PSpice for Analog Communications Engineering

PSpice for Analog Communications Engineering

Paul Tobin

www.morganclaypool.com

ISBN: 1598291602 paperback
ISBN: 9781598291605 paperback

ISBN: 1598291610 ebook
ISBN: 9781598291612 ebook

DOI 10.2200/S00071ED1V01Y200612DCS009

A Publication in the Morgan & Claypool Publishers series
SYNTHESIS LECTURES ON DIGITAL CIRCUITS AND SYSTEMS #9

Lecture #9
Series Editor: Mitchell A. Thornton, Southern Methodist University

Library of Congress Cataloging-in-Publication Data

Series ISSN: 1932-3166 print
Series ISSN: 1932-3174 electronic

First Edition

10 9 8 7 6 5 4 3 2 1

PSpice for Analog Communications Engineering

Paul Tobin
School of Electronic and Communications Engineering
Dublin Institute of Technology
Ireland

SYNTHESIS LECTURES ON DIGITAL CIRCUITS AND SYSTEMS #9

MORGAN & CLAYPOOL PUBLISHERS

ABSTRACT

In PSpice for Analog Communications Engineering we simulate the difficult principles of analog modulation using the superb free simulation software Cadence Orcad PSpice V10.5. While use is made of analog behavioral model parts (ABM), we use actual circuitry in most of the simulation circuits. For example, we use the 4-quadrant multiplier IC AD633 as a modulator and import real speech as the modulating source and look at the trapezoidal method for measuring the modulation index. Modulation is the process of relocating signals to different parts of the radio frequency spectrum by modifying certain parameters of the carrier in accordance with the modulating/information signals. In amplitude modulation, the modulating source changes the carrier amplitude, but in frequency modulation it causes the carrier frequency to change (and in phase modulation it's the carrier phase).

The digital equivalent of these modulation techniques are examined in *PSpice for Digital Communications Engineering*, where we look at QAM, FSK, PSK and variants. We examine a range of oscillators and plot Nyquist diagrams showing the marginal stability of these systems. The superhetrodyne principle, the backbone of modern receivers is simulated using discrete components followed by simulating complete AM and FM receivers. In this exercise we examine the problems of matching individual stages and the use of double-tuned RF circuits to accommodate the large FM signal bandwidth.

KEYWORDS

Amplitude modulation, frequency modulation, phase modulation, radio-frequency amplifiers, superhetrodyne receivers, phase lock loops, Nyquist plot, gain and phase margins.

I dedicate this book to my wife and friend, Marie and sons Lee, Roy, Scott and Keith and my parents (Eddie and Roseanne), sisters, Sylvia, Madeleine, Jean, and brother, Ted.

Contents

1. Amplitude Modulation Techniques 1

1.1 Baseband to Passband 1
1.2 The Communications Channel 1
1.2.1 Amplitude Modulation 2
1.2.2 AM Generation: Method 1 3
1.2.3 AM using Analog Behavioral Models: Method 2 4
1.2.4 AM Generation: Method 3 5
1.3 Power in an AM Signal 6
1.3.1 Transmission Efficiency 7
1.4 Trapezoidal Method: Speech-Modulated DSBFC AM Signal 7
1.5 Spectrum of Speech-Modulated AM Signal 9
1.6 The Four-Quadrant AD633 Multiplier IC 9
1.7 Linear Amplitude Modulator 9
1.7.1 Multiplying, Squaring, and Frequency Doubling 12
1.8 Exercises 15

2. AM Diode Detection and Four-Quadrant Multipliers 19

2.1 AM Detection 19
2.1.1 Precision Rectifier 19
2.1.2 Diagonal Clipping Distortion 19
2.1.3 Choice of Time Constant 21
2.2 Automatic Gain Control 23
2.2.1 Probe Log Command 24
2.3 Double-Sideband Suppressed Carrier 25
2.3.1 Double-Balanced Modulator 26
2.3.2 Coherent Detection 26
2.4 DSBSC Production using Four-Quadrant Multipliers 28
2.4.1 DSBSC Demodulation 28
2.5 Exercises 29

3. System Stability, Nyquist Criterion 33

3.1 Nyquist Criterion 33

3.2 JFET Colpitts Oscillator 33
3.2.1 The Feedback Network 35
3.2.2 Closed-Loop Testing 37
3.2.3 The Output File 38
3.2.4 The Oscillator Output 38
3.3 Hartley Oscillator 38
3.4 Quartz Crystal and Equivalent Circuits 41
3.4.1 Quartz Crystal Response 41
3.4.2 CMOS Colpitts Crystal Oscillator 41
3.5 Phase-Shift Oscillator 43
3.5.1 Gain and Phase Margins 44
3.5.2 Nyquist Plot 44
3.5.3 Nyquist Diagram and Oscillators 45
3.6 Exercises 46

4. Superhetrodyne Amplitude Modulation Receivers 51
4.1 Nonlinear Mixing 51
4.2 JFET Nonlinear Mixer Circuit 52
4.2.1 Mixer Output 52
4.2.2 Superhetrodyning 53
4.2.3 Image Frequency 53
4.2.4 Superhetrodyning and Image Frequencies 56
4.3 AM Superhetrodyne Receiver 56
4.3.1 The Input/Mixer Stage 57
4.3.2 The Local Oscillator: Arithmetic Selectivity 58
4.3.3 JFET-Tuned Radio Frequency Amplifier 60
4.3.4 RF-Tuned Amplifier Measurements 60
4.4 Measuring the Output Impedance of an RF Amplifier 62
4.4.1 AC Equivalent Circuit 64
4.5 BJT Bandpass Amplifier 65
4.5.1 Tuning Capacitance 66
4.6 Diode Detection and Automatic Gain Control 67
4.7 Power Amplifier Stage 67
4.8 Audio Output Signals 67
4.9 RF Signals 69
4.10 Speech Scrambling 69
4.11 Exercises 70

5. **Frequency Modulation Principles** 75
- 5.1 Modulation Index 75
- 5.2 FM Spectrum 76
- 5.3 FM Production using the VSFFM Generator Part 77
  - 5.3.1 Power in an FM Signal 78
- 5.4 Varactor Diode 79
  - 5.4.1 FM Oscilloscope Display 81
  - 5.4.2 FM Preemphasis and Deemphasis 81
- 5.5 FM Stereo Generation 83
  - 5.5.1 FM Baseband Stereo Signals 84
- 5.6 Replacing ABM Parts with Circuitry 86
- 5.7 FM Stereo Reception 87
- 5.8 Exercises 88

6. **Superhetrodyne Frequency Modulation Receivers** 95
- 6.1 FM Superhetrodyne Receiver 95
  - 6.1.1 Mixer Stage 96
- 6.2 Coupled-Tuned RF Amplifiers 97
  - 6.2.1 Double-Tuned Intermediate RF Amplifier 99
- 6.3 Automatic Gain Control 100
- 6.4 The Phase-Lock Loop Detector 101
  - 6.4.1 PLL Compensation 103
  - 6.4.2 The Lock and Capture Range 105
  - 6.4.3 ABM Phase-Locked Loop 105
- 6.5 Frequency Demodulation 106
  - 6.5.1 Receiver Waveforms 109
- 6.6 Exercises 109

7. **Noise** 115
- 7.1 Sources of Noise 115
- 7.2 Noise Factor and Noise Figure 116
- 7.3 Defriis' Formula 116
  - 7.3.1 Common Emitter Amplifier 117
- 7.4 The Output File 117
- 7.5 Probe Expression Commands 118
- 7.6 The "(if, then, else)" Command 118
- 7.7 Importing Noise 121
  - 7.7.1 Adding Noise to the Input Signal 121

7.8 Exercises 122
Appendix A 129
References 133

Index 135

Author Biography 139

Preface

In book 1, PSpice for Circuit Theory and Electronic Devices, we explained in detail the operational procedures for the new version of PSpice (10.5) but I include here a very quick explanation of the project management procedure that must be followed in order to carry out even a simple simulation task. Before each simulation session, it is necessary to create a project file by following the procedure as shown in Figure 1. This will not be mentioned in the text as it becomes tedious for the reader seeing the same statement "Create a project called Figure 1-008.opj etc" before each experiment. After selecting Capture CIS from the Windows start menu, select the small folded white sheet icon at the top left hand corner of the display as shown.

Enter a suitable name in the **Name** box and select **Analog or Mixed A/D** and specify a **Location** for the file. Press **OK** and a further menu will appear so tick **Create a blank project** as shown in Figure 2. In the project management area, expand the **DC_circuits** directory (or

