signal were to be combined with a high frequency carrier of 1050 kHz, the antenna length would be reduced significantly. This would factor out in terms of cost of transmission.

The second reason why a carrier is used is discovered when separation of information is required at the receiver. When you tune your radio in order to select a station of choice, it is the carrier signal that provides the separation of information. Strict communication guidelines prevent overlapping of information through use of assigned high frequency carrier frequencies with tight bandwidths of 10 kHz for AM transmission. Assuming the information is a pure sinusoidal 2 kHz signal, the modulation process results in a lower sideband of $f_c - f_m$, f_c and upper sideband of $f_c + f_m$ resulting in the carrier frequency surrounded by an upper and lower sideband. This results in 3 spectral lines at the output of the AM modulator. With a 10 kHz bandwidth restriction, each sideband is limited to 5 kHz. This means that in practical terms, the bandwidth of the voice or music is limited to 5 kHz.

Multisim provides an AM modulator which we will use to produce a modulated signal from a low frequency message signal f_m and a high frequency carrier f_c. We will observe the modulated output in both time and frequency domains.

Parts

AM Modulator

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Lower Sideband

$$f_{lower} = f_c - f_m$$

Equation 10-1

Upper Sideband

$$f_{upper} = f_c + f_m$$

Equation 10-2

Procedure

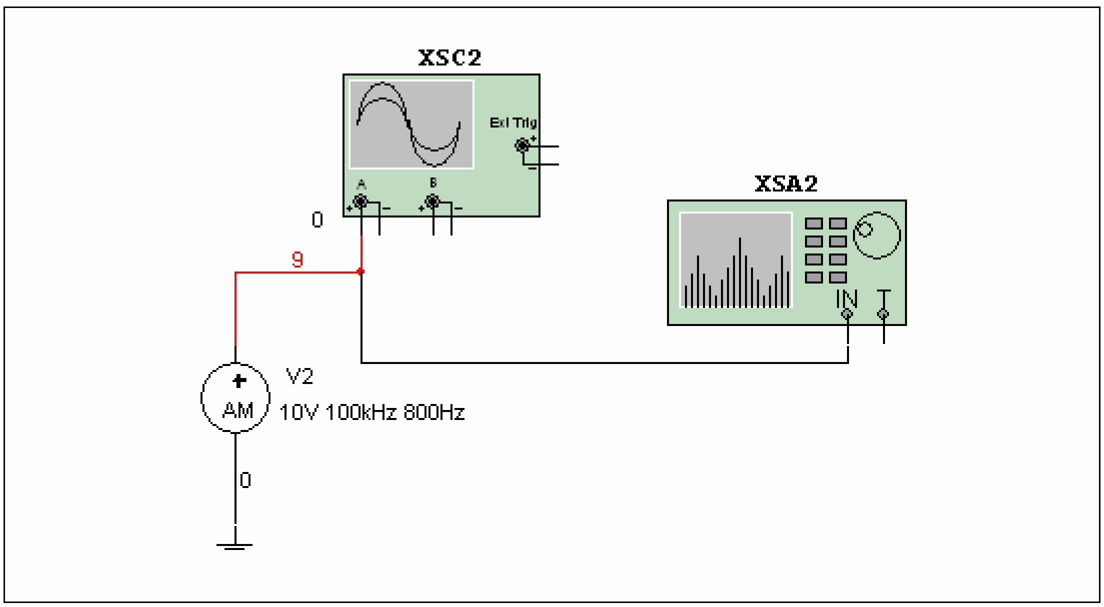


Figure 10-1 AM Modulator Example

1. Connect the circuit illustrated in Figure 10-1.
2. Double-click the AM Modulator. Select Carrier Amplitude = 10 V, Carrier Frequency = 100 kHz, Modulating Frequency = 800 Hz and Modulation Index = 0.6.

3. Double-click the Oscilloscope to view its display. Set the time base to 1 ms/Div and Channel A to 10 V/Div. Select Auto triggering and DC coupling.
4. Double-click the Spectrum Analyzer to view its display. Select *Set Span* and select *Span* = 10 kHz, *Center* = 100 kHz and *Amplitude* = LIN. Press *Enter*.
5. Start the simulation and draw the envelope complete with the carrier in the Data section of this experiment. Measure the modulating and carrier frequencies and verify the results with that of the AM modulator.
6. Double-click the Spectrum Analyzer. Observe the spectrum and use the red vertical marker to measure the frequency of the upper and lower sidebands as well as that of the carrier frequency. Record your results in the Data section of the lab. Verify your results with theoretical values.

Expected Outcome

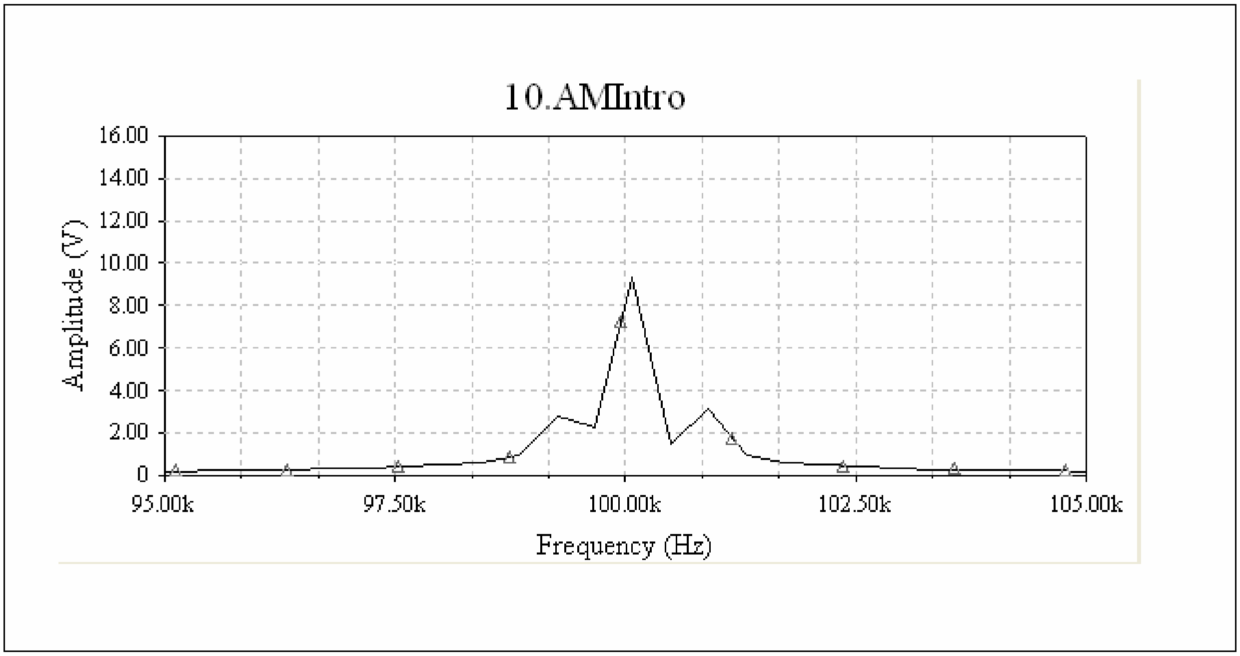


Figure 10-2 Frequency Spectrum of an AM Signal

Data for Experiment 10

Time versus Amplitude Sketch

	Measured Value	Calculated Value
f_{lower}		
f_{upper}		

Table 10-1

Additional Challenge

Double-click on the AM modulator to change the Carrier Frequency to 200 kHz, the Modulating Frequency to 500 Hz and the Modulation Index to 1. Run the simulation and observe the results in both time and frequency domains. Change the Modulation Index to 1.2 and observe the results on the Oscilloscope.

Experiment 11: Modulation Index and Power Considerations

Purpose and Discussion

The purpose of this simulation is to demonstrate the various characteristics of AM modulation measuring techniques and power efficiency in an AM spectrum. The modulation index of an amplitude modulated signal refers to the ratio of the amplitude of the carrier as compared with that of the message signal.

The modulation index or the percentage of modulation is an important part of the modulation process. An under-modulated AM signal is not an efficient means of transmitting information due to power considerations. At a modulation index of 1, both the upper and lower sidebands have amplitudes which are half of that of the carrier amplitude. Over modulation occurs with a modulation index of greater than one. Over modulation causes distortion at the receiver as well as interference with other stations due to the undesired sideband frequencies that are generated. Ideally, a modulation index of one is desirable. In practice, however, the envelope should be slightly under modulated to make allowances for diode clipping losses during the demodulation process.

The modulation index can be determined from the envelope itself which will be displayed on the oscilloscope. As the amplitude of the message signal is increased, extra lobes appear in the envelope. These lobes indicate over modulation. As the amplitude of the message signal is decreased below 100% modulation, measurements of the maximum and minimum envelope amplitude yield the modulation index:

$$m = \frac{v_{\mathrm{max}} - v_{\mathrm{min}}}{v_{\mathrm{max}} + v_{\mathrm{min}}}$$

The modulation index is defined as:

$$\frac{\textit{modulating signal amplitude}}{\textit{carrier signal amplitude}}$$

The percentage of modulation may be found by multiplying the modulation index by 100. The modulation index may also be determined from the frequency spectrum by measuring the difference in dB between the amplitude of the carrier and the amplitude of the sidebands. This difference is then used to calculate the modulation index:

$$m = 2/\,10^{[(\text{carrier dB - sideband dB})/20]}$$

Amplitude modulation is not an efficient means of transmitting voice and music information. At a modulation index of one, AM modulators are only 33% efficient. This is due to the sidebands only containing one third of the total power. The other two thirds are used up by the carrier, which carries no useful information. Other more efficient methods such as Double Sideband which suppresses the carrier and even

Single Sideband which suppresses the carrier and one sideband are sometimes used. The inherent problem with these two methods is the complexity of the receivers necessary to demodulate the signals. Amplitude modulation remains a popular modulation technique because the simplicity of the receivers involved renders them inexpensive and small in size.

The total power contained within an AM signal is the addition of the sideband power and the carrier power, both in mW:

$P_T = P_{USB} + P_{LSB} + P_C$

The transmission efficiency is a comparison of the useful power contained in the sidebands to the unused power contained in the carrier. The transmission efficiency may also be determined from the modulation index m. At:

$u = m^2/(2 + m^2)$,

and with a modulation index of 1, the maximum efficiency

$u = 1^2/(2 + 1^2) = 1/3$ can be calculated.

Parts

AM Source

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Modulation Index

$$m = \frac{v_{\max} - v_{\min}}{v_{\max} + v_{\min}}$$

Equation 11-1

$$m = 2/\,10^{[(\text{carrier dB - sideband dB})/20]}$$

Equation 11-2

$$m = \frac{\text{modulating signal amplitude}}{\text{carrier signal amplitude}}$$

Equation 11-3

Efficiency

$$\mu = \frac{P_{USB} + P_{LSB}}{P_T} = m^2/(2 + m^2)$$

Equation 11-4

Power in Watts

$$P_T = P_{USB} + P_{LSB} + P_C$$

Equation 11-5

$$P = V_{rms}^{\ 2} / R_L$$

Equation 11-6

$$P_T = P_C\,(1 = m^2/2)$$

Equation 11-7

Procedure

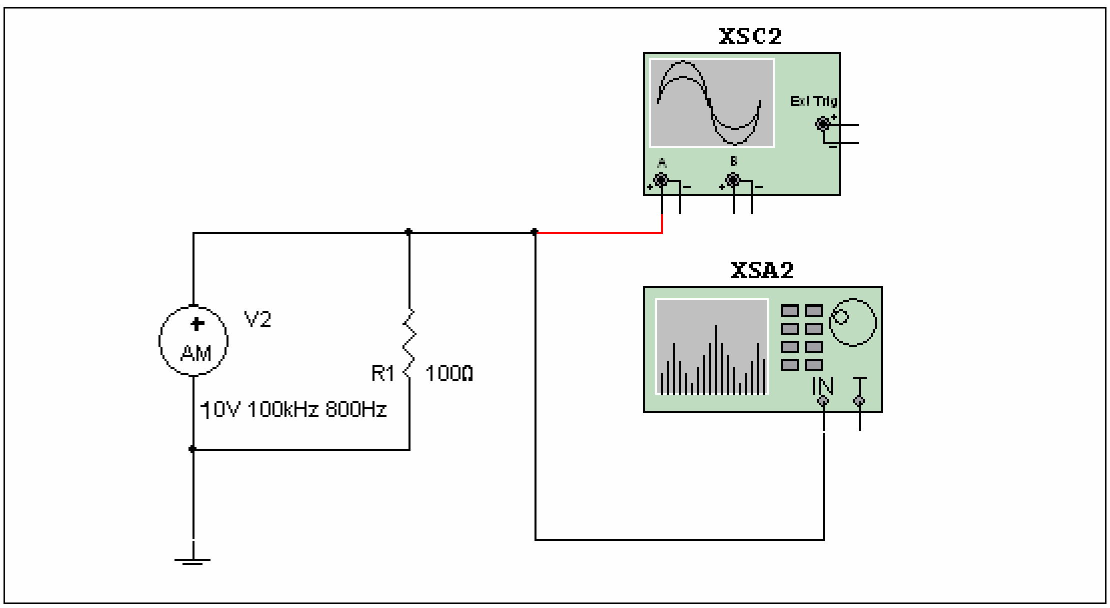


Figure 11-1 Modulation Index and Power Example

1. Connect the circuit illustrated in Figure 11-1.
2. Double-click the AM Source. Select Carrier Frequency = 100 kHz, Modulation Frequency = 800 Hz and Modulation Index = 0.6.
3. Double-click the Oscilloscope to view its display. Set the time base to 1 ms/Div and Channel A to 10 V/Div. Select Auto triggering and DC coupling.
4. Double-click the Spectrum Analyzer to view its display. Select *Set Span* and select *Span* = 10 kHz, *Center* = 100 kHz and *Amplitude* = Lin. Press *Enter*.
5. Start the simulation and double-click the Oscilloscope to view its display. Measure v_{max} which is represented by the maximum peak-to-peak amplitude of

the carrier in the modulated waveform. Measure v_{min} which is represented by the minimum peak-to-peak amplitude of the carrier in the modulated waveform. Calculate the modulation index and compare it with value of the AM Source modulation index. Record your results in Table 11-1.

6. Double-click the Spectrum Analyzer to view its display. Move the red vertical marker over the carrier frequency and measure its amplitude in dB. Measure the number of decibels of one of the sidebands using the same method. Calculate the modulation index and record your results in Table 11-1. Calculate the amplitude of the modulating signal and record your results in Table 11-1.
7. Repeat for a modulation index of 0.33.
8. Double-click the AM Source and set the Carrier Amplitude = 20 V, Carrier Frequency = 100 kHz, Modulation Frequency = 500 Hz and the Modulation Index = 0.5.
9. Restart the simulation and observe the envelope on the Oscilloscope and the spectrum on the Spectrum Analyzer. Using the Spectrum Analyzer, move the red vertical marker to each side band and the carrier, noting and recording the voltage levels. Calculate the power of each, then the total power P_T. Determine a measured value of μ.
10. Set the AM Source modulation index setting to 0.5 in order to determine a calculated value of the efficiency. Record your results in Table 11-2.
11. Change the modulation index to 0.7 and repeat. Record your results in Table 11-2. Repeat the procedure outlined above for a modulation index of 0.8 and 1. Record your results in Table 11-2.