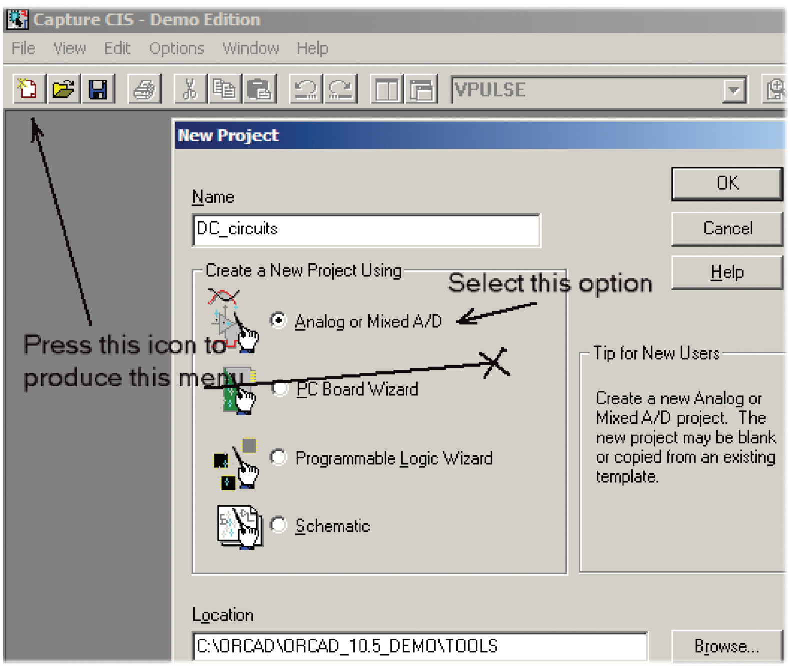


FIGURE 1: Creating new project file

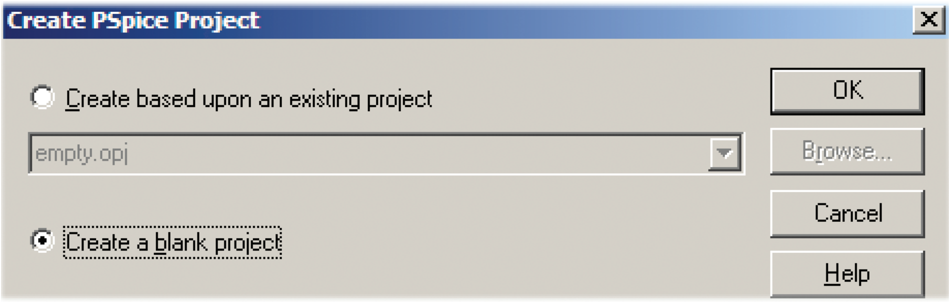


FIGURE 2: Create a blank project

whatever you called the project) to produce an empty schematic area called **Page 1** where components are placed. Libraries have to be added, (**Add library**) by selecting the little **AND** symbol from the right toolbar icons. The easiest method is to select all the libraries. However, if you select **Create based upon an existing project**, then all previously used libraries associated with that project will be loaded.

It is a sad fact that analog communications, over the last few years, has taken a back seat to digital communications. I suppose this is an inevitable step but still, the principles of analog communications have a lot of important concepts that will never go away. To this end, this book covers those important principles through the amazing teaching world of PSpice simulation. In Chapter 1 we look at various amplitude modulation methods. Here we import speech ASCII signals [ref 6] and look at the time and frequency domains signals. The transmission efficiency is considered by looking at the power signals and leads us to examine suppressed carriers methods in Chapter 2. In this chapter we select the correct choice of components for the detection circuit and the AGC circuit. Double sideband carrier suppressed techniques using the AD633 four-quadrant multiplier IC are investigated.

In Chapter 3, we examine a range of oscillators and include the important topic of stability by looking at gain and phase margins in a frequency response plot and a Nyquist plot for assessing stability. Chapter 4 deals with the AM superhetrodyne receiver and all the problems associated when you couple stages together. Here, we examine JFET and BJT mixers for producing the superhetrodyne effect and the associated problem of image frequency. Chapter 5 looks at frequency modulation principles and the phase lock loop for recovering the modulation signal. We generate an FM stereo signal using imported speech signals in order to examine the spectrum associated with stereo production. In Chapter 6 we look at an FM superhetrodyne receiver which uses double-tuned RF amplifiers to accommodate the larger bandwidth associated with FM transmission. Chapter 7 examines the important concept of noise and how we can import noise into circuits for assessment purposes.

ACKNOWLEDGMENTS

I would like to thank Joel Claypool of Morgan and Claypool publishers for taking on my five books on PSpice. I would also like to thank my friend and colleague Anthony (Tony) Kelly for letting me bounce ideas off his very analytical clear-thinking mind and for some circuit contributions.

CHAPTER 1

Amplitude Modulation Techniques

1.1 BASEBAND TO PASSBAND

Fig. 1.1 shows the elements of a transmitter/receiver telecommunications system. Source signals may be classified as an analog signal, $\boldsymbol{m(t)}$, where the signal is defined for all values of time, or a digital signal $\boldsymbol{m(nT)}$ defined for discrete values of n only. However, in this book we deal with analog circuits/systems only.

The transmitter modulates the signal source onto a high-frequency carrier positioning the modulating information to a higher frequency location. In amplitude modulation (AM), the modulating signal varies the carrier amplitude, whereas with frequency modulation (FM), the carrier frequency is changed by the modulating information. Phase modulation is similar to FM but with the carrier phase being changed by the modulation signal and not the frequency. The communications channel has many forms: free-space, air, cable, etc., with each channel having a limited bandwidth which restricts the rate at which information may be transmitted. The function of the receiver is to recover the modulating signal $\boldsymbol{m(t)}$, but introduces noise $\boldsymbol{n(t)}$ and distortion in the process (noise limits the information transmission rate).

1.2 THE COMMUNICATIONS CHANNEL

A communications channel, with bandwidth $\boldsymbol{W}$ Hz, attenuates, distorts, and adds unwanted noise to the transmitted signal. These factors limit the distance over which the signal may be transmitted and received with reasonable fidelity (or, for data signals, minimum errors). The signal wavelength, carrier frequency, and velocity of propagation are related as

$$v_p = \lambda f \Rightarrow \lambda = v_p/f \text{ m s}^{-1}. \tag{1.1}$$

A receiver aerial should have a size that is a multiple of the wavelength of the received signal. For example, a 10-kHz carrier transmitted with a velocity of propagation $v_p = 3 \times 10^8$ m s^{-1} and a wavelength of 3×10^4 m, has an aerial length half this value, or 1.5×10^4 m. A 100-MHz carrier frequency produces an aerial that is more efficient and has a practical aerial length of 1.5 m. The carrier repositions the modulating information to a new frequency location and has the advantage (besides small aerial size and increased efficiency), of allowing different

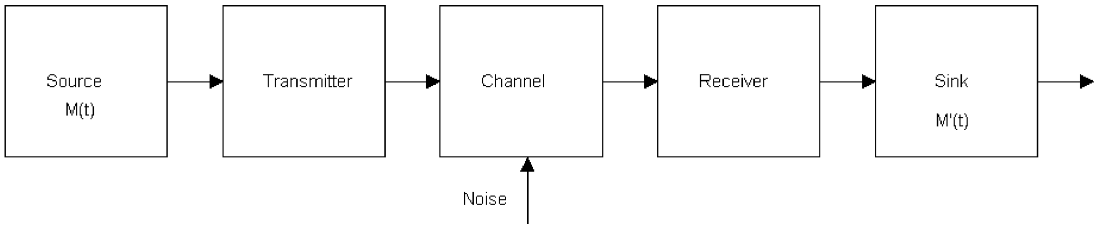


FIGURE 1.1: Basic elements of a telecommunication system

carrier signals to occupy the same channel medium with little interference between each carrier—a process called *frequency division multiplexing* (FDM). The three basic analog modulation techniques are as follows:

- linear amplitude modulation (AM),
- nonlinear frequency modulation (FM), and
- phase modulation (PM).

1.2.1 Amplitude Modulation

Amplitude modulation (AM) is called double sideband full carrier (DSBFC), where the carrier amplitude is changed by the modulating signal, which, in this example, is a single-frequency sinusoidal signal. The instantaneous value of a carrier signal $v_c\ (t)$ is

$$v_c(t) = E_c \cos(2\pi f_c t + \phi) \text{ V} \tag{1.2}$$

where E_c is the peak value of the carrier, and $2\pi ft + \phi$ is the phase at any instant in time. When this carrier is modulated by a sine signal, then the instantaneous value of the carrier is defined as

$$v_c(t) = (E_c + E_m \cos 2\pi f_m t) \cos 2\pi f_c t = E_c \cos 2\pi f_c t + E_m \cos 2\pi f_m t \cos 2\pi f_c t \text{ V}. \tag{1.3}$$

We may expand (1.3) by applying $\cos \mathrm{A} \cos \mathrm{B} = 0.5[\cos(\mathrm{A} - \mathrm{B}) + \cos(\mathrm{A} + \mathrm{B})]$,

$$v_c(t) = E_c \cos 2\pi f_c t + 0.5 E_m \cos 2\pi(f_c - f_m)t + 0.5 E_m \cos 2\pi(f_c + f_m)t \text{ V}. \tag{1.4}$$

The modulation index, m, has a range $0 < m < 1$, is calculated from maximum and minimum carrier amplitudes as

$$m = \frac{\text{maximum} - \text{minimum}}{\text{maximum} + \text{minimum}} = \frac{E_m}{E_c} \Rightarrow E_m = m E_c. \tag{1.5}$$

Replacing E_m from (1.5) into (1.4) yields

$$v_c(t) = E_c \cos 2\pi f_c t + 0.5 m E_c \cos 2\pi(f_c - f_m)t + 0.5 m E_c \cos 2\pi(f_c + f_m)t \text{ V}. \tag{1.6}$$

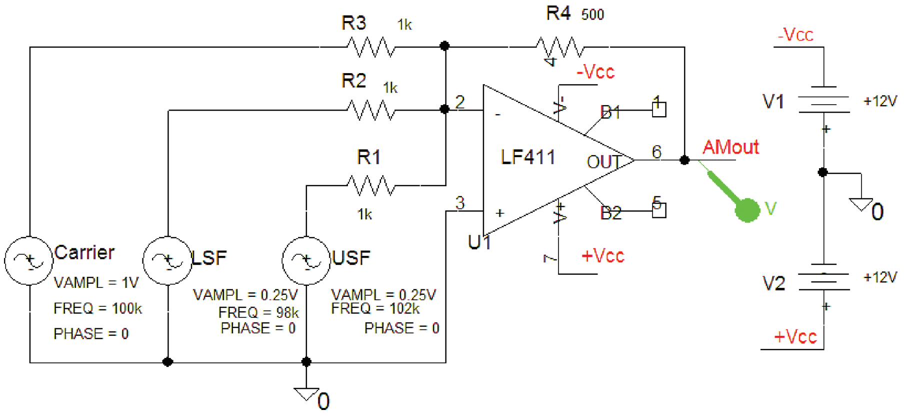


FIGURE 1.2: Amplitude modulator

1.2.2 AM Generation: Method 1

Fig. 1.2 shows AM produced by summing in an inverting summing circuit, three **VSIN** generators representing the components in Eq. (1.6).

A 2-kHz signal modulating a 100-kHz carrier produces two side frequencies at 98 kHz and 102 kHz. Set the side frequency component amplitudes to produce 50% modulation index. Set the **Analysis/Transient** parameters: **Run to time** = 10 ms, **Maximum Step size** = 0.1 μ. Press **F11** to simulate. Note the gain of the opamp is 0.5, so the magnitude of the unmodulated carrier is 0.5 V. Use the two cursors to measure the maximum and minimum values of the AM output displayed in Fig. 1.3. The modulated carrier outline, or envelope, has an upper or lower envelope with the same shape as the modulating signal. The envelope of an AM signal modulated by a complex signal is difficult to interpret on an oscilloscope, so, in general, a

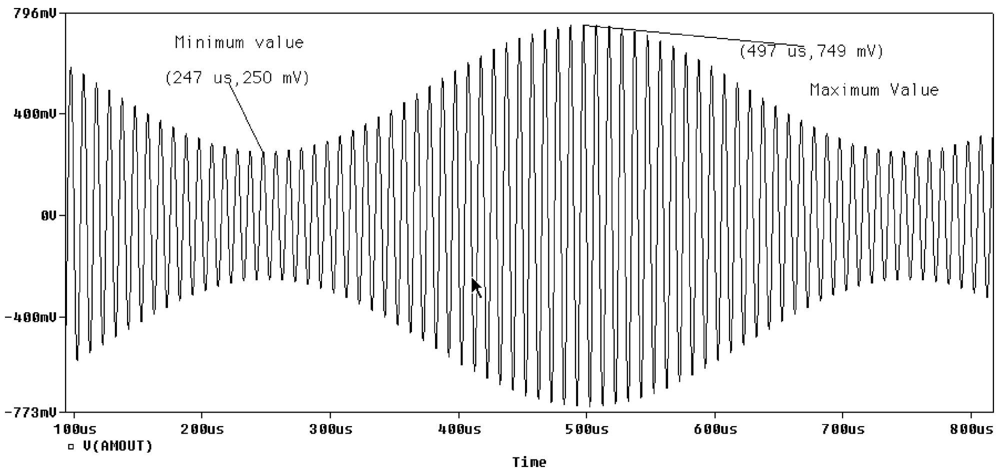


FIGURE 1.3: AM signal

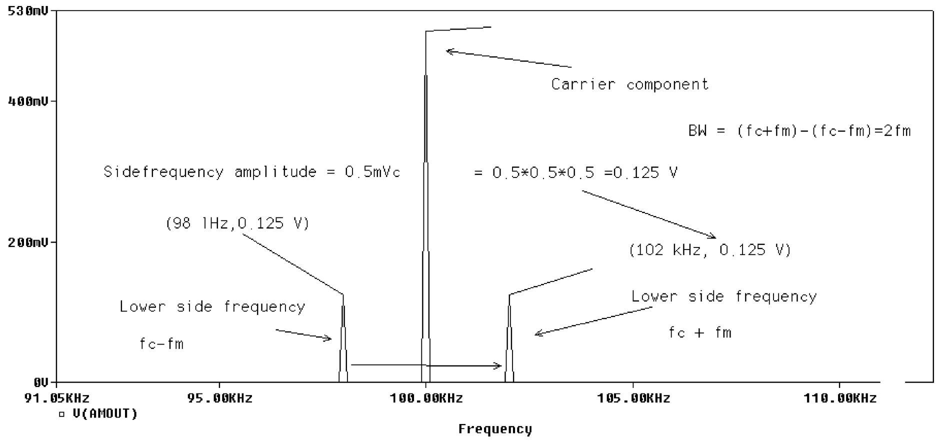


FIGURE 1.4: The AM spectrum

spectrum analyzer is more useful in that it allows us to examine the frequency content of these complex signals. However, for a single modulating frequency the frequency and time domains are both useful.

The Probe **FFT** icon, when pressed produces a spectrum of the AM signal showing the carrier and side frequencies as in Fig. 1.4. Increased spectral resolution is achieved by increasing the **Run to time** and/or reducing **Maximum Step size** in the transient setup menu. The bandwidth is measured as the difference between the upper and lower sidebands:

$$\mathrm{BW} = (f_c + f_m) - (f_c - f_m) = 2f_m \text{ Hz.} \tag{1.7}$$

Thus, the bandwidth for a 2-kHz modulation frequency is $2f_m = 4$ kHz.

1.2.3 AM Using Analog Behavioral Models: Method 2

There are many analog behavioral model (ABM) parts in the **ABM.olb** library and are very useful for producing quick simulation results. Fig. 1.5 shows an ABM part with an output wire called **AMout** for generating AM signals.

Select the **ABM** part, **Rclick** and select **Edit Properties** to display part of the spreadsheet as shown in Fig. 1.6. Enter the AM expression 10*(1 + 0.5* sin(2*3.14*1000*TIME))* cos(2*3.14*1000 000*TIME) in the **EXP1** box. TIME is the **Analysis/Transient** variable and

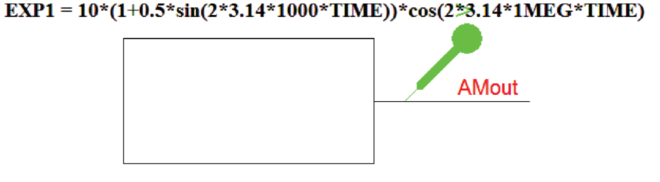


FIGURE 1.5: AM using ABM with one output only

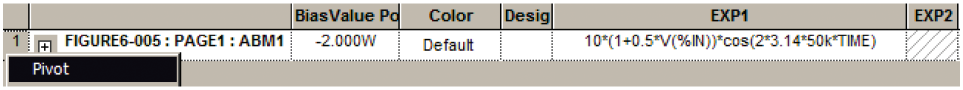


FIGURE 1.6: ABM parameters

be careful about the number of brackets (the number of left brackets always equals the number of right brackets) and **use only round brackets** ( ) not square brackets [ ], with an asterisk "*" between brackets, e.g., (equation1)*(equation2), and between variables and brackets, e.g., (5* sin 20)*10. *Note*: **Rclick** the little square above "**1**" on the left and select pivot to swivel the spreadsheet from the horizontal mode to the vertical mode.

Set the **Analysis/Transient** parameters: **Run to time** = 10 ms **Maximum Step size** = 0.1 μ. Press **F11** to simulate. Use the cursors to measure the maximum and minimum values of the AM output as displayed in Fig. 1.7.

1.2.4 AM Generation: Method 3

Fig. 1.8 shows how an AM signal is generated using an **ABM1** part with a modulating source **VSIN** part for inputting the modulating signal. This source is swept through a range of frequencies to mimic a complex signal containing a number of frequencies and thus generates upper and lower sidebands. The frequency of the modulating **VSIN** part is defined as {**freq**}. **Rclick** the **PARAM part**, select **Edit Properties** and create a **New Row** with **Name** = **FREQ**, and **Value** = 1k in the spreadsheet.

Rclick the **ABM** part, select **Edit Properties** and enter the AM expression in the **EXP1** box. From the **Analysis** menu, set the **Parametric parameters**: **Global parameters, Name** = freq, **Start Value** = 1k, **End Value** = 20k, **Increment** = 2k. Use the same transient plotting

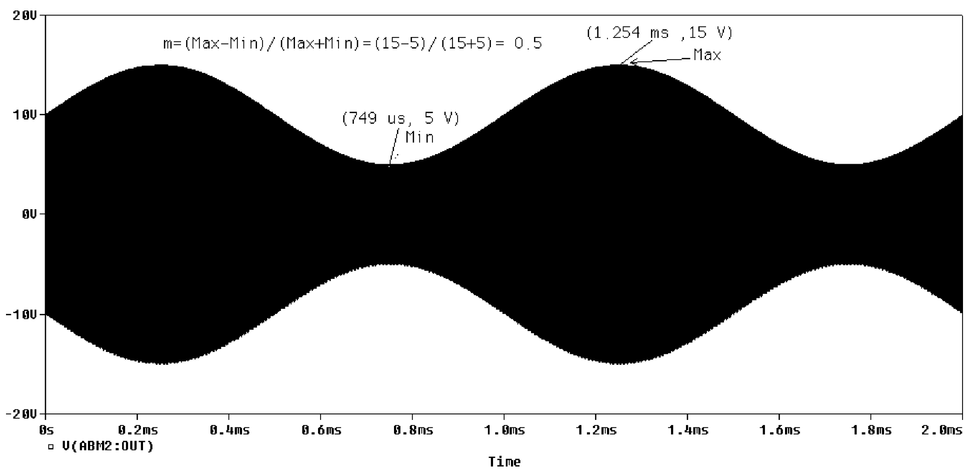


FIGURE 1.7: Amplitude modulation signal

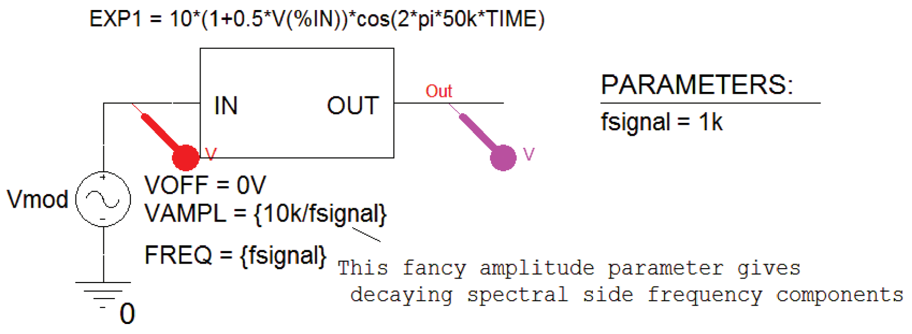


FIGURE 1.8: AM using ABM1 part

parameters as in the previous example, press the **F11** key, and select the **FFT** icon to display the AM spectrum as shown in Fig. 1.9. If you wish to make the amplitudes of the spectral components of the sidebands displayed with different values, make **VAMPL** = 10k/frequency. The 10k is just a gain to make the spectral components large.