Expected Outcome

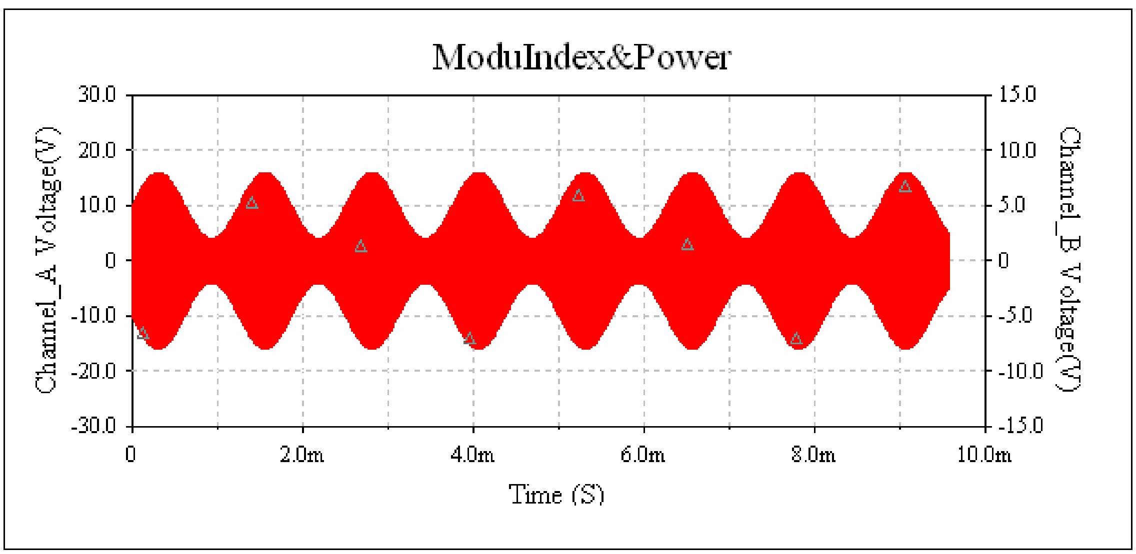


Figure 11-2 Oscilloscope Display of m = 0.5

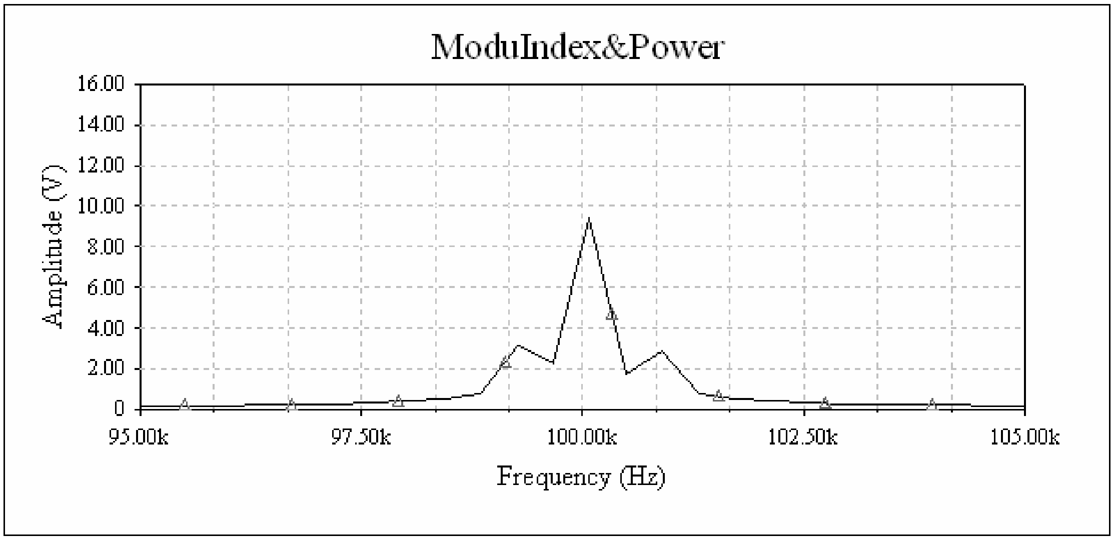


Figure 11-3 Frequency Spectrum of m = 0.5

Data for Experiment 11

	V_{max}	V_{min}	$m_{measured}$	carrier dB	sideband dB	$m_{measured}$	$m_{expected}$	%m	mod. Sig. Amplitude
m = 0.6									
m = 0.33									

Table 11-1

Efficiency

	Measured Value	Calculated Value
m = 0.5		
m = 0.7		
m = 0.8		
m = 1		

Table 11-2

Additional Challenge

Repeat step 2 for a modulation index of 0.80. Set the modulation index to 1.2 and observe the effects of over modulation on the Oscilloscope and Spectrum Analyzer. Set the modulation index to 0.6. Calculate the efficiency of an amplitude modulated signal using power measurements of the carrier and sidebands taken from the frequency spectrum. Use the Spectrum Analyzer in order to obtain your data and the formulae provided for your calculation.

Experiment 12: AM Signal Demodulation Techniques

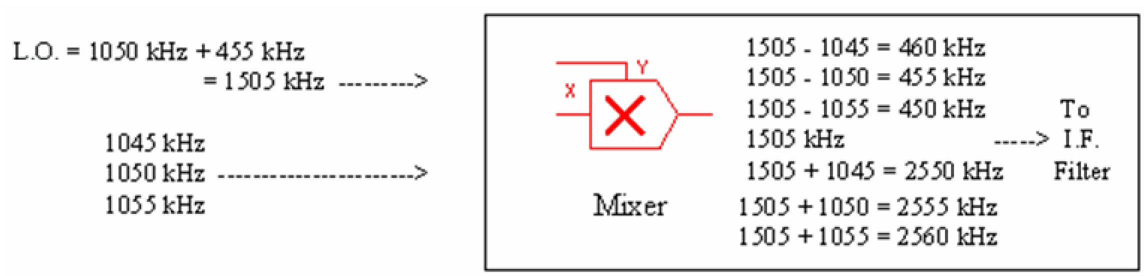
Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of an envelope detector, and to provide a comprehension of the stages that a modulated signal is subjected to at the receiver, so that the original transmitted information is recovered.

An AM signal, once received by a receiver, is subjected to several stages in the demodulation process. Figure 12-1 illustrates the final detection and filter stage using a simple diode detector. Other more complex detectors that use the popular PLL (phase-lock-loop) circuitry allow, together with AGC (automatic gain control) circuitry, modulation indexes of close to one.

Because the circuitry involved in the detection process is fixed, a fundamental requirement for a signal at the detector's input is that the sidebands are situated on either side of a fixed frequency. This fixed frequency is called the IF or intermediate frequency and is produced by the mixing of a local oscillator frequency with the RF spectrum which has been filtered in the RF stage of the demodulation process. The fixed value of the intermediate frequency is 455 kHz. This IF signal is applied to the input of a highly selective IF amplifier.

The local oscillator (LO) frequency in the popular superheterodyne receiver is adjusted through the tuning control to 455 kHz above the RF carrier, $f_{LO} = f_c + f_{IF.}$ Why is the LO necessary? Remember that the detector requires the message signal to be frequency translated to either side of a fixed intermediate frequency. Injecting the RF spectrum and the local oscillator frequency through a mixer will produce the sum and difference of the frequencies involved. It is the difference frequencies that produce the IF spectrum required. Consider a carrier frequency of 1050 kHz carrying a 5 kHz message signal.



The IF filter features steep roll off characteristics which reject all frequencies other than the IF frequency translated spectrum. The output of the filter constitutes the input to the detector.

The envelope detector of Figure 12-1 is designed to subject the signal to a half wave rectification process. The RC time constant should be such that the charge time is fast and the discharge time is slow. This will ensure that the detector follows the amplitude variations of the envelope. The RC time constant of the envelope detector should be designed such that:

$$RC = \frac{1}{2\pi\, m f_m}$$

Not shown in Figure 12-1 is the AGC circuitry which helps to control the level of the input to the detector.

One of the main drawbacks of the envelope detector is the effect of the diode voltage drop Vd. This 0.7 V drop represents a delay between the point where the signal reaches the input and where the capacitor is able to allow the output to react to the input. This ultimately results in power lost because the modulation index is restricted from reaching its optimum level of one. The detector of Figure 12-2 will detect modulation signals over a range of frequencies with the particular low pass filter portion supporting a cutoff frequency of 2 kHz for purposes of demonstration.

Parts

Resistors: 330 Ω, 620 Ω, 3.3 kΩ, 5.2 kΩ, 15 kΩ, 33 kΩ
Capacitors: 2 nF, 4.7 nF, 2.2 nF, 12 nF
Diode: 1N4148
Ideal Opamps
AM Modulator

Test Equipment

- Oscilloscope

Formulae

RC Time Constant

$$RC = \frac{1}{2\pi\, m f_m}$$

Equation 12-1

Procedure

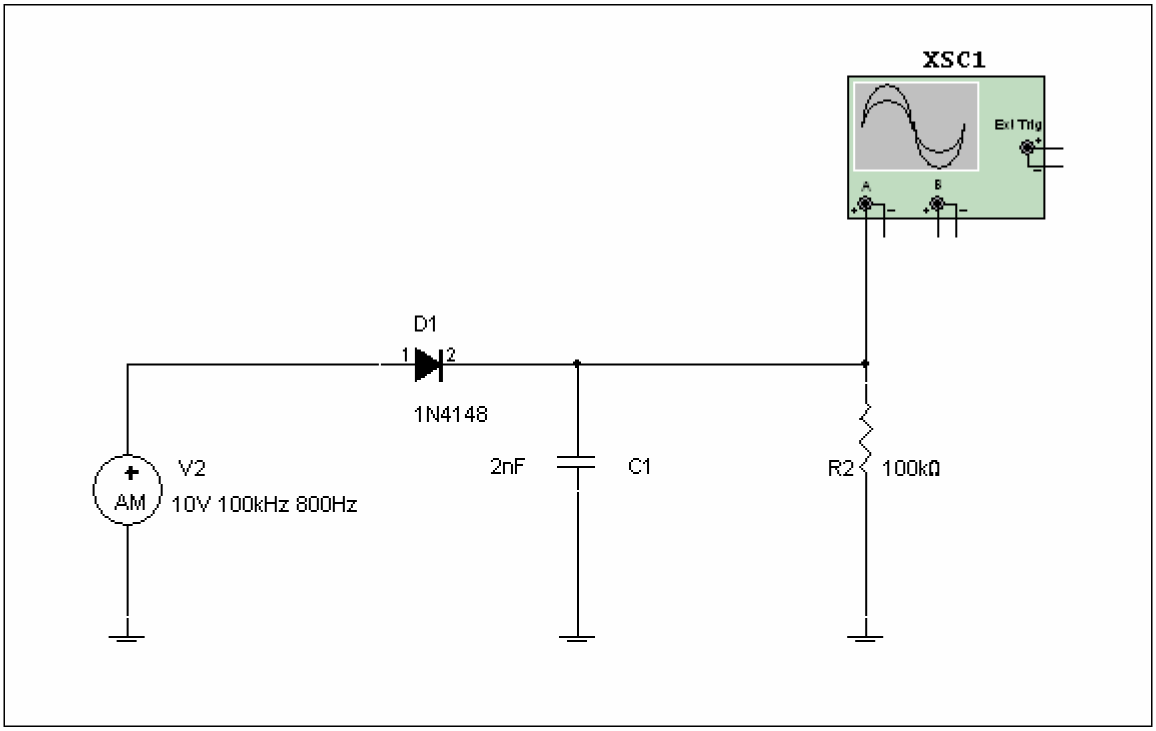


Figure 12-1 Envelope Detector Example

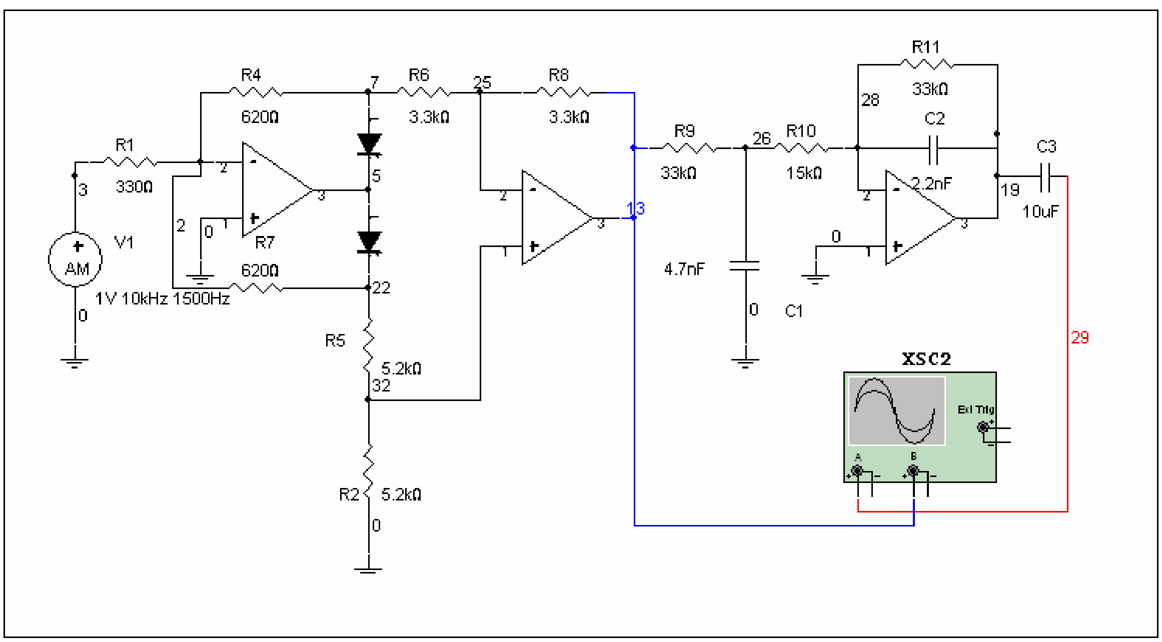


Figure 12-2 Diode Detector Example

1. Connect the circuit components illustrated in Figure 12-1.
2. Double-click the Oscilloscope to view its display. Set the time base to 1 ms/Div and Channel A to 10 V/Div. Select Auto triggering and DC coupling.

3. Double-click the AM Source to change its parameters. Set the carrier amplitude = 10 V, the carrier frequency = 100 kHz, the modulation index = 0.6 and the modulation frequency = 800 Hz.
4. Start the simulation and measure the frequency of the demodulated waveform and compare it with its expected value. Record your results in the Data section of this experiment.
5. Double-click on the resistor to change its value. Select a 500 kΩ resistor. Run the simulation again. Draw the waveform associated to a time constant which is too large. Next, replace the 500 kΩ resistor with a 10 kΩ resistor. Run the simulation and draw the waveform associated to a time constant which is too small.
6. Re-design the detector in order to provide optimum detection for a 500 Hz modulating signal. Replace the components, re-set the AM Source modulating frequency parameter and run the simulation.
7. Connect the circuit components illustrated in Figure 12-2. Connect both Oscilloscope channels as shown. Set the time division to 500 μs/Div, Channel A to 500 mV/Div and Channel B to 5 V/Div. Set the AM Source as indicated in Figure 12-2. Run the simulation. Note your observations.

Expected Outcome

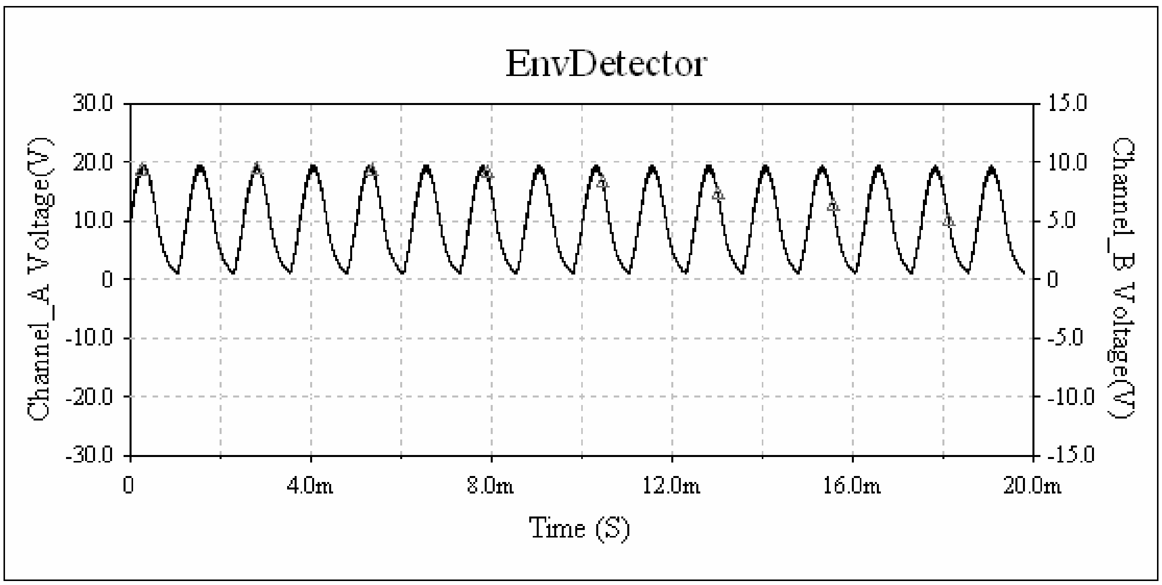


Figure 12-3 Output of Envelope Detector at m = 0.6

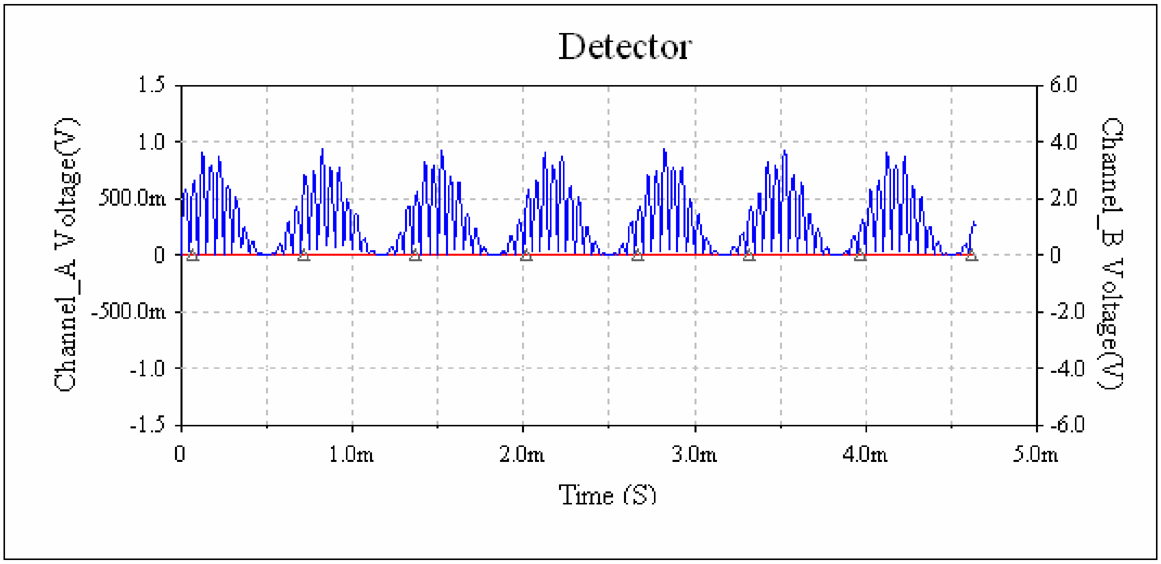


Figure 12-4 Output of Detector Stage of Figure 12-2

Data for Experiment 12

f_m at output of detector _______________

f_m expected _________________________

Waveform of an RC time constant which is too large

Waveform of an RC time constant which is too small

From step 5, re-designed value of R = ________ *and C =* _______________.