When the input source frequency is varied by the parametric sweep, it produces upper and lower sidebands centered on the 50-kHz carrier. The AM bandwidth is the lowest frequency (generated by the highest modulating frequency) in the lower sideband subtracted from the highest frequency in the upper sideband.

1.3 POWER IN AN AM SIGNAL

Double sideband full carrier (DSBFC) modulation efficiency is examined by comparing the carrier power to the information (sideband) power. The unmodulated carrier power developed

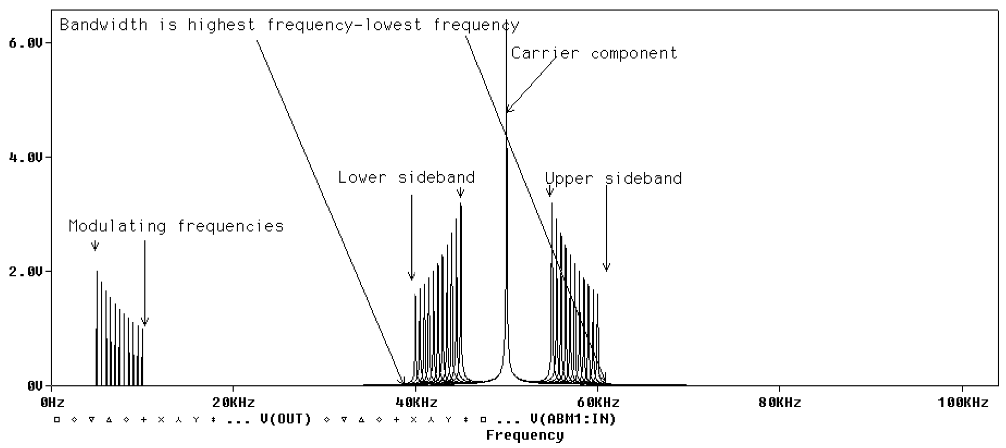


FIGURE 1.9: AM spectrum showing upper and lower sidebands

in a resistance $R\,\Omega$, is

$$P_c = \left(\frac{E_c}{\sqrt{2}}\right)^2 \frac{1}{R} = \frac{E_c^2}{2R}\,\text{W}. \tag{1.8}$$

The power in each sideband is

$$P_{sb} = \left(\frac{mE_c}{2\sqrt{2}}\right)^2 \frac{1}{R} = \frac{m^2E_c^2}{8R}\,\text{W}. \tag{1.9}$$

Combining the carrier and two sidebands components yields an expression for the total power

$$P_T = \frac{E_c^2}{2R} + \frac{m^2E_c^2}{8R} + \frac{m^2E_c^2}{8R} = P_c\left[1 + \frac{m^2}{2}\right]\text{W}. \tag{1.10}$$

We can see from (1.10) how DSBFC is inefficient because a large percentage of the transmitted AM signal is carrier power and a smaller amount is information power.

1.3.1 Transmission Efficiency

A lot of power is wasted by transmitting the carrier of a DSBFC system. For 100% modulation, only 33% of the total transmitted power is information power. The ratio of sideband power to carrier power is a measure of the efficiency of the transmission system:

$$\mu = \frac{P_{sb}}{P_C}. \tag{1.11}$$

The efficiency is increased by increasing the ratio of the modulation sideband power to the total transmitted power. Double-sideband suppressed carrier (DSBSC) eliminates the carrier and transmits the sidebands only. Load the schematic shown in Fig. 1.27 and simulate. Fig. 1.10 shows the sideband and carrier power. Expressions for the carrier, total and sideband power dissipated in a 50-Ω load are entered in the **Trace Expression** box (accessed by pressing the insert button on your keyboard) using Eqs. (1.8)–(1.10). Expressions to plot the power components are: Unmodulated carrier power = V1(Vcarrier)*V1(Vcarrier)/(2*50), sideband power = 0.25*V1(Vcarrier)*V1(Vcarrier)/(8*50), and total power = V1(Vcarrier)*V1(Vcarrier)/(2*50)*(1+0.5*0.5).

1.4 TRAPEZOIDAL METHOD: SPEECH-MODULATED DSBFC AM SIGNAL

Measuring the modulation index from a DSBFC signal is easy for a sinusoidal modulating signal. It is impossible, however, to measure the modulation index from the max and min values of a carrier modulated by a complex signal such as speech. A better technique is the trapezoidal method, where maximum and minimum values are plotted by averaging the complex signal

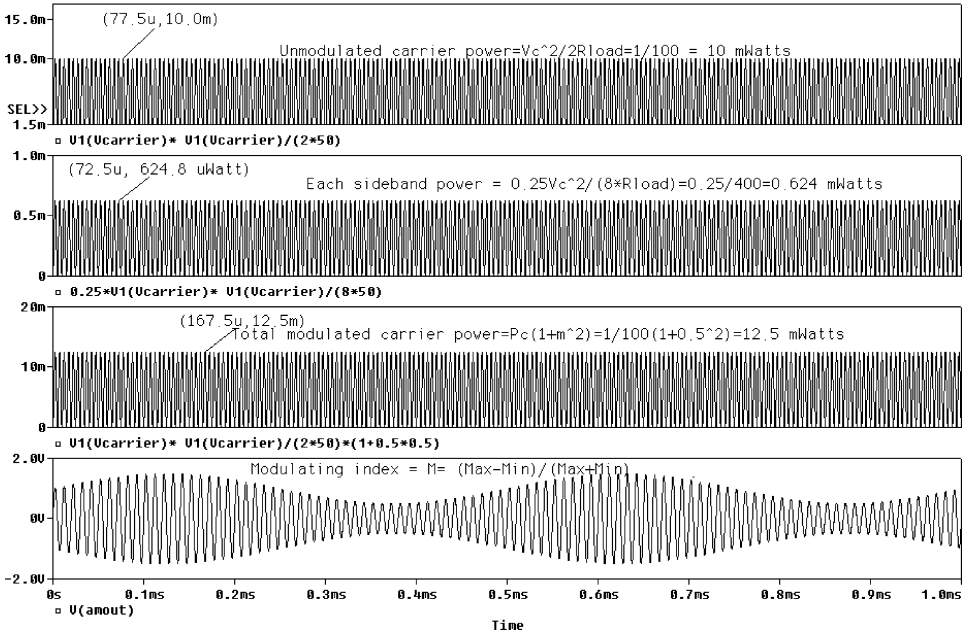


FIGURE 1.10: Sideband and carrier power

over a time. In practice, a trapezoid is produced by connecting the modulation signal to the *y*-plates of the oscilloscope and turning off the internal sweep signal by changing to the *X*–*Y* mode. We simulate this in PSpice by carrying out a transient analysis and resetting the *x*-axis variable from **TIME** to the modulation signal. Fig. 1.11 shows a modulating speech file applied using the **VPWL_F_RE_FOREVER** part. If the magnitude of the speech file is small, then increase the amplitude using the voltage scale factor (**VSF**). The time scale factor (**TSF**) can be used to compresses the time for quick simulation. AM is produced here using ABM parts such as a **CONST** to provide carrier DC level shifting, and a **MULT** part to modulate the carrier and the level-shifted modulating signal.