Step 6 ____.

Additional Challenge

Double-click on the AM Source of Figure 12-1 to change its modulation index parameter to 1. Run the simulation and note the difference in the waveform at the output of the detector. Change the modulation index to 1.4. Run the simulation and note the difference in the waveform at the output of the detector.

Experiment 13: Double Sideband Analysis

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of Double Sideband (DSB) transmission. As discussed previously, the purpose of the RF carrier frequency is to carry the information to its destination, radio receivers within its restricted physical transmission perimeter. In order to accomplish this in AM transmission, both sidebands are sent together with the carrier frequency. Since the carrier uses 2/3 of the total power necessary to send the signal to its restricted area, and power costs money, the suppression of the carrier frequency is a cost effective means of transmitting a signal. In Double Sideband transmission, this suppressed carrier is then reconstructed and re-inserted in the receiver. It should be noted that Single Sideband (SSB) transmission suppresses both the carrier and one sideband before transmission. The problem with both DSB and SSB is the required complexity of the receivers necessary to demodulate the signals. Because the phase of the reconstructed carrier frequency must be the same as the phase of the suppressed carrier, a pilot carrier is used in DSB transmission.

In our analysis, a modulating signal and a carrier will be applied to a multiplier which will be used to simulate a suppressed carrier output.

Parts

Resistor: 1 kΩ
Multiplier (found in the Control Function Blocks parts Family)
AC Voltage Source (2)
AM Modulator

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Power in Watts

$$P = V_{rms}{}^{2} / R_L$$

Equation 13-1

Procedure

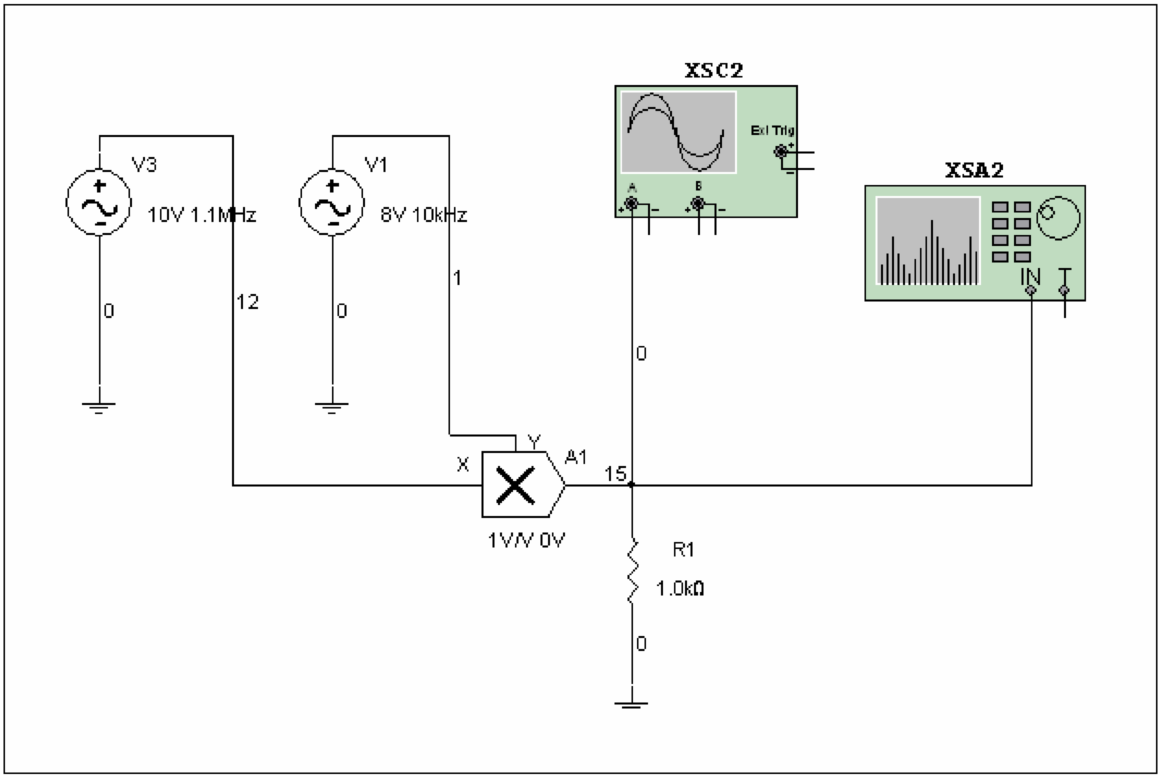


Figure 13-1 Double Sideband with Multiplier Example

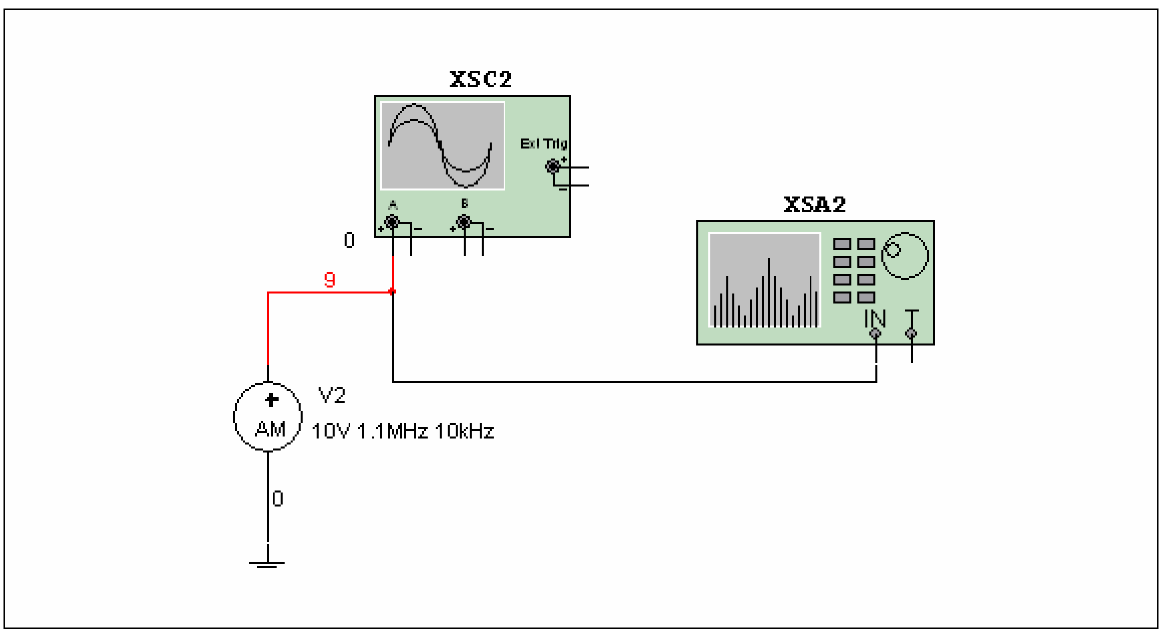


Figure 13-2 AM Modulator Example

1. Connect the circuit components illustrated in Figure 13-1.
2. Double-click the first AC Source and set the carrier amplitude = 10 V and the carrier frequency = 1.1 MHz. Double-click the second AC Source and set the modulating frequency amplitude = 8 V and the modulating frequency = 10 kHz.
3. Double-click the Oscilloscope to view its display. Set the time base to 20 µs/Div and Channel A to 50 V/Div. Select Auto triggering and DC coupling.
4. Double-click the Spectrum Analyzer to view its display. Select *Set Span*. Select *Start* = 980 kHz, *End* = 1.2 M Hz, *Range* = 20 V/DIV, *Resolution Frequency* = 3.906 kHz and *Amplitude* = LIN. Press *Enter*.
5. Start the simulation and double-click on the Oscilloscope to view the DSB output and verify its waveform with that of Figure 13-3.
6. Double-click on the Spectrum Analyzer to view the associated frequency spectrum. Measure the frequency of each of the sidebands by moving the red vertical marker over each sideband and noting its frequency in the Spectrum Analyzer window. Verify that they are the same as your expected values. Record your data in Table 13-1.
7. Note the voltage levels of the sidebands analyzed above. Calculate the power of each sideband. Record your data in Table 13-2.
8. Connect the circuit shown in Figure 13-2. This is an AM Modulator. Set the modulation index = 1. This will represent the best power efficiency possible for this modulator. Confirm that all other AM modulator circuit parameters are the same as Figure 13-1.
9. Double-click the Spectrum Analyzer. Set *Span* = 100 kHz and *Center* = 1.1 MHz. Press *Enter*.
10. Run the simulation and measure the voltage of each sideband. Calculate the power of each and compare with your DSB data. Tabulate your results in Table 13-2. Comment on your observations.

Expected Outcome

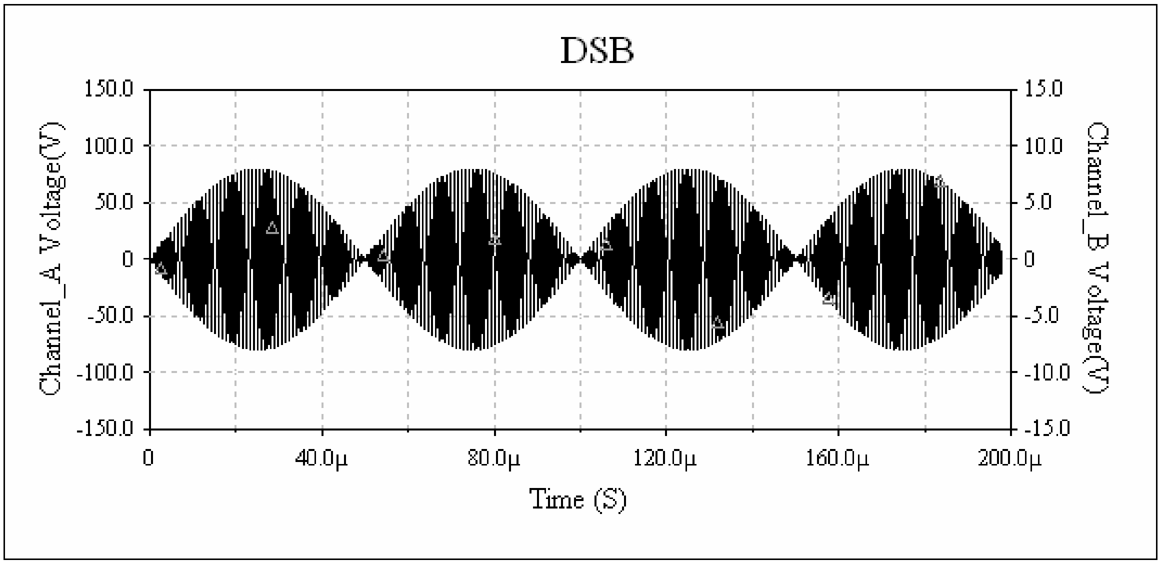


Figure 13-3 Oscilloscope Display of a Simulated DSB Signal

Data for Experiment 13

	Measured Value	Calculated Value
Frequency of lower sideband		
Frequency of upper sideband		

Table 13-1

	LSB (V)	USB (V)	LSB (mW)	USB (mW)
DSB				
AM				

Table 13-2

Additional Challenge

Repeat steps 2 through 6 for a modulating frequency of 12 kHz and a carrier frequency of 1 MHz. Double-click on the AC Sources in order to change their parameters. Re-adjust the Oscilloscope and Spectrum Analyzer parameters in order to best display the data. Run the simulation and verify your results.

Experiment 14: Introduction to FM Modulation and Detection

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of a simple frequency modulator and detector. In amplitude modulation we observed the amplitude of the modulated wave vary with changes in the message signal. In frequency modulation (FM), increases in the carrier frequency result when the modulating signal amplitude is increased. The FM modulator used in this experiment is made up of Multisim's Voltage Controlled Sine Wave Oscillator. This sine wave generator acts as a voltage controlled oscillator (VCO). This generator is driven by a 500 Hz modulating frequency which is supplied by the Function Generator as shown in Figure14-1. The Voltage Controlled Sine wave parameters are set such that a zero volt control voltage produces an output frequency of 200 Hz. A control voltage of 12 V will result in an output frequency of 25 kHz.

The detector circuit of Figure 14-2 is an example of a simple FM demodulator which makes use of a crystal found in the miscellaneous section of the parts bin. This circuit will provide basic demodulation for a signal provided by Multisim's FM modulator and will output a much lower amplitude version of the original modulating signal. This signal normally would be amplified by an audio amplifier for practical purposes.

Parts

Resistors: 50 Ω, 220 Ω, 1 kΩ, 50 kΩ
Capacitors: 35 nF, 51 nF, 1.5 μF, 15 μF, 20 μF, 30 μF
Inductors: 12 μH, 30 μH, 3 mH
Ideal Crystal
Ideal Diodes
FM Modulator

Test Equipment

- Oscilloscope
- Function Generator

Procedure

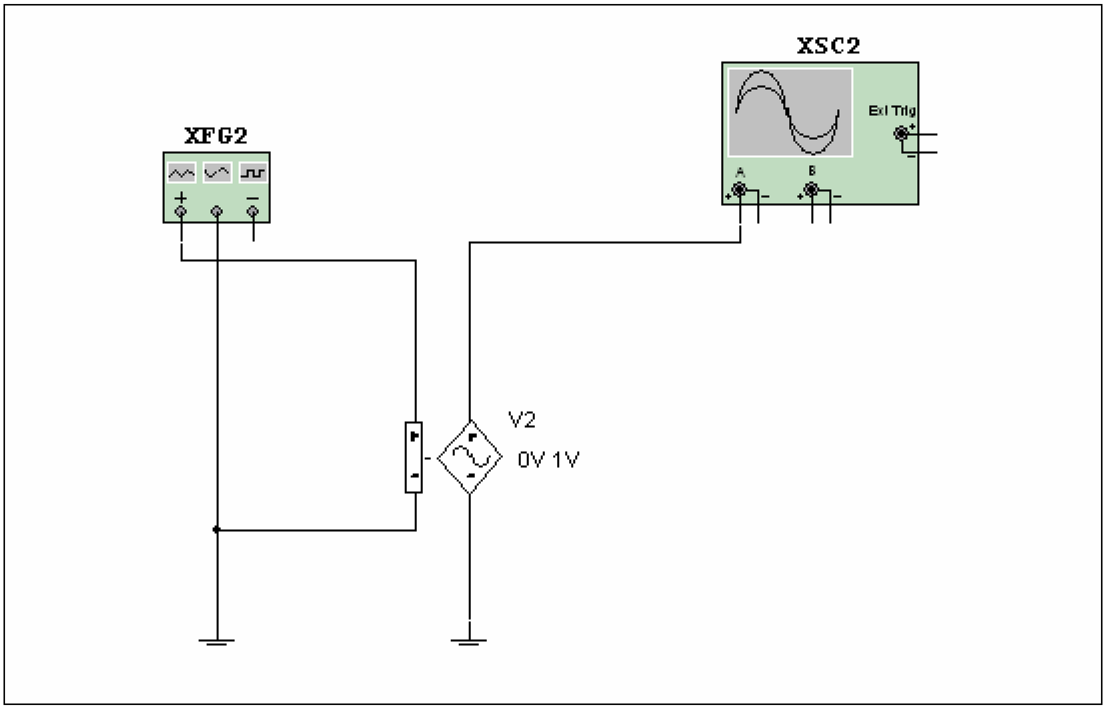


Figure 14-1 VCO Detection Example

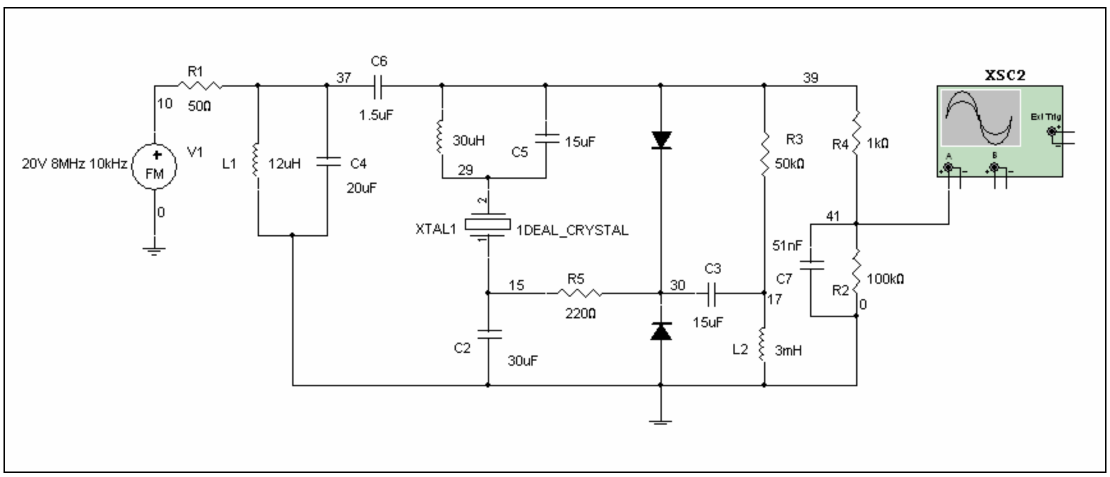


Figure 14-2 Simple FM Demodulator Example

1. Connect the circuit as illustrated in Figure 14-1.
2. Double-click the Function Generator to set its parameters. Select Triangle Wave, Frequency = 500 Hz, Duty cycle = 50 %, Amplitude = 6 V and Offset = 7 V.
3. Double-click the Voltage Controlled Sine wave generator to set its parameters. Set the first Control Array selection = 12 with an associated Frequency Array = 25000. Set the second Control Array selection = 0 with an associated Frequency Array = 200.

4. Start the simulation and observe the frequency modulated signal in the time versus amplitude domain. Draw the associated output waveform in the Data section of this experiment.
5. Connect the circuit as illustrated in Figure 14-2.
6. Double-click the FM Modulator to set its parameters and set Voltage Amplitude = 20 V, Carrier Frequency = 8 MHz, Modulation Index = 5 and Signal Frequency = 10 kHz.
7. Double-click the Oscilloscope to view its display. Set the time base to 100 µs/Div and Channel A to 100 µV/Div. Select Auto triggering and DC coupling. Run the simulation noting the frequency of the sine wave at the output of the detector. Compare this frequency with the input modulation frequency to verify that they are the same.

Data for Experiment 14

FM time versus amplitude sketch

f_m = ____________________

Additional Challenge

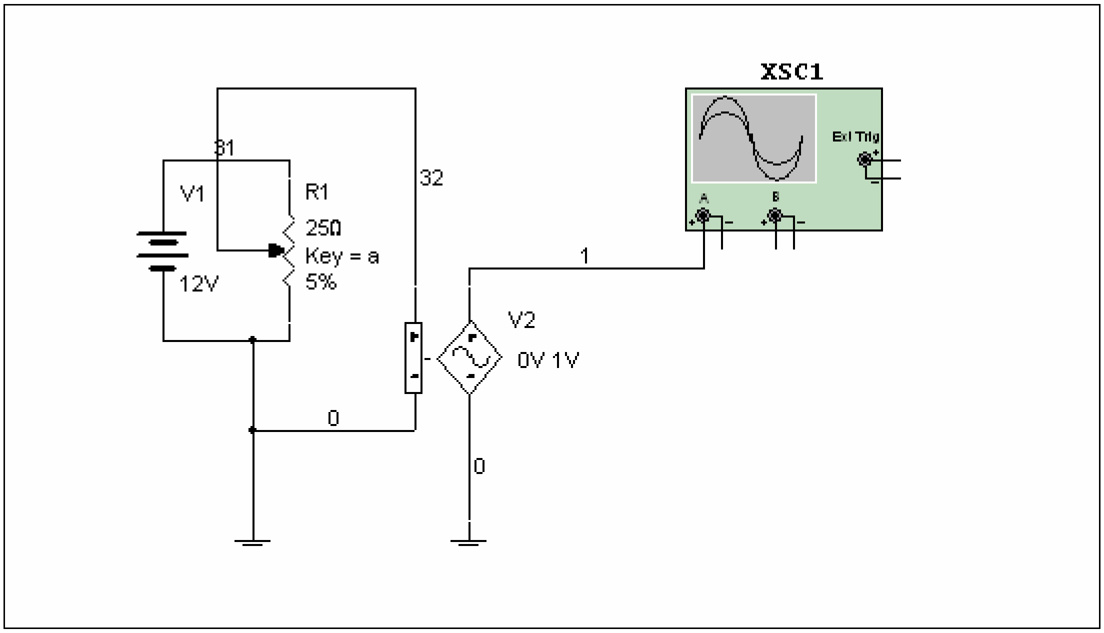


Figure 14-3 VCO Using a Potentiometer as a Modulating Signal Example

Connect the circuit as illustrated in Figure 14-3. In this configuration, the VCO is controlled by your operation of the potentiometer which will be simulating a low frequency message signal. Run the simulation and, while observing the output on the Oscilloscope, highlight the potentiometer and press the “A” key continuously. You can also, hover your cursor over the potentiometer and drag the slider bar that appears. Observe the simulated frequency modulated output that results.

Experiment 15: Frequency Modulation

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of frequency modulation using Multisim's Frequency Modulator. In frequency modulation (FM), variations in the frequency of the modulated wave are observed with changes in the message signal. Amplitude modulation is easily affected by noises in the atmosphere as well as any other interference from sources of close proximity that generate frequencies in the range of the modulating signal. These spurious frequencies ride upon the modulated signal even after demodulation and appear as static. Noise also amplitude modulates FM signals but since the modulating relationship results in frequency variations, not amplitude variations, the interference has virtually no affect and is easily removed in the demodulation process. Even when the noise falls between the sine waves of the carrier, thus potentially affecting the frequency, almost complete noise suppression is possible by ensuring a large carrier deviation. Frequency deviation is the amount that the frequency deviates from that of the carrier frequency. FM broadcasting limits the maximum frequency deviation to 75 kHz.

A large amplitude modulating signal results in a large maximum frequency deviation. A low amplitude modulating signal results in a small maximum frequency deviation. The amplitude of the modulated wave is not affected by the amplitude of the message signal. The modulation index m is established by the maximum frequency carrier deviation divided by the frequency of the modulating signal which produces the deviation $m = \Delta f/f_m$.

Since amplitude variations directly affect frequency deviation in FM, it follows that a lower frequency modulating signal will cause a slower rate of frequency deviation since the FM signal will be subjected to less message signal amplitude variations per second than that of a higher frequency message signal. Hence, it is important to note that the rate of frequency deviation is dependant upon the frequency of the message signal. FM broadcasting regulations limit the maximum audio frequency to 15 kHz. The bandwidth W of an FM signal is dependant on the number of sideband pairs which are not more than 20 dB down from the height of the highest spectral line. These sideband pairs are the most significant and represent approximately 98% of the total power. The number of sideband pairs is dependant on the modulation index and its value is predictable. For example, for a modulation index of 3, six significant sideband pairs are expected.

Associated time domain variations can be observed using the oscilloscope. Recall that in the frequency spectrum of an AM wave, two sideband frequencies are produced for every modulating signal. One sideband frequency is equal to $f_c - f_m$ and is below the carrier frequency. The other sideband is equal to $f_c + f_m$ and is above the carrier frequency. In FM, however, in addition to the basic pair of sideband frequencies produced by a single modulating signal, an infinite number of sideband frequencies are also produced. $f_c - 3f_m$, $f_c - 2f_m$, $f_c - f_m$, f_c, $f_c + f_m$, $f_c + 2f_m$, $f_c + 3f_m$ are only a few of the spectral lines that are observed in the frequency domain. The spectral lines decrease in power as they move further from the center frequency. As the modulation index is increased, the power is distributed over more spectral lines.

Parts

FM Modulator

Test Equipment

- Oscilloscope
- Spectrum Analyzer

Formulae

Bandwidth

6 significant sideband pairs @ m = 3,

$W = 2(\#\ \text{significant sideband pairs})f_m$

Equation 15-1

Modulation Index

$m = \Delta f/f_m$

Equation 15-2

Procedure

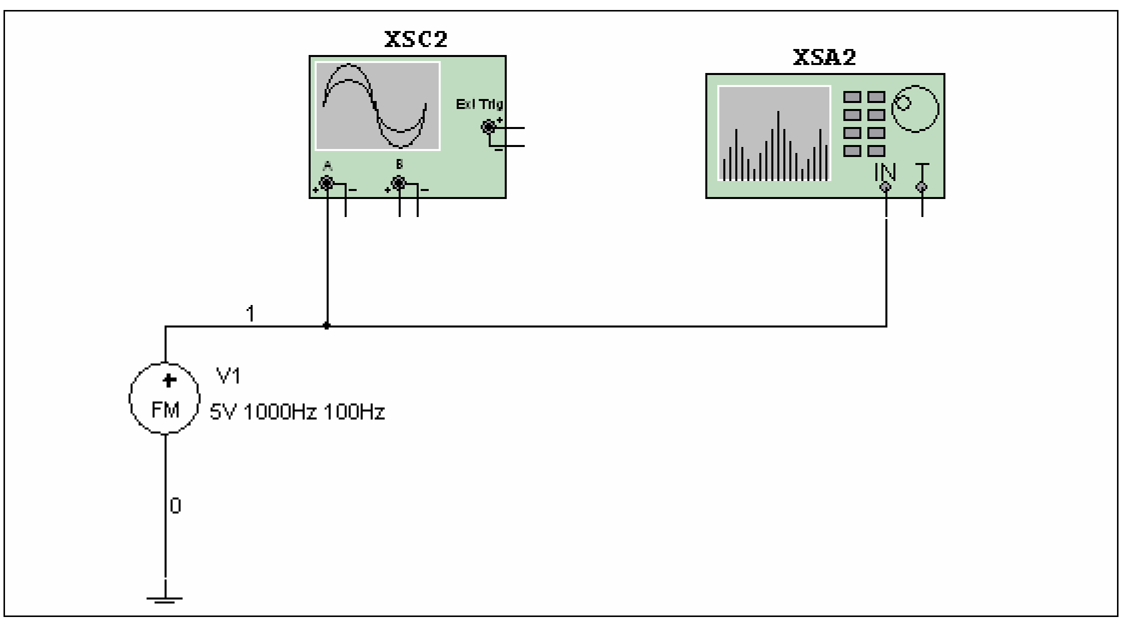


Figure 15-1 VCO AM Modulator Example

1. Connect the circuit as illustrated in Figure 15-1.
2. Double-click the FM Modulator to set its parameters. Set Voltage Amplitude = 10 V, Carrier Frequency = 100 kHz, Modulation Index = 5 and Signal Frequency = 10 kHz.
3. Double-click the Oscilloscope to view its display. Set the time base to 20 us/Div and Channel A to 10 V/Div. Select Auto triggering and DC coupling.
4. Start the simulation and observe the frequency modulated signal in the time versus amplitude domain. Draw the associated output waveform in the Data section of this experiment.
5. Double-click the FM Modulator and change the Modulation Index to 3.
6. Double-click the Spectrum Analyzer to view its display. Select *Set Span.* Set *Span* = 125 kHz and *Center* = 100 kHz. Press *Enter*. This will allow us to view the carrier frequency along with several sideband sets. Calculate and note the expected frequency deviation.
7. Observe the frequency spectrum. Use the red vertical marker to locate the carrier frequency of 100 kHz. Verify that the upper and lower sideband frequencies correspond with f_c - $3f_m$, f_c - $2f_m$, f_c - f_m, f_c, f_c + f_m, f_c + $2f_m$, f_c + $3f_m$ etc. Move the red marker over the carrier to determine its position.
8. Noting the amplitude of each spectral line, find the highest line to the right of the carrier. Note its amplitude and count the number of spectral lines to the right of the carrier which are no more than 20 dB down from the highest line. Calculate the bandwidth W in the Data section of this experiment.
9. Change the modulation index to 5, run the simulation and observe the spectrum. Change the modulation index to 1.5 and describe what you are observing.

Expected Outcome

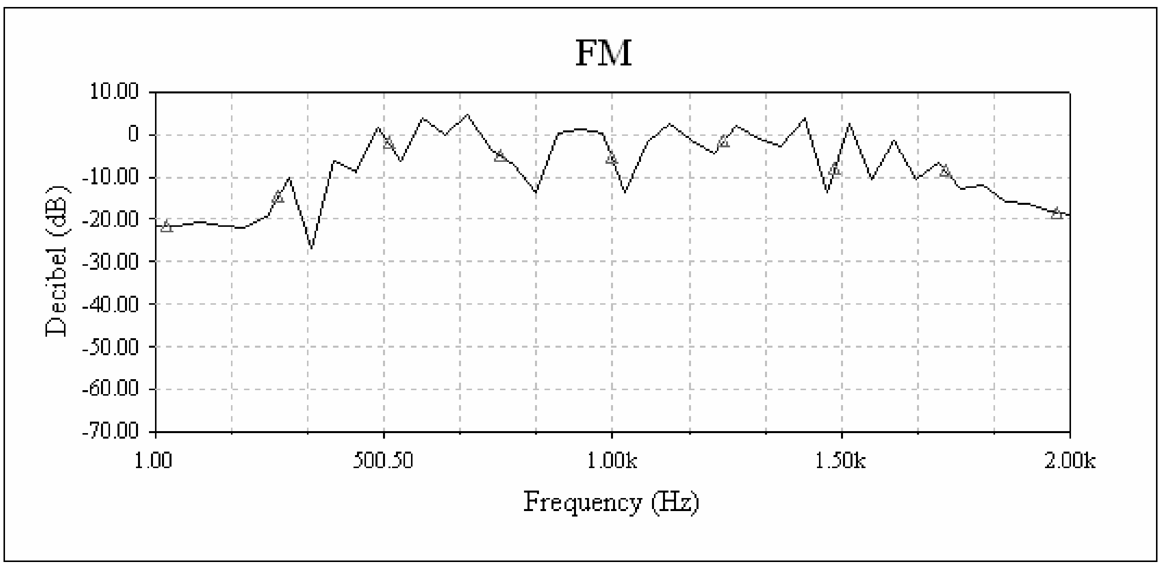


Figure 15-2 Frequency Spectrum of an FM Signal

Data for Experiment 15

FM time versus amplitude sketch

Δf @ (m = $3_{calculated}$, f_m = 10 kHz) = ____________________

Bandwidth W @ (m = 3, f_m = 10 kHz) = __________________

Describe the difference in the frequency spectrum between a modulation index of 5 and a modulation index of 1.5:

____

____

Additional Challenge

Double-click on the FM Modulator and change the modulation index to 2.4. Run the simulation. Describe the spectrum and explain the characteristics noted (refer to Bessel Coefficients for your explanation).

Experiment 16: DC and AC Operating Point Analysis of an RF Amplifier

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics and operation of an RF amplifier using DC and AC analysis in the course of our study. Radio frequency amplifiers perform the function that their name implies. They select and amplify a narrow band of radio frequency signals. Their various properties are utilized in many areas of RF communications. Power amplifiers, voltage amplifiers, buffer amplifiers and frequency multipliers are all RF amplifiers employed in the electronics industry. Figure 16-1 is an example of a radio frequency amplifier used to amplify a signal in the range of 1.5 MHz.

In order to specify the DC voltages around the circuit, we will be employing Multisim's DC Operating Point Analysis. In this analysis, all AC sources are set to zero and all capacitors are opened. Next, we will employ Multisim's AC Analysis in order to obtain a sketch of the output voltage over a specified frequency range. These features provide the designer with a quick overview of the performance of the circuit design resulting in minimum to no necessary manual calculations.