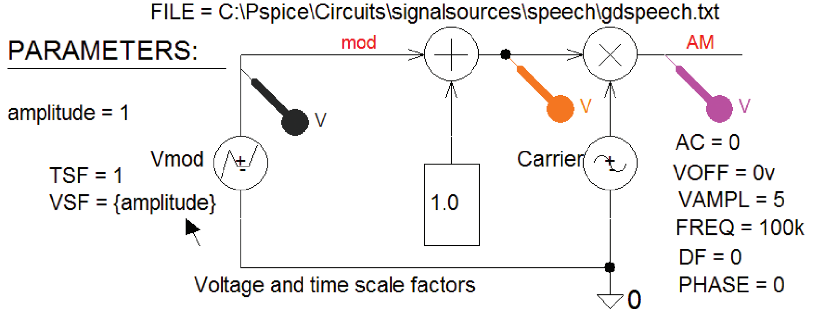


FIGURE 1.11: AM production

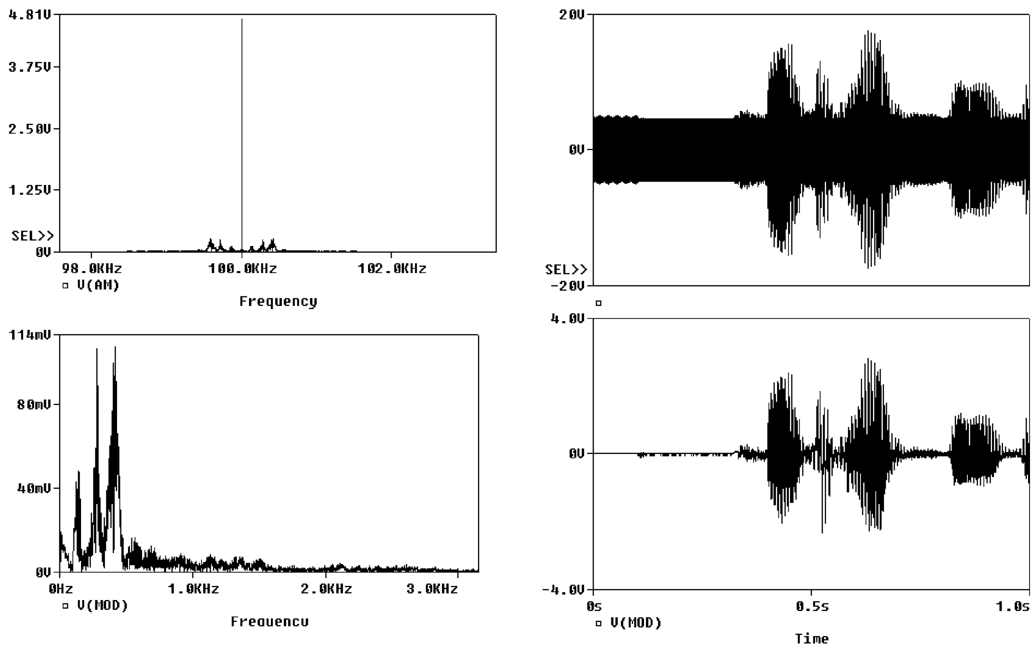


FIGURE 1.12: The speech-modulated carrier

The generator reads in an ASCII speech file comprising time-voltage pairs, by selecting the generator, **Rclick** and select **Edit Properties**. From the spreadsheet enter **C:\Pspice\Circuits\signalsources\speech\speech.txt** in the **File** box. Set the **Analysis Set up/Transient** parameters: **Run to time** = 1 ms. Press **F11** to display the speech-modulated carrier signal as in Fig. 1.12. From the analysis setup, set **Run to time** = 4 s, resimulate and compare simulation times. The **FFT** icon in Probe when selected, displays the spectrum of the DSBFC AM signal.

1.5 SPECTRUM OF SPEECH-MODULATED AM SIGNAL

To investigate different values of the **VSF** scaling factor, use the **Param** part to sweep the amplitude of the VSF part called {**amplitude**}. In the transient analysis menu, selecting **Parametric Sweep/Global parameter** and **Start Value** = 1, **End Value** = 3, and **Increment** = 1. This gives us three plots in Fig. 1.13. A bow-tie pattern is indicative of carrier overmodulation, as shown by plotting **V(mod)@3**. Set the transient analysis parameters so that a few cycles of the AM signal are displayed. From Probe screen change the *x*-axis variable by clicking the space besides the *x*-axis numbers and select **Axis Variable** and enter in the **Trace Expression** box **V1(Vmod)**. The modulation index is easily measured from maximum and minimum values of the trapezoid.

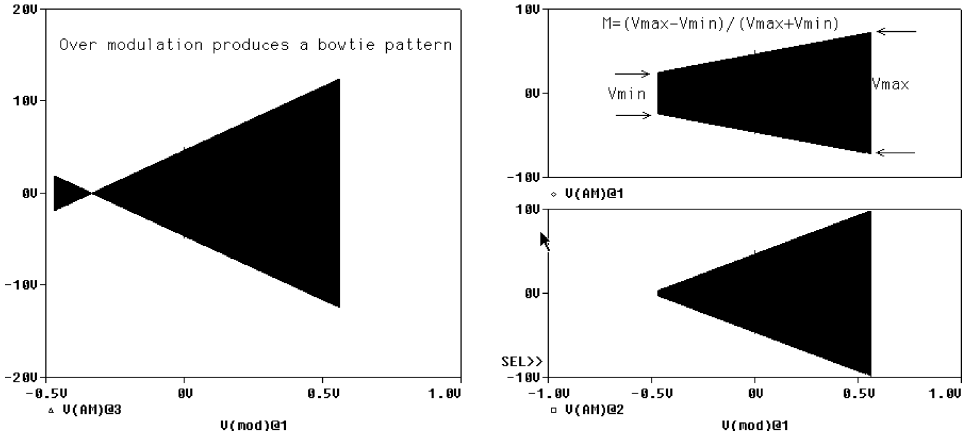


FIGURE 1.13: Trapezoid for carrier modulated by speech

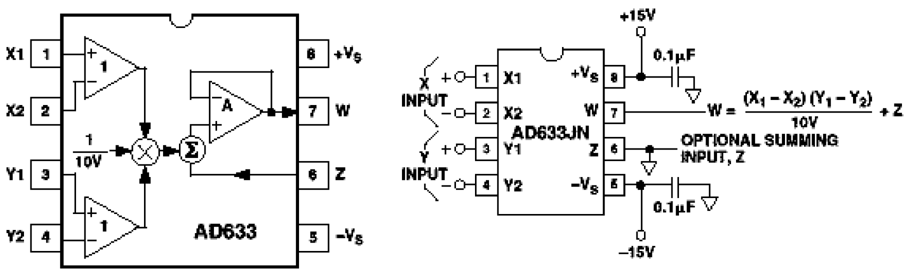


FIGURE 1.14: Pin layout for the AD633 multiplier

1.6 THE FOUR-QUADRANT AD633 MULTIPLIER IC

The AD633 IC is a four-quadrant analog multiplier with differential X and Y inputs, and a high impedance summing input (Z). Fig. 1.14 shows the device connected as a multiplier. Differential X and Y inputs are converted to differential currents by voltage-to-current converters and an internal Zener diode reference to provide a scale factor of 10 V. The transfer function for the AD633 is

$$W = 0.1(X1 - X2)(Y1 - Y2) + Z. \tag{1.12}$$

1.7 LINEAR AMPLITUDE MODULATOR

Fig. 1.15 shows the AD633 multiplier IC configured as a linear amplitude modulator. The carrier and modulation signals are multiplied together to produce a DSBSC signal.

The Z-input adds a further signal to the output, so to create a double-sideband full carrier (DSBFC) we add the carrier signal to the Z input. The Z input is buffered and enables the

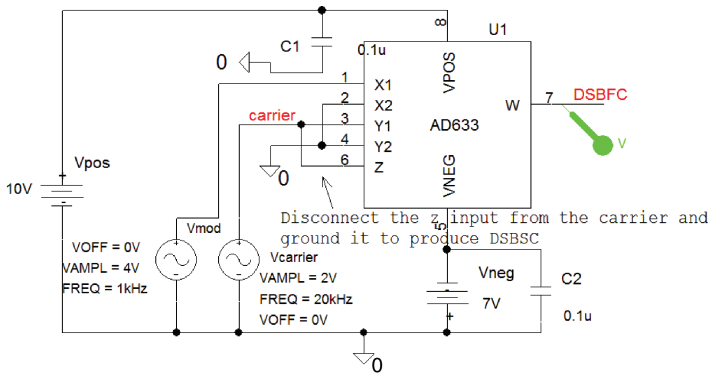


FIGURE 1.15: Amplitude modulator

user to sum the outputs of two or more multipliers. Set **Output File Options/Print values in the output file** to 2 ms, **Run to time** to 10 ms, and **Maximum step size** to 1 μs. Press **F11** to simulate and produce the DSBFC signal as shown in Fig. 1.16.

Note: If convergence problems occur, make Vneg less than Vpos. Select **Probe/Windows/New Window/Tile vertically**, copy the variable into the new window and press the **FFT** icon.

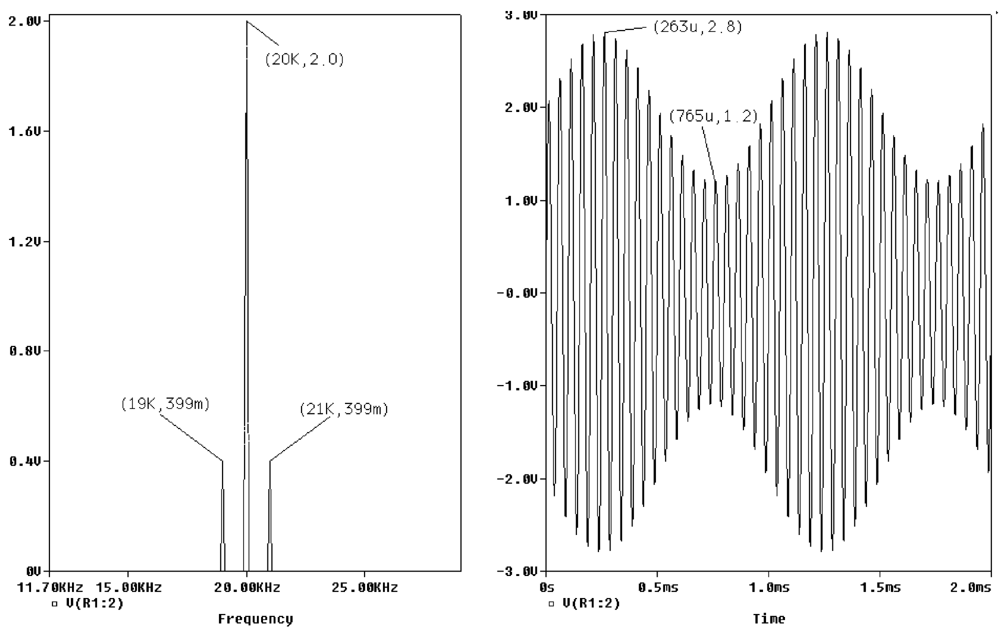


FIGURE 1.16: Pin layout for the AD633 multiplier

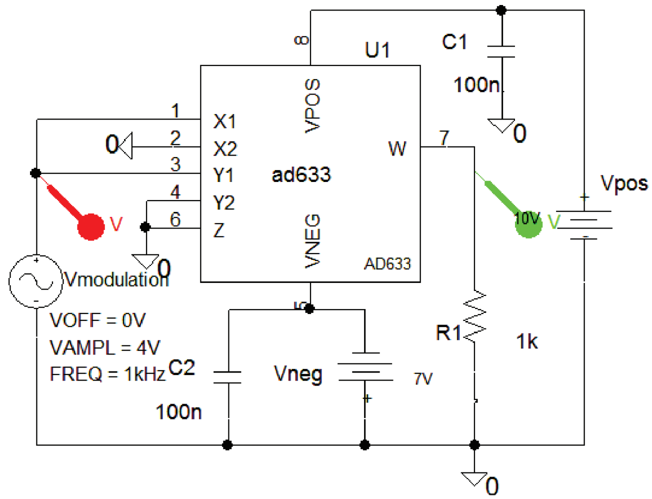


FIGURE 1.17: Squarer (frequency doubler)