Parts

Resistors: 10 kΩ, 1 kΩ, 5 kΩ, 50 kΩ
Inductor: 125 μH
Capacitors: 80 pF, 1 μF
BJT NPN Transistor IDEAL
AC Voltage Source

Test Equipment

- Oscilloscope

Formulae

DC Analysis

$$V_{BB} = \frac{R_2}{R_1 + R_2} V_{cc}$$

Equation 16-1

$$I_B = \frac{V_{BB} - V_{BE}}{R_{BB} + (\beta + 1) R_E}$$

Equation 16-2

$$I_C = \beta I_B$$

Equation 16-3

AC Analysis

$$f_c = \frac{1}{2\pi\sqrt{LC}}$$

Equation 16-4

$$Z_{||} = \frac{L}{RC}$$

Equation 16-5

$$Q = \frac{Z_{||}}{X_L}$$

Equation 16-6

$$BW = \frac{f_c}{Q}$$

Equation 16-7

Procedure

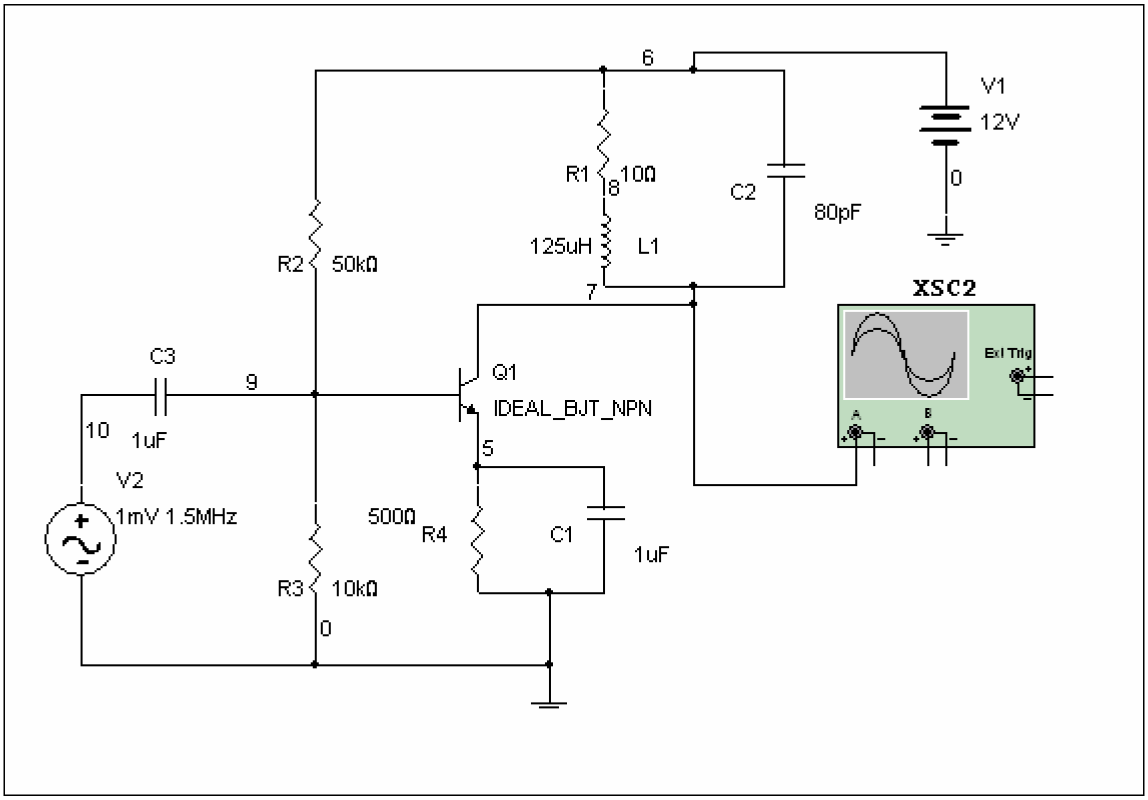


Figure 16-1 RF Amplifier Example

1. Connect the circuit components illustrated in Figure 16-1.
2. Double-click the Oscilloscope to view its display. Set the time base to 1 µs/Div and Channel A to 5 mV/Div. Select Auto triggering and DC coupling. Select Simulate/Interactive Simulation Settings and choose Automatically determine initial conditions under Initial Conditions.
3. Start the simulation and measure the frequency of oscillation at the output of the RF amplifier. Use the equations provided to compare with your expected value. Provide DC analysis using the equations provided.
4. Select Simulate/Analyses/DC Operating Point. Specify the output node at the collector of the BJT and the node number for the voltage at the base of the transistor.
5. Click Simulate and compare the DC analysis with your calculations.
6. Select Simulate/Analyses/AC Analysis. Select the Output Variable tab and specify the output node number. Click Simulate to obtain a sketch of amplitude over the default frequency range. Calculate the bandwidth to verify your results.

Expected Outcome

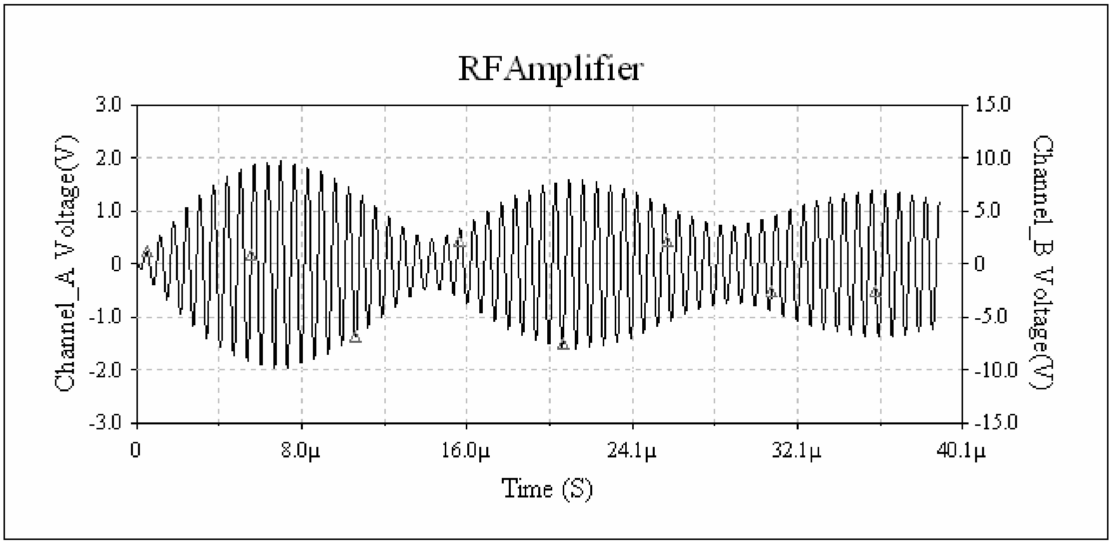


Figure 16-2 RF Amplifier Time versus Voltage Output

Data for Experiment 16

	Measured Value	**Calculated Value**
f_c		
V_c		
V_B		
BW		

Table 16-1

Additional Challenge

Re-design the RF amplifier to amplify a frequency of 1.3 MHz. Run the simulation including AC Operating Point analysis in order to verify your design parameters.

Experiment 17: S-Parameters in a Two Port Network

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of scattering parameters utilizing Multisim's Network Analyzer. When subatomic particles are accelerated, collisions with atoms result, and scattering parameters are generated. Though their name was derived through particle physics, a portion of the theory of scattering parameters is used in areas such as transmission line theory and microwave active devices. In transmission line theory, forward scattering corresponds to transmission and backward scattering corresponds to reflection. Interchanging the source and load produces reverse coefficients S_{22} and S_{12}. Transmission and reflection coefficients are denoted by S_{21} and S_{11}. Knowledge of the values of s-parameters in describing active devices means that circuits may be designed around the device with little or no understanding of the internal device operation. The reflection or scattering of electromagnetic waves is fundamental to the analysis and design of microwave circuitry. In this experiment, scattering parameters will be calculated using Multisim's Network Analyzer with a simple two port network.

Parts

Resistors: 25 Ω, 50 Ω, 100 Ω
Inductor: 30 nH

Test Equipment

- Oscilloscope

Formulae

$$S_{11} = \Gamma_i$$

Equation 17-1

$$S_{12} = V_i / V_o^+$$

Equation 17-2

$$S_{21} = V_o / V_i^+$$

Equation 17-3

$$S_{22} = \Gamma_o$$

Equation 17-4

Procedure

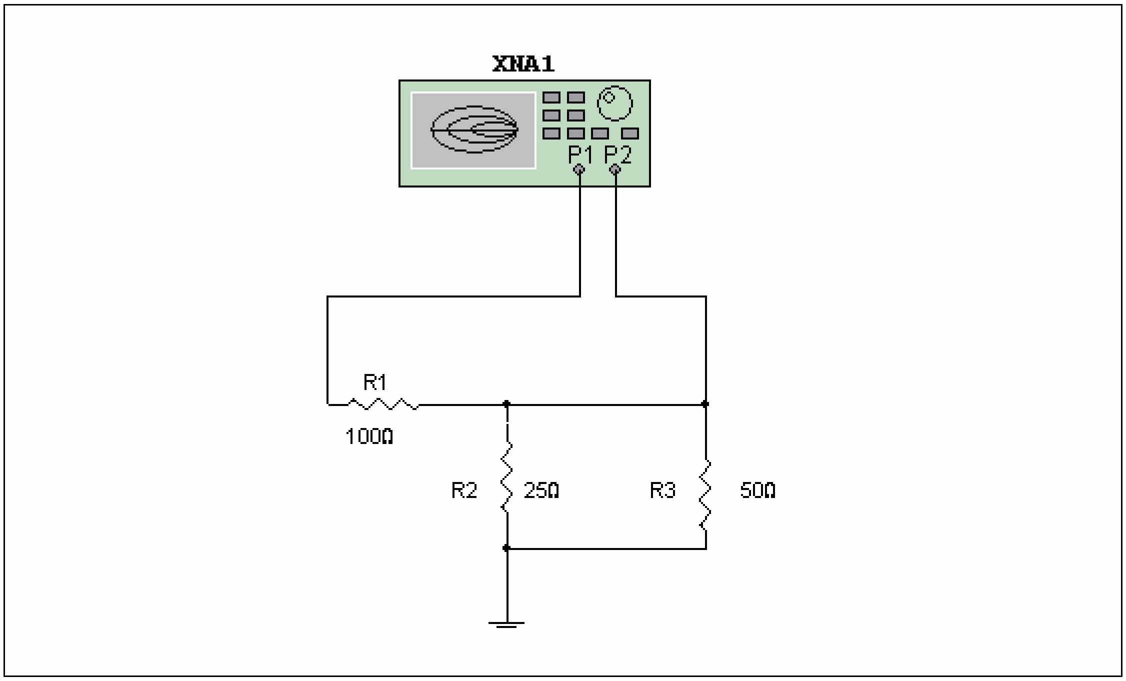


Figure 17-1 A Simple Two Port Network Example

1. Connect the circuit components illustrated in Figure 17-1.
2. Double-click the Network Analyzer to view its display.
3. Start the simulation and ensure that the Mode box in the Network Analyzer is set to Measurement. Select the Parameters drop down menu and select S-parameters. Select Polar graph. Record the values of S_{11}, S_{12}, S_{21} and S_{22} in Table 17-1. Analyze the two port network and compare your calculations with your measured data.

Data for Experiment 17

	Measured Value	**Expected Value**
S_{11}		
S_{12}		
S_{21}		
S_{22}		

Table 17-1

Additional Challenge

Replace the series 100 Ω resistor with a 30 nH inductor. Run the simulation. Press the right arrow on the scroll bar in the Marker section of the Network Analyzer until you read a frequency of 158.49 MHz. This value represents the an inductive impedance of +j30 Ω. Note the s-parameters as well as the forward power gain and verify your answers with calculations using an impedance of +j30 Ω in place of the 100 Ω resistor.

Experiment 18: Forward Gain in a Two Port Network

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of forward power gain in a two port network. In RF applications, current and voltage gains are often required in order to provide sufficient power to designated loads. In Multisim, both source and load impedances have default values of 50 Ω. The Network Analyzer calculates general power gain (PG), available power gain (APG) and transducer power gain (TPG) at a specified frequency. Scattering parameters are used to calculate these gains. In this experiment, we will be using the Network Analyzer to calculate the transducer power gain (TPG) which is the ratio of the power delivered to the load to the power available at the source. This represents the forward power gain of the two port network and is represented in dB as 10 log|TPG|. |TPG| is calculated from the scattering parameter $|S_{21}|$.

Parts

Resistors: 25 Ω, 50 Ω, 100 Ω
Inductor: 30 nH, 60 nH

Test Equipment

- Network Analyzer

Formulae

$$\textit{Forward Power Gain} = |S_{21}|^2$$

Equation 18-1

$$S_{21} = V_o^- / V_i^+$$

Equation 18-2

Procedure

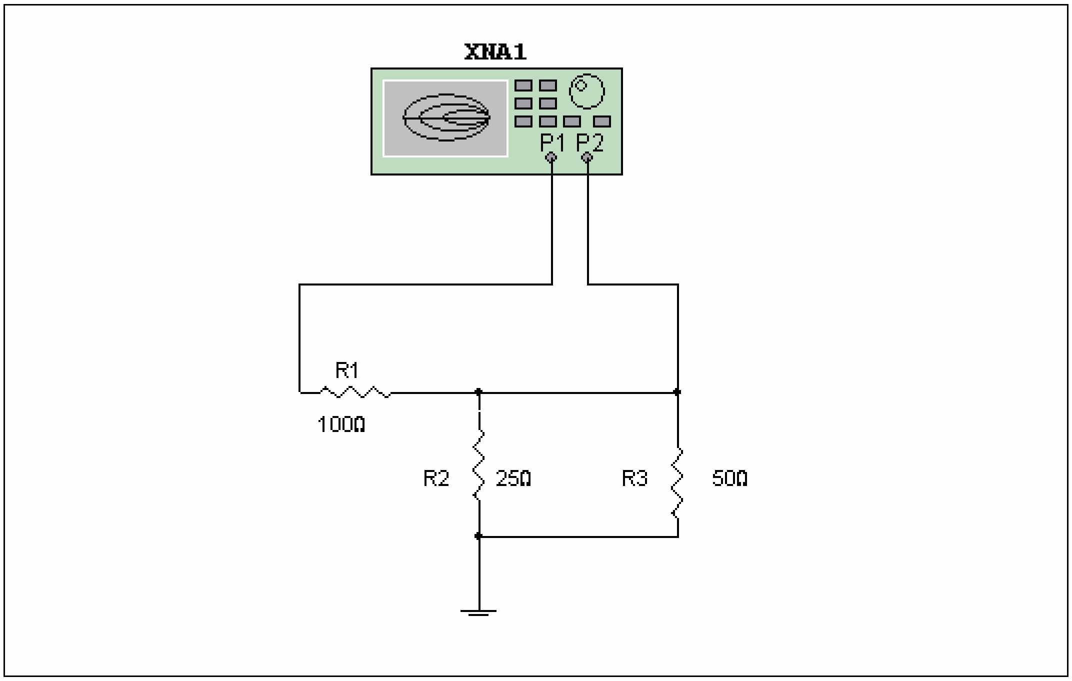


Figure 18-1 A Simple Two Port Network Example

1. Connect the circuit components illustrated in Figure 18-1.
2. Double-click the Network Analyzer to view its display.
3. Start the simulation and ensure that the Mode tab on the Network Analyzer is set to RF Characterizer.
4. Record the value of Transducer Power Gain (TPG).
5. Analyze the two port network and compare your calculations with your measured data.
6. Replace the series 100 Ω resistor with a 30 nF inductor and restart the simulation. Press the right arrow on the scroll bar in the Marker section of the Network Analyzer until you read a frequency of 158.4893 MHz. This value represents an impedance of j30Ω.
7. Calculate the new value of TPG and compare with measured data.

Data for Experiment 18

	Measured Value	Expected Value
Forward Power Gain R1 = 100		
Forward Power Gain L1 = 30 nH		

Table 18-1

Additional Challenge

Replace the series 100 Ω resistor with a 60 nH inductor. Run the simulation. Press the right arrow on the scroll bar in the Marker section of the Network Analyzer until you read a frequency of 251.19 MHz. This value represents an inductive impedance of +j50 Ω. Note the s-parameters and verify your answers with calculations using an impedance of +j50 Ω in place of the 100 Ω resistor.

Experiment 19: Stability Circles

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of stability circles. In order for a circuit to be stable, any reflections caused by the amplifier design, the frequency of oscillation or the device itself should be less than the signal found at the output. Stability circles use various frequencies in order to determine the stability of the circuit being analyzed.

The stability circle featured in Multisim utilizes a Smith Chart with both an input and an output stability circle. Any unstable regions are represented by hashing on the Smith Chart. The term “unconditionally stable” means that any source or load impedance may be selected without stability concerns. In this case, no hashing will be observed. The term “potentially unstable” means that any input impedance selected for the design should fall outside of the hatched area of the input stability circle while any output impedance selected for the design should fall outside of the hatched area of the output stability circle.

The delta and K values shown in the Match Net designer window will also indicate stability conditions. If K > 1 and delta < 1, the circuit is “unconditionally stable”. If K < 1 or delta > 1, oscillations may occur with inadequate impedance matching and as such, the amplifier will be “potentially unstable”.

Parts

Resistors: 10 Ω, 1 kΩ, 5 kΩ, 25 kΩ
Inductor: to be calculated
Capacitors: 0.01 F, to be calculated
RF NPN Transistor MMBR901L

Test Equipment

- Network Analyzer

Formulae

For Stability

$$delta < 1 \text{ and } K > 1$$

Equation 19-1

Resonant Frequency

$$fc = \frac{1}{2\pi\sqrt{LC}}$$

Equation 19-2

Procedure

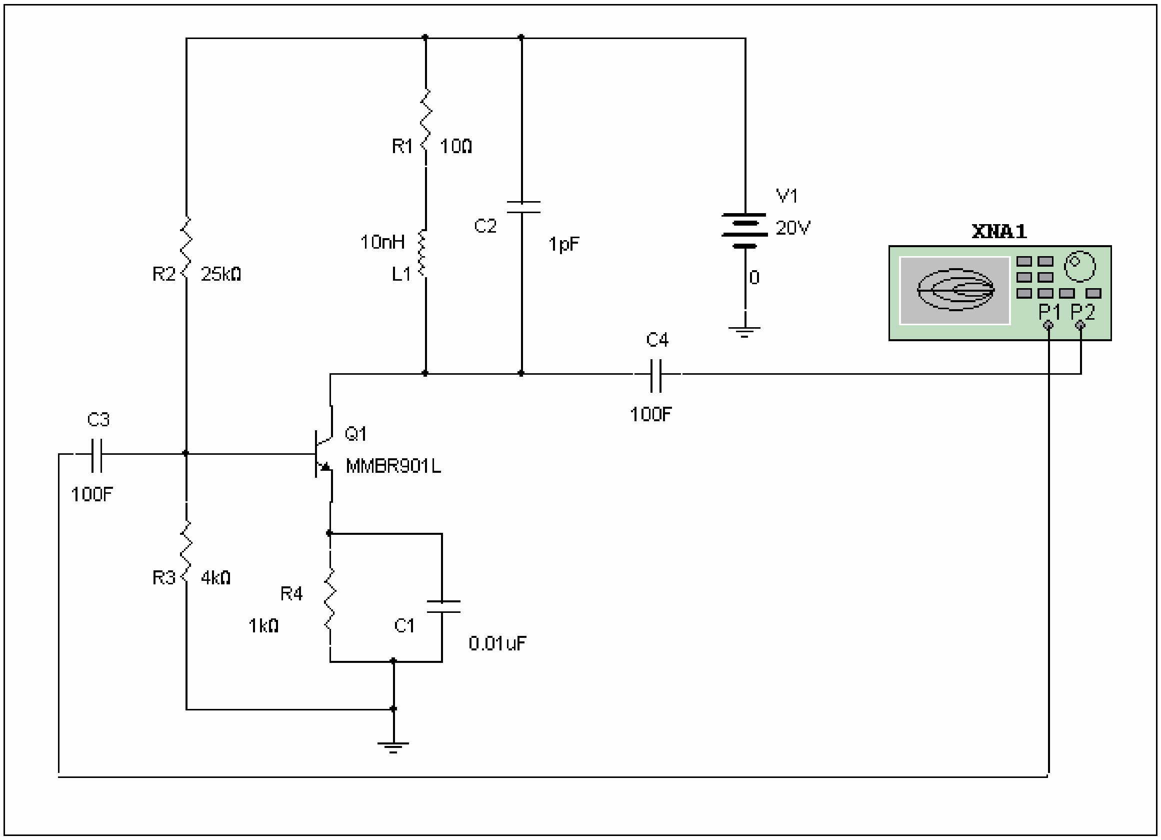


Figure 19-1 RF Amplifier Example

1. The circuit shown in Figure 19-1 is similar to the circuit design that we examined in our RF Amplifier experiment. Re-design the circuit of Figure 19-1 for a resonant frequency of 1.59 GHz. Replace the ideal NPN transistor with an RF transistor. Select a MMBR901L transistor as indicated. Insert 100F capacitors as shown for simulation purposes. Select the remaining components including your design values of L_1 and C_2 from the component toolbar.
2. Double-click the Network Analyzer to view its display.
3. Start the simulation and select *Stability Factor* under *Parameter*. Change the frequency using the scroll bar to 2.29 GHz. Note the value of delta and K at this frequency.
4. Select *Match Net Designer* under *Mode*. Change the frequency to 2.29 GHz. Observe the hatched area beginning to appear on the Smith Chart. What does this indicate?
5. Further increase the frequency to 3.981 GHz. Observe the hatching characteristics. Note the values of delta and K. Decrease the frequency to the resonant design frequency of 1.59 GHz. Note the values of delta and K. Describe any hatching characteristics.

Data for Experiment 19

Design value for C_2 @ f = 1.59 GHz ______________________________

Design value for L_1 @ f= 1.59 GHz ______________________________

	delta	**K**
f = 2.29 GHz		
f = 3.36 GHz		
f = 1.59 GHz		

Table 19-1

What does the hatched area indicate? ______________________________

Describe the hatching characteristics at the resonant frequency of 1.59 GHZ.

____

____

Additional Challenge

Re-design the RF amplifier for a resonant frequency of 5 GHz. Implement your design by replacing the necessary components. Repeat step1. Comment on the amplifier's performance.

Experiment 20: Impedance Matching

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of impedance matching. Input and output impedance considerations are generally implemented with both power transfer and bandwidth requirements of the circuit design in mind. Impedance matching is important because an undesired mismatched input or output will result in wasted power. Power costs money. Perfect power transfer with no losses is not possible because of the internal resistance inherent in voltage and current sources. This internal resistance restricts the current that can be provided by the source and decreases the source output voltage that is delivered to the load. There is, however, an optimum impedance value where maximum power is achieved. Maximum power transfer limits the circuit design to a narrow bandwidth which may or may not provide optimum design performance depending on the application.

Multisim features an Impedance Matching option which automatically designs circuitry that will match both an amplifier's input to the source impedance and output to the output impedance. This impedance matching feature is useful when a design is "unconditionally stable".

Parts

Resistors: 10 Ω, 1 kΩ, 5 kΩ, 25 kΩ
Inductor: to be calculated
Capacitors: 100 F, to be calculated
RF NPN Transistor MMBR901L

Test Equipment

- Network Analyzer

Formulae

For Stability

$$delta < 1 \text{ and } K > 1$$

Equation 20-1

Resonant Frequency

$$fc = \frac{1}{2\pi\sqrt{LC}}$$

Equation 20-2

Procedure

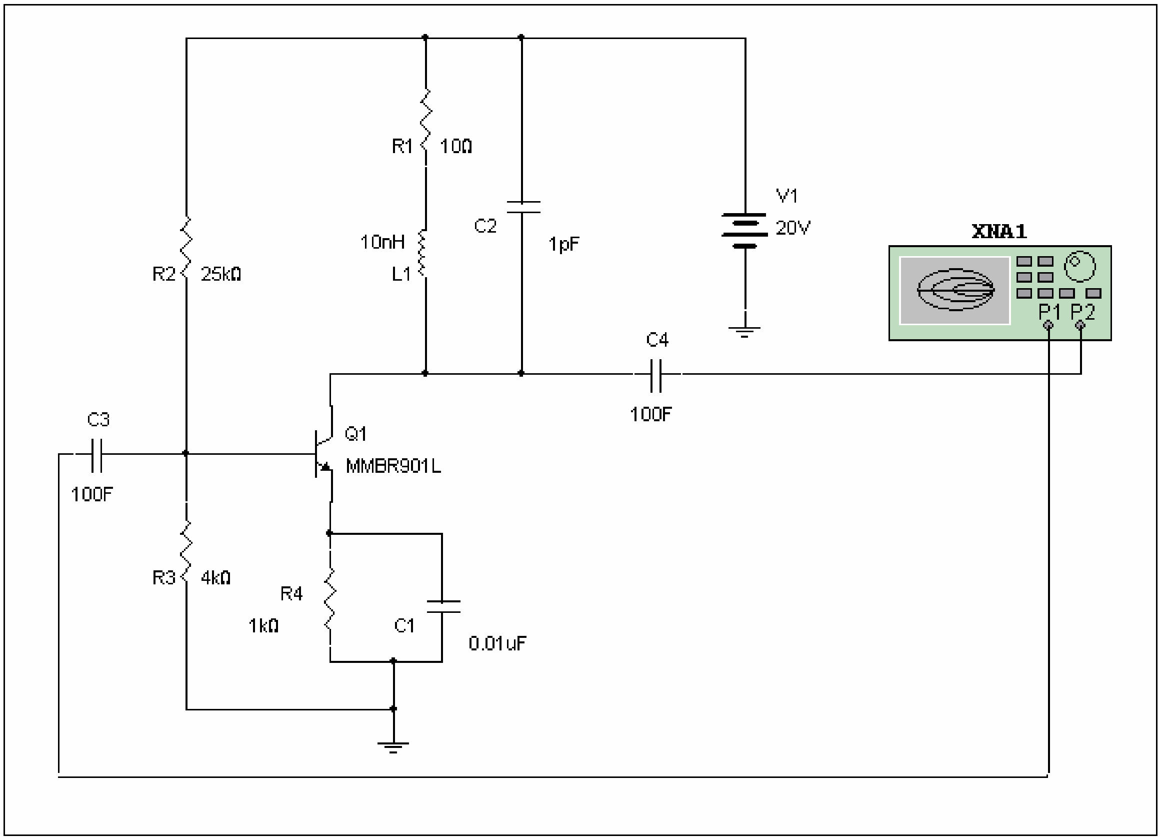


Figure 20-1 RF Amplifier Example

1. The circuit shown in Figure 20-1 is similar to the circuit design that we examined in our RF Amplifier experiment. Re-design the circuit of Figure 20-1 for a resonant frequency of 5.3 Ghz. Replace the ideal NPN transistor with an RF transistor. Select a MMBR901L transistor. Insert 100F capacitors as shown for simulation purposes. Select the remaining components including your design values of L_1 and C_2 from the component toolbar.
2. Double-click the Network Analyzer to view its display.
3. Start the simulation and select *Stability Factor* under *Parameter*. Change the frequency using the scroll bar to 5.2481 GHz. Note the value of delta and K at this frequency. Verify that the amplifier is "unconditionally stable" at this frequency.
4. Select *Match Net Designer* under *Mode*. Change the frequency to 5.2481 GHz as a desired operating point and select the *Impedance Matching* tab. Enable the *Auto Match* feature. The design components and values will be shown. Draw the structure in the Data section of this experiment. A total of eight design structures can be accessed by clicking to the left and right of the structure. All eight of these designs, however, will not provide optimum matching.
5. Re-design the circuit for a resonant frequency of 7 GHz and replace the affected components.

6. Restart the simulation and select *Stability Factor* under *Parameter*. Change the frequency using the scroll bar to 6.9183 GHz. Continue with steps 2 to 4, using a frequency of 6.9183 GHz.

Data for Experiment 20

Design value for C_2 *@* $f = 5.3$ *GHz* ________________________________

Design value for L_1*@* $f = 5.3$ *GHz* ________________________________

Design value for C_2 *@* $f = 7$ *GHz* ________________________________

Design value for L_1*@* $f = 7$ *GHz* ________________________________

Additional Challenge

Re-design the RF amplifier for a resonant frequency of 1.59 GHz. Implement your design by replacing the necessary components. Repeat steps 2 to 4 using an operating frequency as close to 1.59 GHz as possible.

Experiment 21: Waveguides

Purpose and Discussion

The purpose of this simulation is to demonstrate the characteristics of waveguides. A waveguide is any medium which guides waves. Waveguides are used for frequencies above several giga-hertz where coaxial cable begins to exhibit skin effects and radiation attenuation and losses. The electromagnetic wave that is injected at the input by a signal launcher is bounded by the waveguide and is reflected off of the conducting walls. A wide spacing is provided which allows the transfer of hundreds of thousands of watts of power without breaking down any non-existing dielectric barriers between the conductors as is the case with coaxial cable. This results in very little loss. The signal is then received by a signal absorber. Attention to waveguide dimensions is crucial. The frequency bands of waveguides have assigned designated letters to bands of specified dimensions, frequencies and cutoff wavelengths. The cutoff frequency of a waveguide is the lowest frequency that will propagate through the conducting tube in an actual waveguide. This frequency is normally discussed in terms of wavelength as it is the length of the wave that limits its ability to propagate. Actual waveguides behave much like high pass filters providing high attenuation at frequencies below cutoff.

The characteristic impedance is the impedance that would be measured at the input of an infinite length of waveguide and is given by:

$$Z_{OL} = \frac{120\pi}{\left[1-\left(\frac{\lambda_{fs}}{\lambda_{CL}}\right)^2\right]^{1/2}} \qquad \text{where} \qquad \lambda_{fs} = \frac{c}{f} = \textit{freespace wavelength}$$

and λ_{CL} = *2(broadwall dimension of the waveguide)*

Parts

Resistors: 55 Ω
Sample Waveguide
AC Voltage Source

Test Equipment

- Oscilloscope

Formulae

Characteristic Impedance

$$Z_{OL} = \frac{120\pi}{[1-(\frac{\lambda fs}{\lambda CL})^2]^{1/2}}$$

Equation 21-1

$$\lambda_{fs} = \frac{c}{f} = \text{ freespace wavelength}$$

Equation 21-2

λ_{CL} = *2(broadwall dimension of the waveguide) = cutoff wavelength*

Equation 21-3

$f_{CL} = c/\lambda_{CL}$ = *cutoff frequency where*
$c = 2.9974 \times 10^{10}$ *cm/s*

Equation 21-4

C-Band Waveguide
Frequency range = 4.9 - 7.05 GHz
Broadwall Dimension of waveguide in cm = 4.039

P-Band Waveguide
Frequency range = 18 - 26.5 GHz
Broadwall Dimension of waveguide in cm = 1.580

Procedure

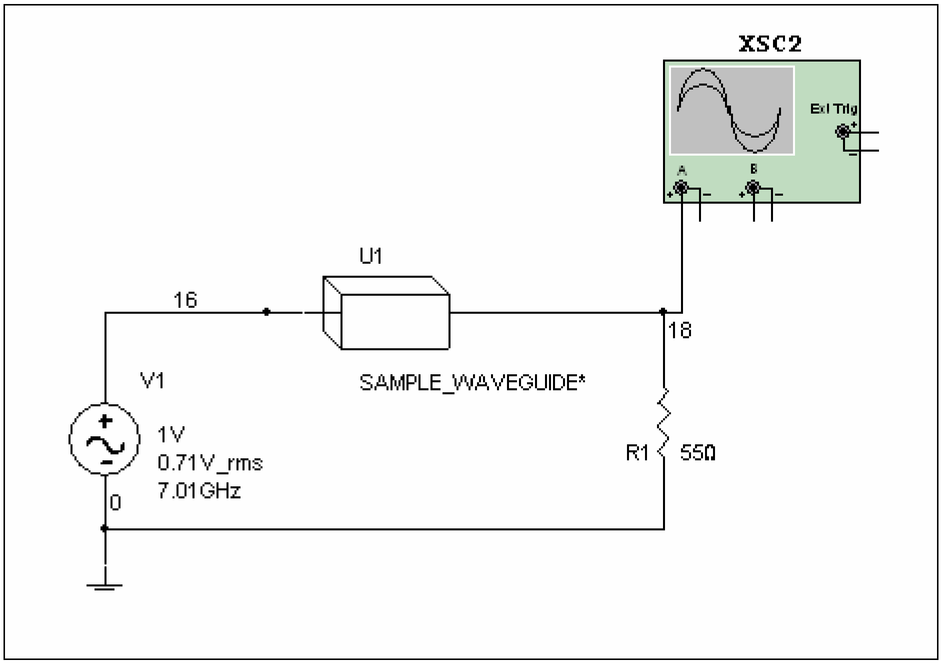


Figure 21-1 Waveguide Example

1. Connect the circuit illustrated in Figure 21 -1.
2. Calculate the lowest acceptable propagation frequency for a C-Band waveguide. Note your results in the Data section of this experiment. Select the frequency of the AC source = 4.9 GHz.
3. Double-click the sample waveguide. Choose EDIT MODEL. Change the SPICE parameters so that LEN = 4.039 e-002.
4. Double-click the Oscilloscope to view its display. Set timebase to 0.2 ns/Div and Channel A = 1 V/Div. Run the simulation and observe the output waveform. Measure the frequency to verify that the expected signal was propagated by the simulated waveguide. Note your results.
5. Select a frequency of 7.05 GHz for the AC voltage source. Run the simulation again and note your results. Verify that the expected signal was propagated by the simulated waveguide. Run the simulation again and note your results.
6. Calculate the characteristic impedance for the C-Band waveguide.

Expected Outcome

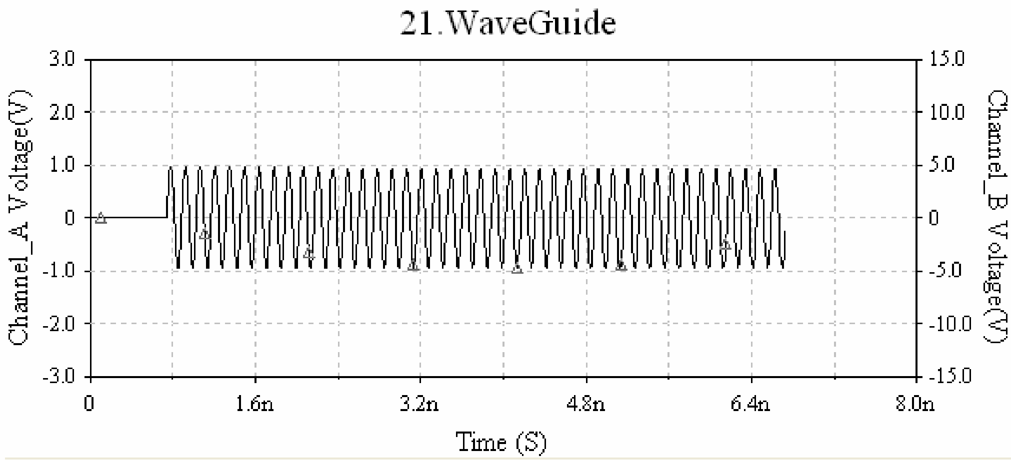


Figure 21-3 Time versus Amplitude of Waveguide Output

Data for Experiment 21

f_{CL} = ___________________

	Expected (lower)	**Measured (lower)**	**Expected (upper)**	**Measured (upper)**
Propogation Frequency C-Band				

Table 21-1

Z_{OL} = ___________________

Additional Challenge

Repeat all steps for a P-Band waveguide. Verify your data.