1.7.1 Multiplying, Squaring, and Frequency Doubling

The AD633 multiplies X and Y inputs when both negative terminals of the signals are grounded. The inputs are fully differential and in many applications the grounded inputs may be reversed (or both may be driven). Fig. 1.17 is a frequency-doubler circuit where the X and Y inputs are connected in parallel and hence will square the input signal. Equation (1.13) shows how the output contains a DC term which changes with a change in the amplitude of the input signal E:

$$0.1(E \sin \omega t)^2 = 0.05E^2(1 - \cos 2\omega t). \tag{1.13}$$

Set the **Output File Options/Print values in the output file** to 2 ms, **Run to time** to 10 ms, and **Maximum step size** to 1 µs. Press **F11** to simulate.

DC in the output signal is evident in Fig. 1.18 where the signal is now raised above zero.

DC is eliminated in the output using the RC network shown in Fig. 1.19. Set the transient parameters: **Run to time** = 2 ms, and **Maximum step size** = 10 µs. Press **F11** to simulate.

The **VSIN** generator frequency is set to the cut-off frequency 1591 Hz (the −3 dB frequency where the resistance R_f equals X_{cf}, so that $f_c = 1/(2\pi R_f C_f)$. The X input signal leads the input by 45° and is attenuated by $1/\sqrt{2}$. The Y input lags the input by 45° and is attenuated by $1/\sqrt{2}$, so that the X and Y inputs are now 90° out of phase. You may prove this by placing a pair of differential markers across R and C as in Fig. 1.20. Simulate by pressing **F11**.

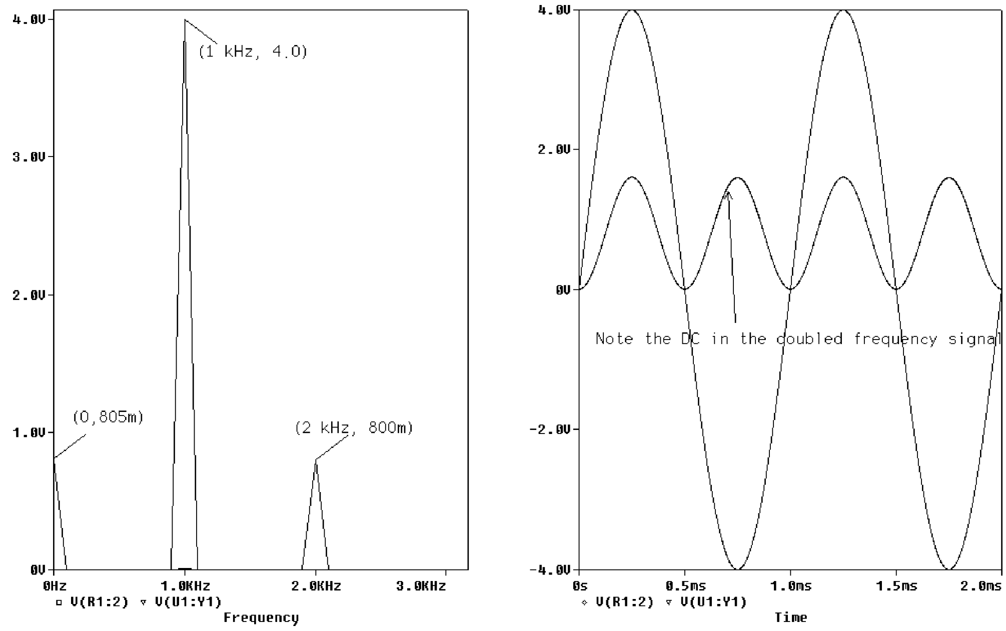


FIGURE 1.18: DC content and double the input frequency

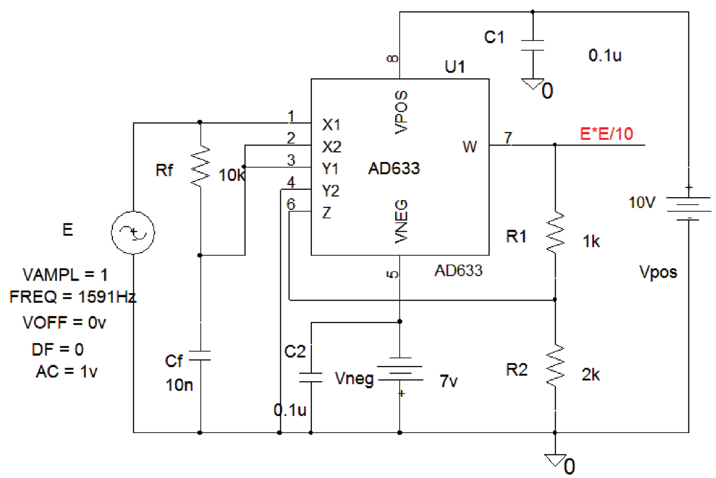


FIGURE 1.19: Eliminating DC in the output

The phase difference between the two voltage signals in Fig. 1.21 is determined indirectly using two cursors to measure the time difference between the maximum of each voltage peak as

$$\theta = 360(\Delta t / T) \tag{1.14}$$

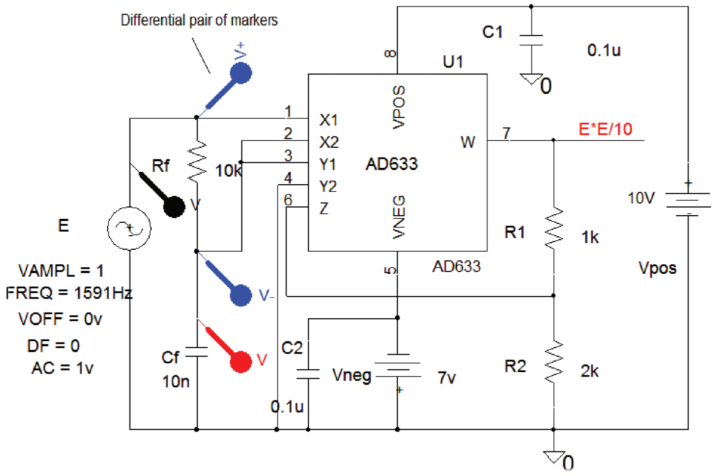


FIGURE 1.20: Differential marker to measure phase

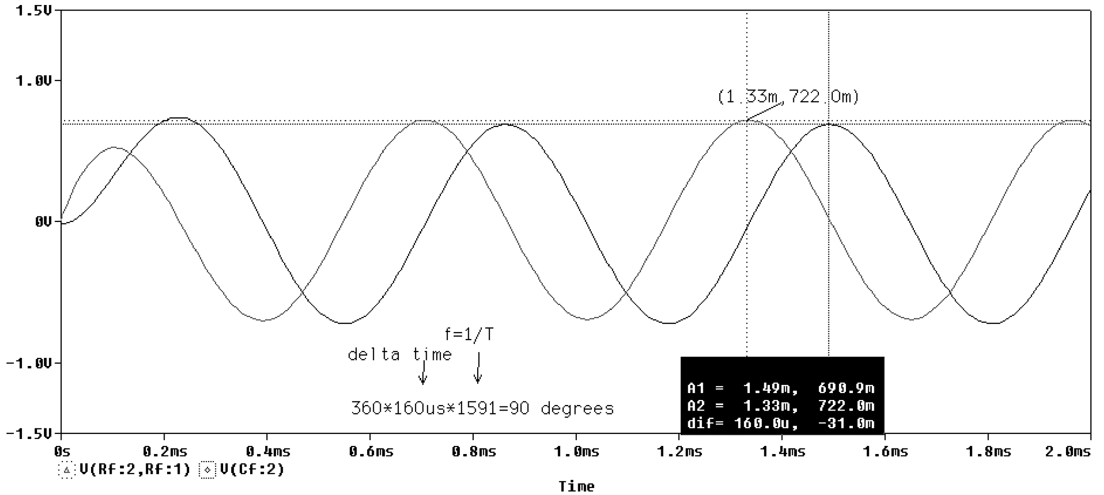


FIGURE 1.21: 90° phase shift between X and Y

where T is the period of the signal. Transient signals should be allowed to die down before time measurements are taken.

The output is expressed as

$$W = E/(10\sqrt{2})(\sin\omega_0 t + 45^\circ)E/\sqrt{2}(\sin\omega_0 t - 45^\circ) = E/40(\sin 2\omega_0 t). \qquad (1.15)$$

The output signal in Fig. 1.22 is symmetrical around zero and hence has no DC content.