Solutions to Experiments and Additional Challenges

Experiment 1

Using $P = V^2/R_L$
Power calculated (1 kHz) = $(9.997\ V)^2/1 \times 10^3\ \Omega$ = 99.9mW
Power calculated (10 kHz) = $(9.875\ V)^2/1 \times 10^3\ \Omega$ = 97.5mW

Frequency (measured)	LIN Voltage (V) (measured)	Power (mW) (calculated)	Frequency (Hz) (expected)	LIN Voltage (V) (expected)
1 kHz	9.997	99.9	1 kHz	10
10 kHz	9.875	97.5	10 kHz	10

Table 1-1

Span = End - Start = 15 kHz - 5 kHz = 10 kHz

Frequency (Hz)	Start (Hz)	End (Hz)	Span (Hz) (measured)	Span (Hz) (expected)
10 kHz	5 kHz	15 kHz	10 kHz	10 kHz

Table 1-2

Additional Challenge (Experiment 1)

Span / # divisions = 10 / 4 divisions = 2.5 kHz/div and
since the center frequency = 8 kHz, then Start = 8 - 5 = 3 and End = 8 + 5 = 13.

Experiment 2

Squarewave

Using equation 2-2:
Expected rms voltage = 4 (10) / 4.44 = 9V then,

Expected mW power $P = V^2/R_L = 9^2 / 1 \times 10^3 = 81$ *mW*

	Measured Frequency (Hz)	Measured rms Voltage (V)	Measured Power (mW)	Expected rms Voltage (V)	Expected Power (mW)
f_0	10 kHz	8.045	64.7	9	81
f_3	30 kHz	3.6	12.96	3	9
f_5	50 kHz	2.187	4.783	1.8	3.24

Table 2-1 Square Wave Results

P_T *(mW) = 64.7 + 12.96 + 4.783*

= Total Power in mW (measured) <u>82.44 mW</u>

Total Power in mW from wattmeter <u>100 mW</u>

Sawtooth wave

$$rms = \frac{amplitude}{\sqrt{3}}$$

Using equation 2-4:
Expected rms Voltage = 2 (10)/ 5.44 = 3.68 V, then,

Expected mW Power $P = V^2/R_L = 3.68^2 / 1 \times 10^3 = 13.54$ *mW*

	Measured Frequency (Hz)	Measured rms Voltage (V)	Measured Power (mW)	Expected rms Voltage (V)	Expected Power (mW)
f_0	10 kHz	4.295	18.45	3.68	13.54
f_2	20 kHz	1.736	3	1.84	3.39
f_3	30 kHz	0.77	0.59	1.23	1.5

Table 2-2 Sawtooth Wave Results

P_T *(mW) = 18.45 +3 + 0.59*

Total Power in mW (measured) <u>22.04 mW</u>

Total Power in mW from wattmeter <u>33 mW</u>

Additional Challenge (Experiment 2)

	Measured Frequency (Hz)	Measured rms Voltage (V)	Measured Power (mW)
f_0	1 kHz	2.34	5.47
f_3	3 kHz	0.26	0.068
f_5	5 kHz	0.087	0.0075

$P_T\,(mW) = 5.47 + 0.068 + 0.0075 = 5.55\ mW$

Total Power in mW from wattmeter __8.33 mW__

Experiment 3

Using equation 3-1:

$$f_C = \frac{1}{2\pi\sqrt{(200e-6)\,(220e-12)}} = 758.7\ kHz$$

decibels @ f = 8 kHz = 20 log 490 x 10^{-3} *= - 6.2 dB*

Frequency	Amplitude (mV)	Decibel Gain (dB)
7.6 kHz	900	-0.915
76 kHz	100	-20
f_c = ________	0.260	-71.7
760 kHz	0.300	-70.46
7.6 MHz	100	-20
76 MHz	800	-1.94

Table 3-1

The slope exceeds -40 dB/decade around the center frequency for this 2nd order filter.

Additional Challenge (Experiment 3)

Re-arranging equation 3-1:
$C = 1/\,4B f_c^{\,2} L = 124\ pF$

Frequency	Amplitude (mV)	Decibel Gain (dB)
10.1 kHz	490	-6.19
101 kHz	700	-3.1
1010 kHz	0.20	-73.98
10.1 MHz	75	-22.4
101 MHz	600	-4.44

The slope exceeds the expected -40 dB/decade around f_c.

Experiment 4

Using equation 4-1 and 4-2:

$$BW = \frac{2.2}{2\pi(33e-6)} = 10.6\ kHz \qquad Q = \frac{1}{2.2}\sqrt{\frac{33e-6}{2.4e-9}} = 53.3$$

Using equation 4-4:

$$f_C = \frac{1}{2\pi\sqrt{(2.4e-9)\ (33e-6)}} = 565\ kHz$$

	Measured Value	**Calculated Value**
BW	10 kHz	10.6 kHz
f_c	565.5 kHz	565.5 kHz
Q	-----	53.3

Table 4-1

decibels @ $f_c = 20 \log 600 \times 10^{-6} = -64$

Frequency	**Amplitude (V)**	**Decibel Gain (dB)**
f_c = 565.5 kHz	600×10^{-6}	-64
600 Hz	1×10^{-9}	-180
6 kHz	1×10^{-6}	-120
60 kHz	2.5×10^{-6}	-112
600 kHz	140×10^{-6}	-77
6 MHz	2.5×10^{-6}	-112
60 MHz	1×10^{-6}	-120
600 MHz	1×10^{-9}	-180

Table 4-2

The slope is approaching -40 dB/decade around the center frequency for this 2nd order filter.

Experiment 5

For a higher order circuit configuration, the Stop Frequency menu selection must be decreased.
For a lower order circuit configuration, the Stop Frequency menu selection must be increased.

Experiment 6

Using equation 6-1:

$$f_c = \frac{1}{2\pi\ (10e-9)\ \ (10e3)\ \sqrt{6}} \quad = 649.7\ Hz$$

	Measured Value	Calculated Value
Frequency (Hz)	650	649.7
Rf/R at point where oscillations begin	29	29

Table 6-1

Additional Challenge (Experiment 6)

Re-arranging equation 6-1:

$$C = \frac{1}{2\pi\ (900)(10e3)\sqrt{6}} = 7.2nF$$

Experiment 7

Using equation 7-1:

$$f_c = \frac{1}{2\pi\sqrt{(0.5+0.5)e-3(1e-6)}} = 5032.9\,Hz \quad calculated$$

$f_c = 5$ *kHz measured*

Step 5

$f_c = 5033$ *Hz*

Step 6

$I_B = V_B\text{-}V_{BE}/R_{BB} + (\beta+1)R_E = 5.3/5e3+(101)500 = 95\ \mu A$

$I_E = 9.5\ mA$

$re = 25\ mv/9.5\ mA = 2.6$

$Av = R_{C/}re = 78.5/2.6 = 30.2 > 1/\beta = 1$ *therefore oscillations will be sustained*

Step 7

$$fc = \frac{1}{2\pi\sqrt{(L_1+L_2)C}} \quad @\ L_1 = 0.5\ mH,\ C = 1\ \mu F \ \text{ and } f = 3.5\ kHz$$

$f_c^2 = 1/4\Pi^2(L_{1+}L_2)C \qquad L_2 = (1 - 4\Pi^2 f_c^2 C L_1)\,/\,4\Pi^2 C f_c^{\ 2} = 1.57\ mH$

Additional Challenge (Experiment 7)

$L = 822\ \mu H$

Experiment 8

Using equation 8-1:

$$fc = \frac{1}{2\pi\sqrt{\frac{(60e-6)(180e-12)(22e-12)}{180e-12+22e-12}}} = 4.64\ MHz$$

Re-arranging equation 8-1:

$L_1 = (C_1 + C_2)/4\Pi^2 C_1 C_2 f_c^2 = 20.2\ \mu H$

$Av = -g_m\, r_d = (-1.6mS)(12e3) = -19.2$ Since $C_1/C_2 = 180/22 < |Av| = 19.2$ oscillations will be sustained.

Additional Challenge (Experiment 8)

Choose $C_1 = 200\ pF$ since $fc = \frac{1}{2\pi\sqrt{\frac{L_1 C_1 C_2}{C_1 + C_2}}}$

then $C_2 = C_1 / 4\pi^2 f_c^2 L_1 C_1 - 1 = 61\ pF$

Experiment 9

Re-arranging equation 9-1:

$C_3 = 1/4\pi^2 L f_r^2 = 1/4\pi^2(68e\text{-}6)(2e6)^2 = 93\ pF$

$$fc = \frac{1}{2\pi\sqrt{(68e-6)(120e-12)}} = 1.76\ MHz$$

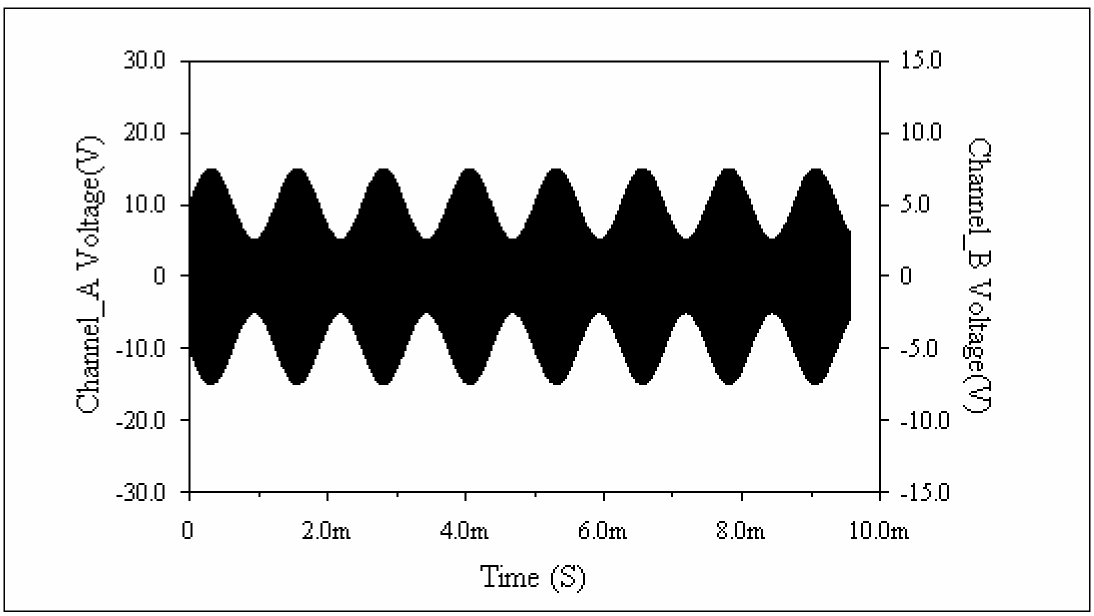
	Measured Value	Calculated Value
f_c (step 2)	1.6 MHz	1.76 MHz
f_c (step 3)	2.06 MHz	2 MHz

Table 9-1

Additional Challenge (Experiment 9)

Upper frequency limit measured = 3.5 MHz

Experiment 10



$f_{Lower} = 100 \times 10^3 - 800 = 99200\ Hz$

$f_{Upper} = 100 \times 10^3 + 800 = 100800\ Hz$

Additional Challenge (Experiment 10)

Requires observation only.

Experiment 11

Using equations 11-1 and 11-3:

$m = (32\text{-}8) / (32+8) = 0.6 m_{measured} = 2 / 10^{[(20 - 10)/20]} = 0.63$

$mod.\ Signal\ Amplitude = m(carrier\ amplitude) = 0.6(10) = 6V$

$m = (26\text{-}13) / (26+13) = 0.33 \qquad m_{measured} = 2 / 10^{[(18.8 - 3.2)/20]} = 0.33$

$mod.\ Signal\ Amplitude = m(carrier\ amplitude) = 0.33(10) = 3.3V$

	V_{max}	V_{min}	$m_{measured}$	carrier dB	sideband dB	$m_{measured}$	$m_{expected}$	%m	mod. Sig. Amplitude
m = 0.6	32 V	8 V	0.6	20	10	0.63	0.6	60	6 V
m = 0.33	26 V	13 V	0.33	18.8	3.2	0.33	0.33	33	3.3 V

Table 11-1

Using equations 11-1 and 11-3:

$@m = 0.5\ P_{LSB} = 3.356^2/100 = 0.1126W$

$P_{USB} = 2.314^2/100 = 53\ mW \quad P_C = 9.881^2/100 = 976\ mW \quad P_T = 1.142$

$\mu = (112.6 + 53)\ mW/1.142W = 0.145$

Using equation 11-4:

	Measured Efficiency	Calculated Efficiency
m = 0.5	0.172	0.11
m = 0.7	0.213	0.197
m = 0.8	0.261	0.24
m = 1	0.341	0.33

Table 11-2

Additional Challenge (Experiment 11)

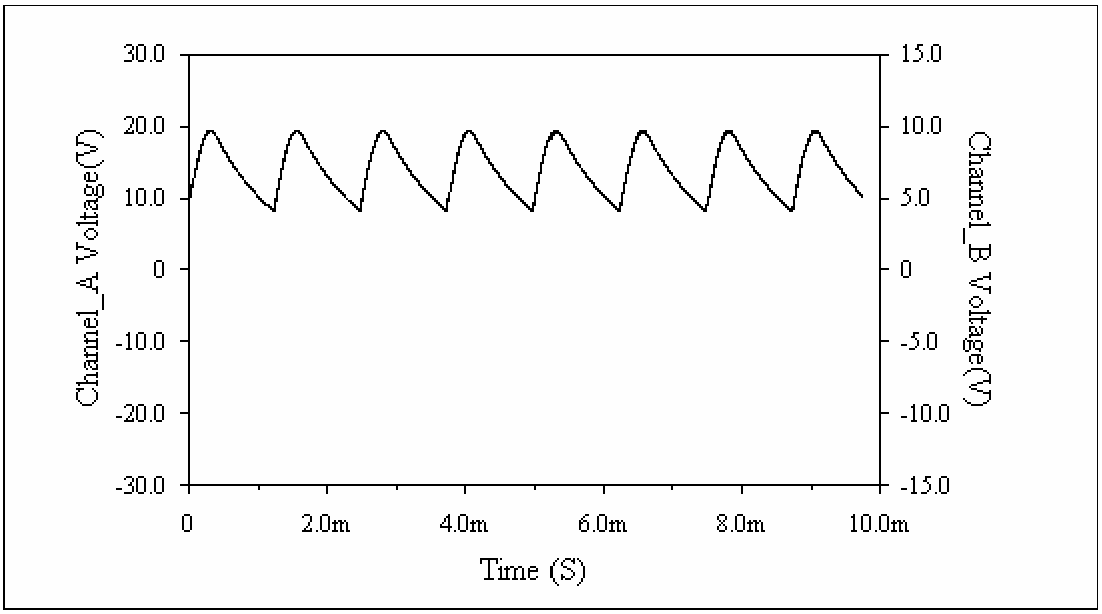
	V_{max}	V_{min}	$m_{measured}$	carrier dB	sideband dB	$m_{measured}$	$m_{expected}$	%m	mod. Sig. Amplitude
m = 0.8	34 V	0.79 V	0.8	19	9	0.8	0.8	80	8 V

$\mu = 0.1525$

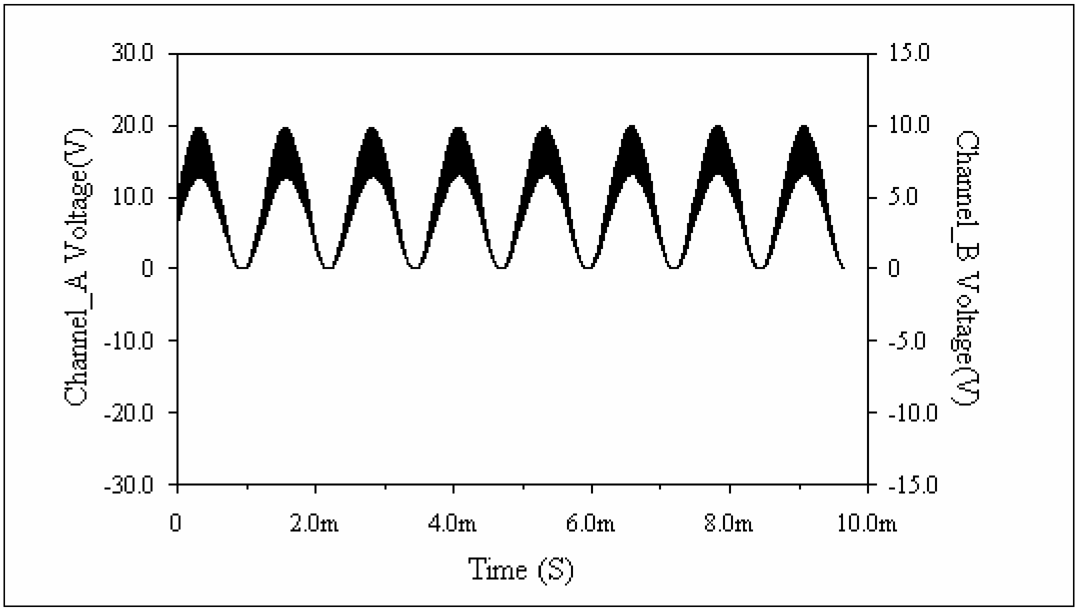
Experiment 12

f_m at output of detector ______*800 Hz*______ *$f_{m\ expected}$*______*800 Hz*______

Waveform of an RC time constant which is too large:



Waveform of an RC time constant which is too small:



Using equation 12-4:

$$RC = \frac{1}{2\pi\,(0.6)500} = 530.5 \times 10^{-6} \quad @R = 100\ k\Omega \quad C = 530.5 \times 10^{-6}/100 \times 10^{3} = 5.3\ nF$$

Additional Challenge (Experiment 12)

Requires observation only.