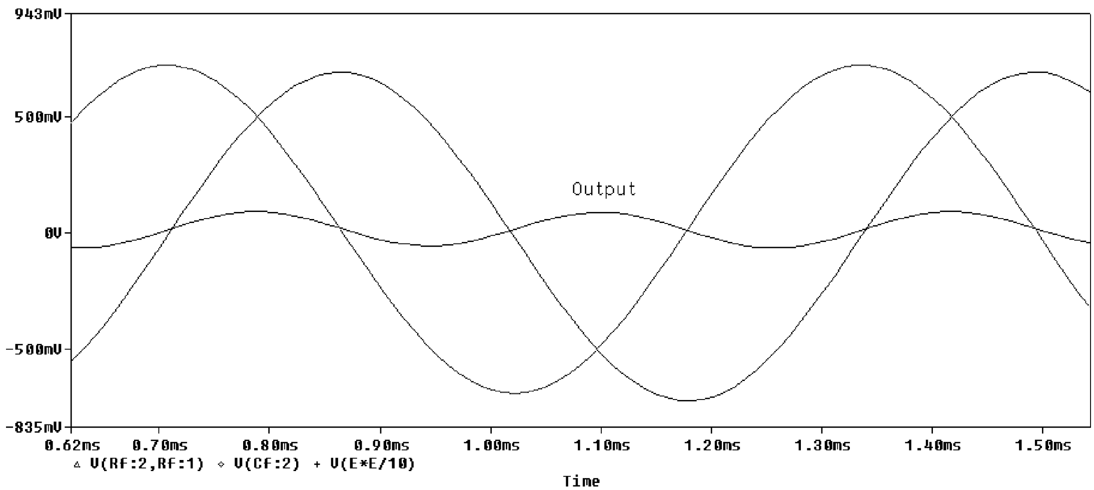


FIGURE 1.22: Probe output

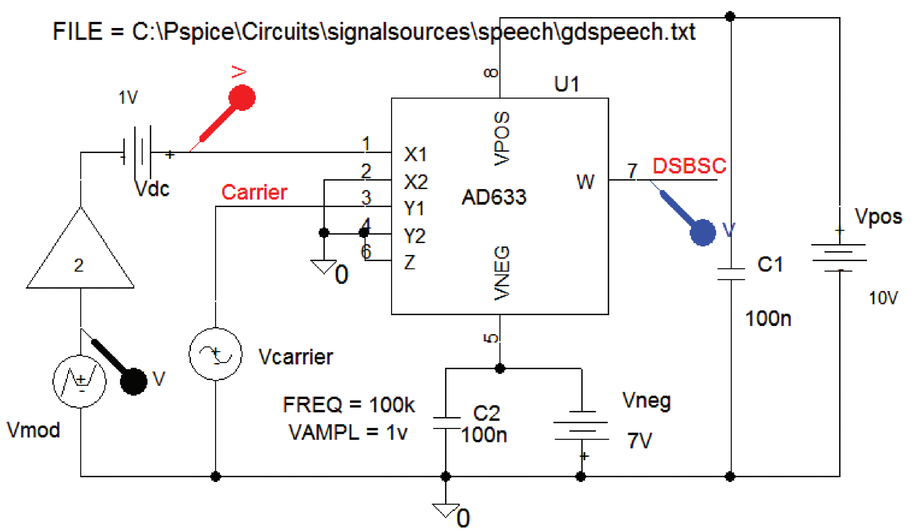


FIGURE 1.23: DSBSC using a speech file

1.8 EXERCISES

(1) Investigate AM DSBFC production in Fig. 1.23 using the speech file "gdspeech.txt" from the signalsources directory. Connect the *Z* input to the carrier and simulate. What is the difference in the output signal when compared to DSBFC? You should change the Probe output so that it collects data at the output marker only (Select the Simulation Setting icon and open the **Data Collection** Tab to prevent a large **.dat** being generated). Reduce the negative supply if convergence problems arise. Use a

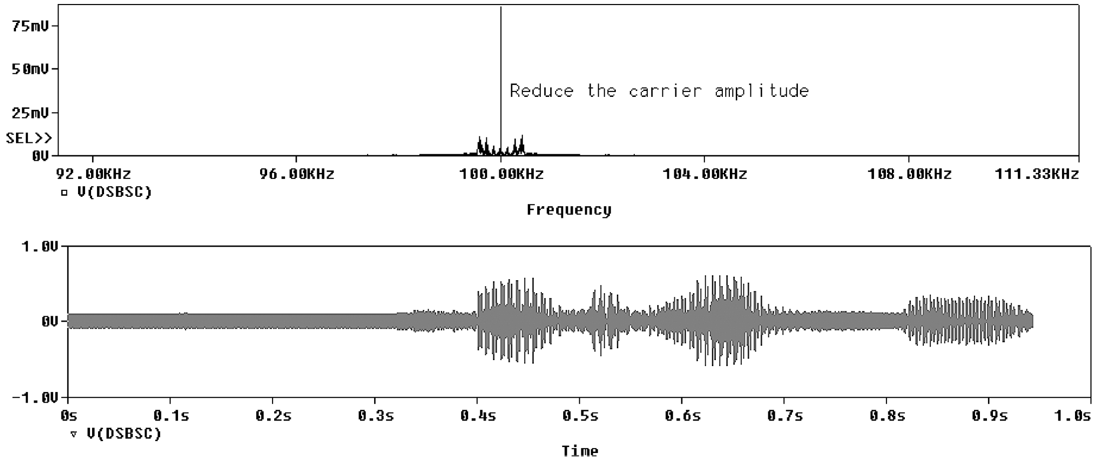


FIGURE 1.24: Speech-modulated AM in time and frequency

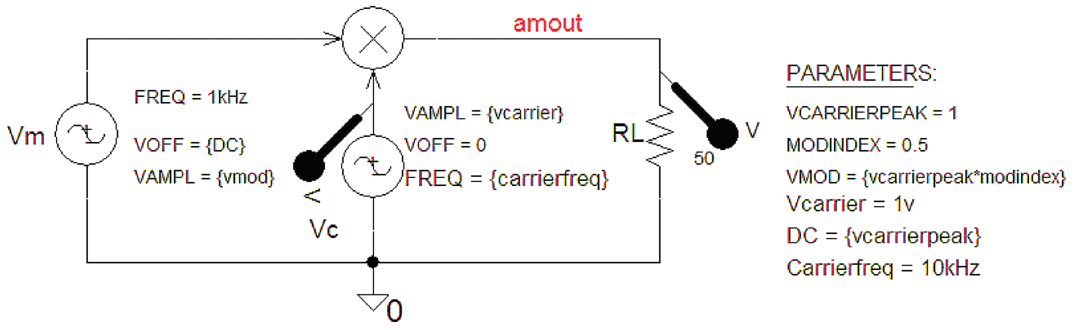


FIGURE 1.25: AM using Param part

Gain part to increase the magnitude of the speech signal, or increase the **VSF** (voltage scaling factor) in the **VPWL_F_RE_FOREVER** generator part.

Use the **Plot/Unsynchronize** facility to display time and frequency waveforms as shown in Fig. 1.24.

(2) Another technique for producing *AM* uses the **Param** part to define the parameters as shown in Fig. 1.25.

The Param parameters are entered in the spreadsheet with names as shown. The AM signal in Fig. 1.26 shows the modulation signal superimposed on the carrier envelope.

(3) Investigate AM using the hierarchical modular block method of drawing a schematic as shown in Fig. 1.27.

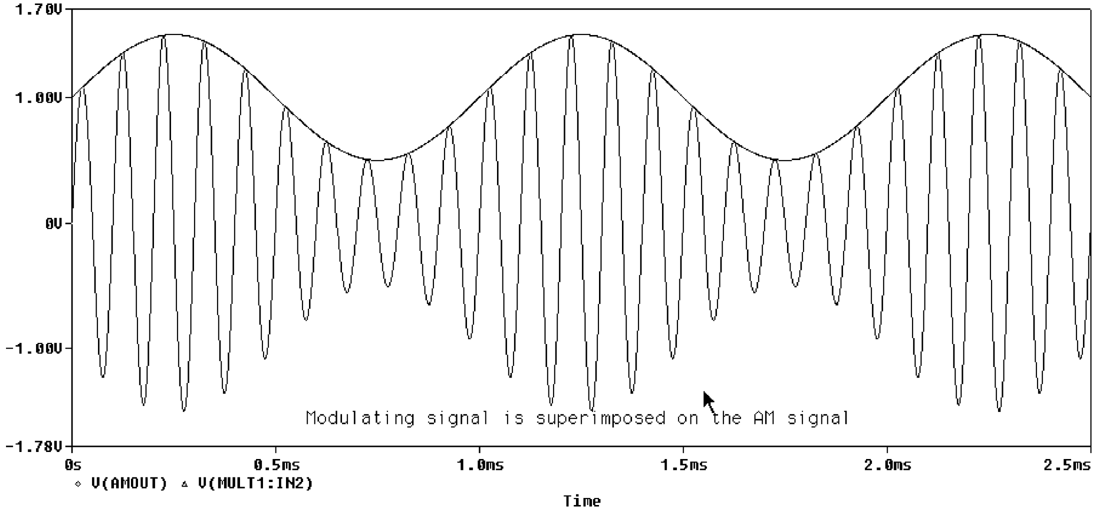


FIGURE 1.26: Carrier and modulated carrier signals

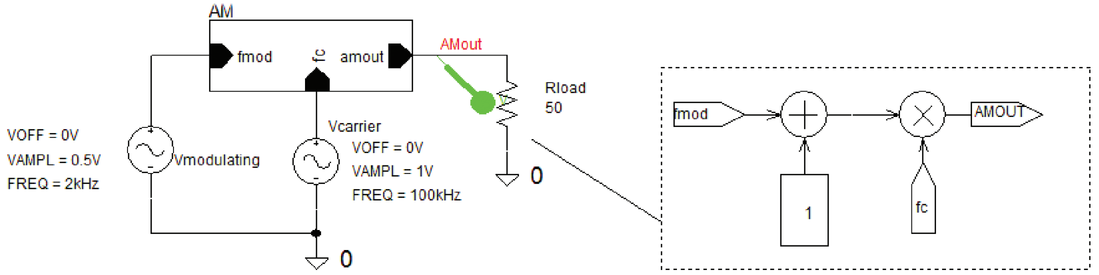


FIGURE 1.27: AM generation

CHAPTER 2

AM Diode Detection and Four-Quadrant Multipliers

2.1 AM DETECTION

The modulating signal is recovered from an AM signal using an envelope detector in Fig. 2.1. Disconnect the capacitor first and investigate the simple rectifier circuit. For the detector to work correctly, it requires a carrier input signal with a minimum amplitude of 200 mV, which is the minimum germanium diode cut-in voltage. For silicon diodes (DIN4148), the minimum input carrier amplitude must be at least 600 mV.