Experiment 13

$f_{LSB} = 1.1\ MHz - 10\ kHz = 1090000\ Hz \quad f_{USB} = 1.1\ MHz + 10\ kHz = 1110000\ Hz$

	Measured Value	**Calculated Value**
Frequency of lower sideband	1.09 MHz	1.09 MHz
Frequency of upper sideband	1.11 MHz	1.11 MHz

Table 13-1

Using equation 13-1:

$P_{LSB} = 38.552^2/1x10^3 = 1.486\ W$ $\quad P_{USB} = 40.756^2/1x10^3 = 1.666\ W$

$P_{LSB} = 4.74^2/1x10^3 = 22.47\ mW$ $\quad P_{USB} = 5.377^2/1x10^3 = 28.9\ mW$

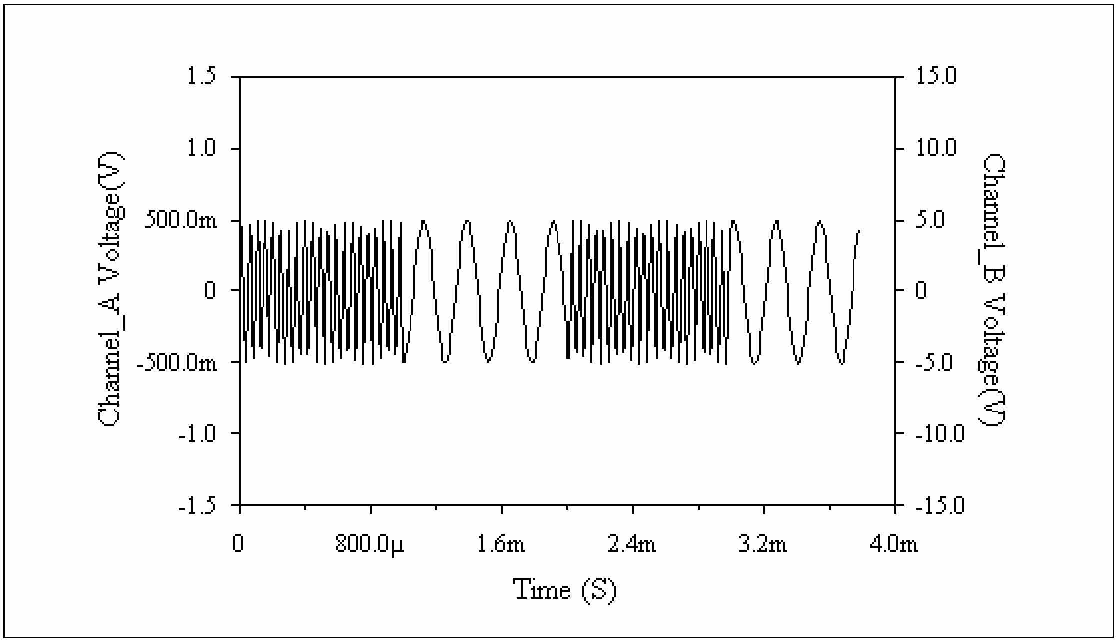
	LSB (Vrms)	**USB (Vrms)**	**LSB (W)**	**USB (W)**
DSB	38.552	40.746	1.490	1.660
AM	4.740	5.377	22.470 mW	28.9 mW

Table 13-2

Additional Challenge (Experiment 13)

$f_{LSB} = 1\ MHz - 12\ kHz = 988000\ Hz$ $f_{USB} = 1\ MHz + 12\ kHz = 1012000\ Hz$

Experiment 14



$f_m = 10\ kHz$

Experiment 15

Using equations 15-1 and 15-2:

Δf @ $(m = 3_{calculated}, f_m = 10\ kHz) = \Delta f = m/f_m = 3/\ 10e3 = 300e\text{-}6$

Bandwidth W @ $(m = 3, f_m = 10\ kHz) = 2(6)(10\ kHz) = 120\ kHz$

At m = 1.5, the power is distributed over less spectral lines.

Additional Challenge (Experiment 15)

At a modulation index of 2.4, the carrier disappears. This is shown by the Bessel coefficient (J_0) which is equal to zero when the modulation index is equal to the value of 2.4.

Experiment 16

$Z|| = L/RC = 156.25\ k\Omega$ $\qquad Q = Z||/X_L = 124.6$

$X_L = 2\pi(1.6e6)(125e\text{-}6) = 1.26\ k\Omega$ $\qquad BW = 1.6\ MHz/124.6 = 12.84\ kHz$

	Measured Value	Calculated Value
f_c	1.4 MHz	1.6 MHz
V_c	12 V	11.98 V
V_B	1.9 V	2 V
BW	-----	12.84 kHz

Table 16-1

Additional Challenge (Experiment 16)

$$fc = \frac{1}{2\pi\sqrt{LC}} \quad \text{Choose } L = 100\ \mu H$$

$$C = 1/\ 4\pi^2 f_c^{\,2} L = 1/\ 4\Pi^2(1.3e6)^2(100e\text{-}6) = 150\ pF$$

Experiment 17

$@R = 100$

$Z_1 = 100 + 16.67 = 116.67$

$S_{11} = \Gamma_i = (Z_1 - 50)/\ (Z_1 + 50) = 116.67 - 50/116.67 + 50 = 66.67/166.67 = 0.4 + j0$

$S_{22}\ Z_2 = (50 + 100)||25 = 21.43\ \Omega$

$S_{22} = \Gamma_o = (21.43 - 50)\ /\ (21.43 + 50) = -\ 0.4 = -0.4 + j0$

$S_{21} = V_o/V_i^+$ $\qquad V_o = (25||50)\ 2Vi^+/(25||50) + 150 \quad V_o/V_i^+ = 0.2 + j0$

Additional Challenge (Experiment 17)

$@L = 30\ nH,\ f = 158.4893\ MHz,\ \text{then } X_L = j30$

Terminate the output at the input 50 Ω and calculate Z_1 and Γ_i at the input

$Z_1 = j30 + 16.67 \quad S_{11} = \Gamma_i = (Z_1 - 50)/\ (Z_1 + 50) = (16.67 + j30) - 50/(16.67 + j30) + 50$

$= -33.33 + j30/66.67 + j30$

$= 44.8\pi 138E/\ 73.1\pi 24.23E = 0.613\pi 113.77E = -0.247 + j0.56$

Terminate the input with 50 Ω and calculate Z_2 and Γ_o at the output

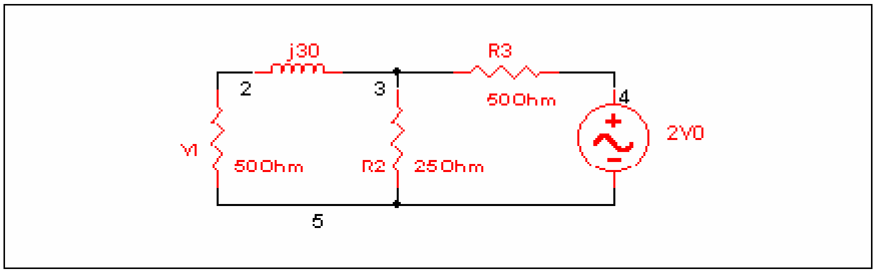
$S_{22}\ Z_2 = (40 + j30)||25 = 16.97 + j3.05$ then

$S_{22} = \Gamma_o = Z_2 - 50/Z_2 + 50 = -33.03 + j3.05/66.97 + j3.05 = -0.49 + j0.68$

Supply port1 with an open circuit voltage of 2Vi⁺ and a 50 Ω source impedance

$S_{21}V_o = 16.67(2V_i^+)/(50+j30) + 16.67 = 0.457V_i^+ \; \pi\text{-}24.22E$

$S_{21} = V_o/V_i^+ = 0.457\pi\text{-}24.22E \quad = 0.417 - j\,0.188$



(A)

Thevenize (A) at nodes 3 and 5

Then, $R_{TH} = 25||50 = 16.67\ \Omega$ $V_{TH} = 25(2V_o^+)/(25 + 50) = 0.67V_o^+$

Then, $S_{12} = Vi = 50(0.67)V_o^+ / 50 + j30 + 16.67$ and $Vi/V_o^+ = 33.5/66.67 + j30 = 0.46\pi\text{-}24.22E$

Experiment 18

Forward Power Gain = TPG = $|S_{21}|^2 = 0.457^2 = 0.21 = -10\log 0.21 = 6.8$ *dB loss*

measured value = - 6.8153 dB

Additional Challenge (Experiment 18)

$@L = 60$ nH, $f = 251.19$ MHz, then $X_L = j50$

Terminate the output with the input 50 Ω and calculate Z_1 and Γ_i at the input

$Z_1 = j50 + 16.67$

$S_{11} = \Gamma_i = (Z_1 - 50)/(Z_1 + 50) = (16.67 + j50) - 50/(16.67 + j50) + 50 = -33.33 + j50/66.67 + j50 = 60.1\,\underline{/123.7^o} / 83.336\,\underline{/36.9^o} = 0.7\,\underline{/2\ 86.8^o} = 0.039 + j0.72$

Terminate the input with 50 Ω and calculate Z_2 and Γ_o at the output

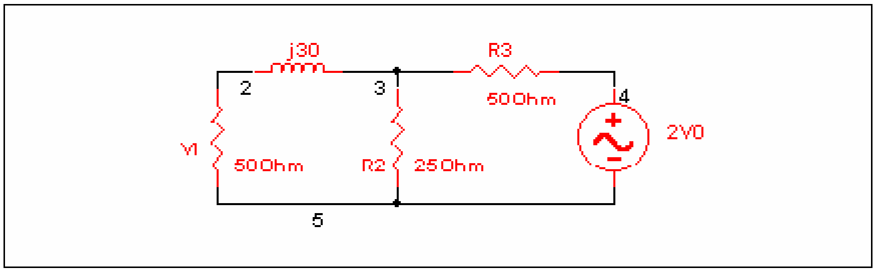
$S_{22}\ Z_2 = (50 + j50)||25 = 19.5 + j0.39$ then

$S_{22} = \Gamma_o = Z_2 - 50/Z_2 + 50 = -30.4 + j0.39/69.6 + j0.39 = -0.437 + j0.008$

Supply port1 with an open circuit voltage of 2Vi⁺ and a 50 Ω source impedance

$S_{21}\ V_o = 16.67(2V_i^+)/(50+j50) + 16.67 = 0.457V_i^+\ \underline{/36.9^o}$

$S_{21} = V_o/V_i^+ = 0.4\underline{/36.9^o} \quad = 0.32 - j\,0.24$



(A)

Thevenize (A) at nodes 3 and 5

Then, $R_{TH} = 25||50 = 16.67\ \Omega$ $V_{TH} = 25(2V_o^{\;+})/(25 + 50) = 0.67V_o^{\;+}$

Then, $S_{12} = Vi = 50(0.67)V_o^{\;+} / 50 + j50 + 16.67$ *and* $Vi/V_o^{\;+} = 33.5/83.336\ \underline{/36.8^o}$

$= 0.4\underline{/36.8^o} = 0.32 - j\ 0.24$

Forward Power Gain TPG = $|S_{21}|^2 = 0.4^2 = 0.16 = -10\log 0.16 = 7.96$ *dB loss*

Measured = - 7.94 dB

Experiment 19

Design value for $C_{2\ @\ f = 1.59\ GHz}$, *choose* $L_1 = 10$ *nH, then re-arranging equation 19-2,*

$C_2 = 1/4\pi^2 f_c^{\,2} L_1 = 1/4\pi^2(1.59\ Ghz)^2 10e\text{-}9 = 1\ pF$

	delta	K
f = 2.29 GHz	0.239	0.9104
f = 3.63 GHz	0.4908	0.4957
f = 1.59 GHz	0.1791	1.3738

Table 19-1

The hatched area indicates a potentially unstable frequency depending on the input/output impedance matching design.

At the resonant frequency of 1.59 GHz, there is no hatching.

Additional Challenge (Experiment 19)

Design value for $C_{2\ @\ f = 5\ GHz}$, *choose* $L_1 = 1$ *nH, then re-arranging equation 19-2,*

$C_2 = 1/4\pi^2 f_c^{\,2} L_1 = 1/4\pi^2(5\ GHz)^2 1e\text{-}9 = 1\ pF$

At an operating frequency of 5 GHz, the amplifier is close to being potentially unstable.

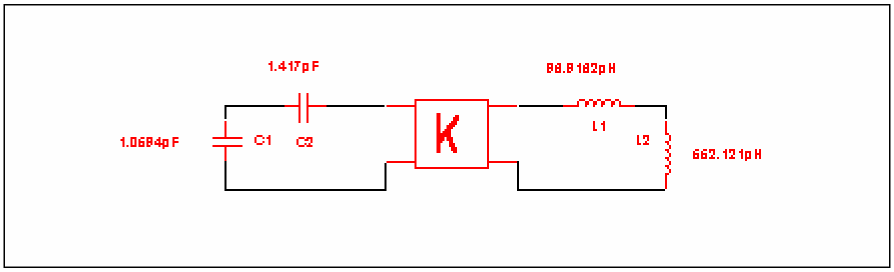
Experiment 20

Re-arranging formula 20-2:

at C = 1 pF and 5.3 GHz, then

$L = 1/4\pi^2 f_c^{\,2} C = 1/\ 4\pi^2(5.3e9)^2\ (1e\text{-}12) = 0.9\ nF$

then, at an operating frequency of 5.2481 GHz,

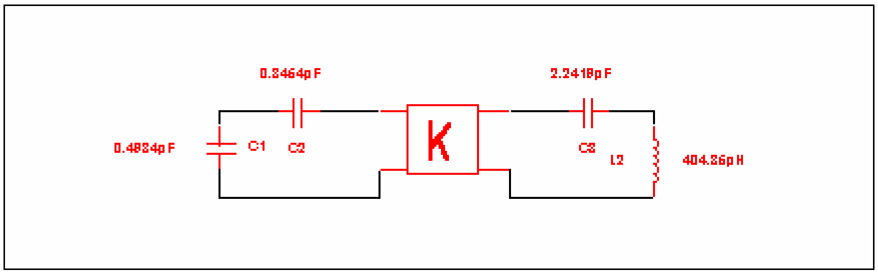


Re-arranging formula 20-2:

at C = 1 pF and 7 GHz, then

$L = 1/4\pi^2 f_c^{\,2} C = 1/\ 4\pi^2 (7e9)^2\ (1e\text{-}12) = 0.5\ nF$

Then, at an operating frequency of 6.9183 GHz,

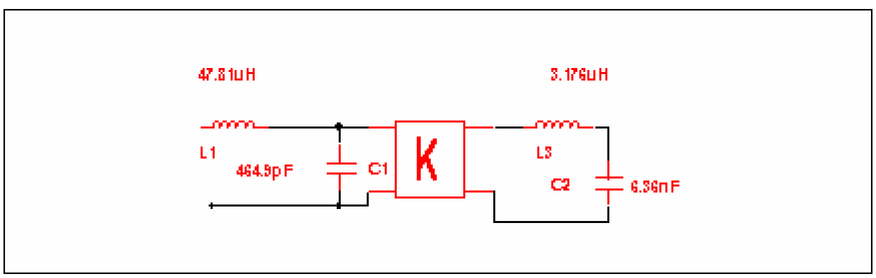


Additional Challenge (Experiment 20)

Re-arranging equation 20-2:

at C = 1 pF and 1.59 GHz, then

$L = 1/4\pi^2 f_c^{\,2} C = 1/\ 4\pi^2 (1.59e9)^2\ (1e\text{-}12) = 10\ nF$



Experiment 21

Using formula 21-2:

$$f = \frac{c}{\lambda_{fs}} = \frac{2.9974e10\,\frac{cm}{s}}{2(4.039)cm} = 3.7\,GHz$$

	Expected (lower)	Measured (lower)	Expected (upper)	Measured (upper)
Propogation Frequency C-Band	4.9 GHz	4.9 GHz	7.05 GHz	7.05 GHz

Table 21-1

Using equation 21-1 @4.9 GHz

$$Z_{OL} = \frac{120\pi}{[1-(\frac{\lambda fs}{\lambda_{CL}})^2]^{1/2}} = \frac{120\pi}{[1-(\frac{6.12}{8.078})^2]^{½}} = 577\Omega$$