2.1.1 Precision Rectifier

We may overcome the lower limit of the ordinary diode by using the precision rectifier shown in Fig. 2.2.

The precision rectifier signals and the diode characteristic are shown in Fig. 2.3. (To see how to produce the diode characteristic consult the relevant sections in ref 1: Appendix A.)

Fig. 2.4 shows how the negative part of the AM signal is suppressed so that the signal across R1 is a rectified AM signal. Note the distortion components in the spectrum of the rectified wave.

A capacitor connected across the resistor forms a simple *CR* low-pass filter and produces a higher output voltage. During a positive cycle of the AM wave, the diode is forward-biased and the capacitor charges up to the peak voltage. There is a ripple component in the output when the capacitor discharges through the resistor during the time-period between peaks of the AM wave. The ripple magnitude is decreased by increasing the carrier frequency, or changing the *RC* time constant.

2.1.2 Diagonal Clipping Distortion

Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 20ms, and **Maximum step size** = 1us. Tick **Skip the initial transient** and press F11 to simulate.

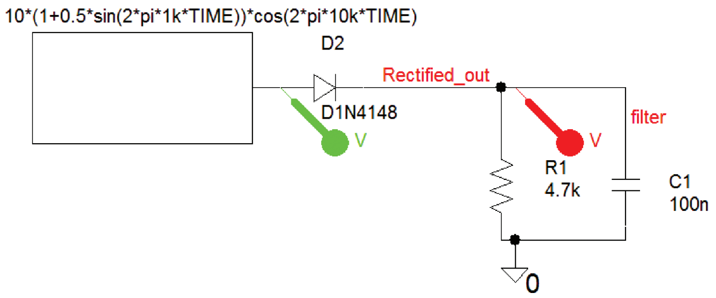


FIGURE 2.1: Diode rectification

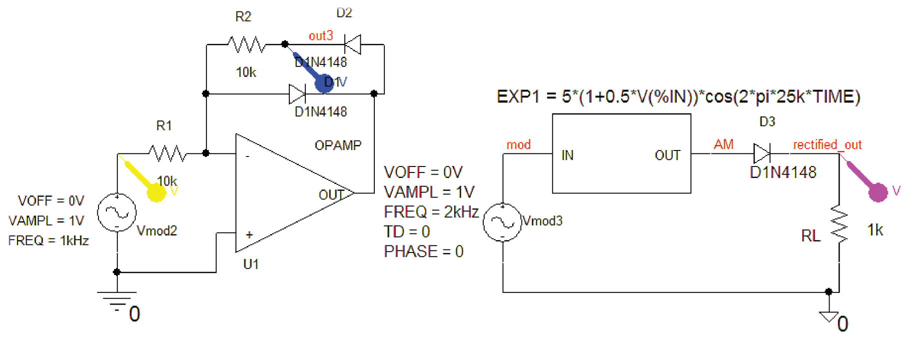


FIGURE 2.2: Precision rectifier

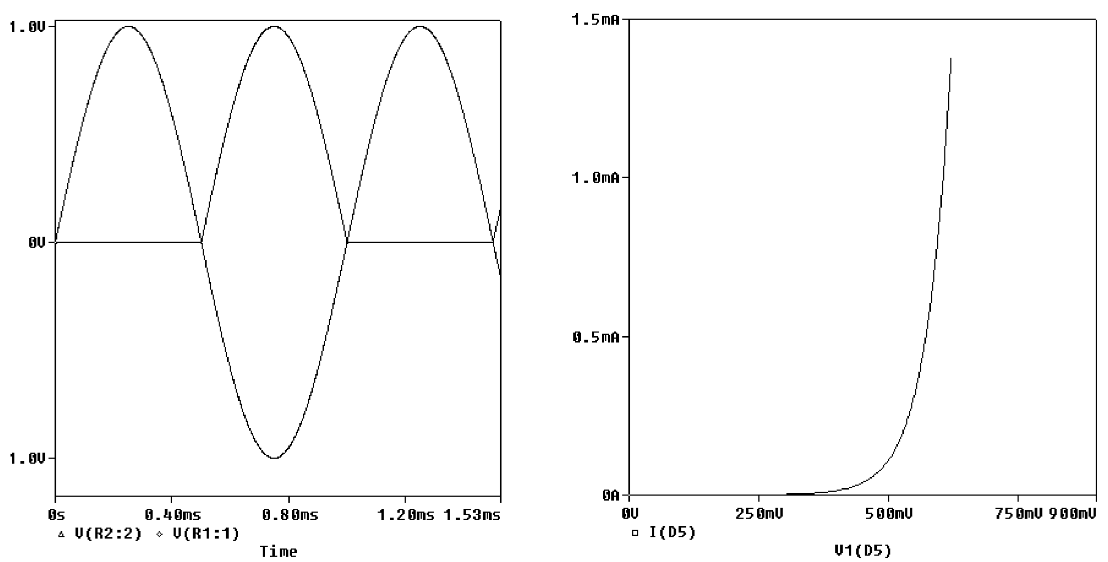


FIGURE 2.3: Precision rectifier output and diode characteristic

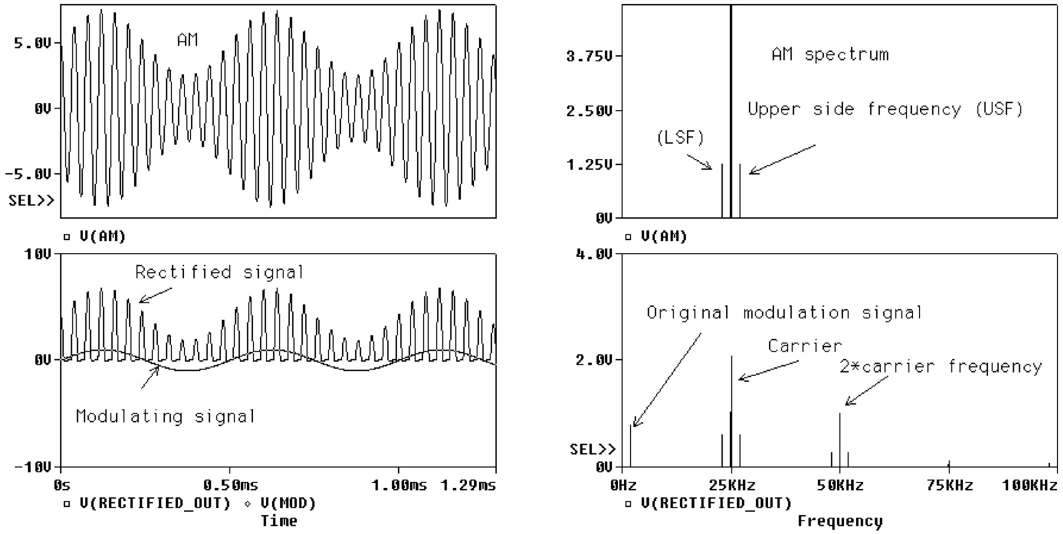


FIGURE 2.4: Half-wave rectification

Diagonal clipping distortion in the recovered modulating signal is shown in Fig. 2.5 and is due to the "tank circuit" time constant ($\tau = CR$) being too large. This is fixed by redesigning the filter "tank" CR circuit.

2.1.3 Choice of Time Constant

If the *RC* time constant is too large, the capacitor voltage cannot follow the rate of decrease in the envelope and results in a distorted demodulated waveform. The correct choice of time

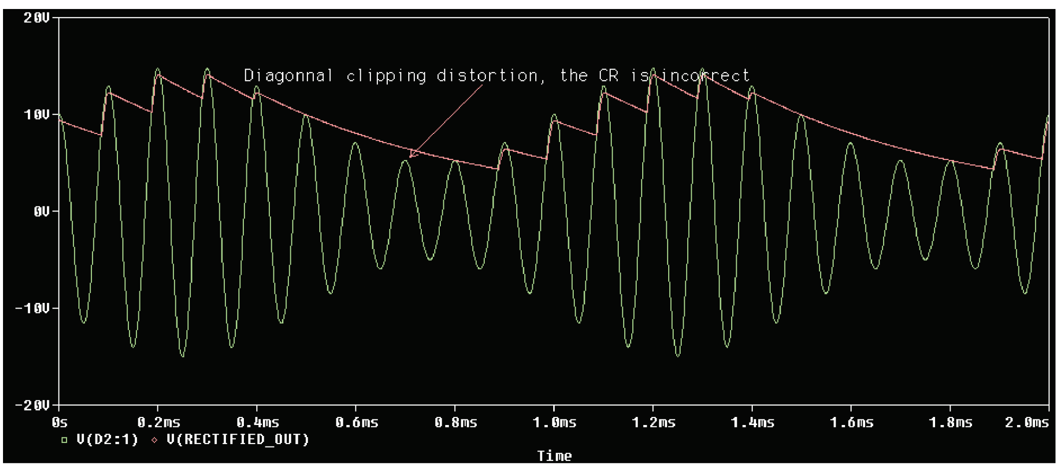


FIGURE 2.5: Diagonal clipping distortion

constant is found as follows:

$$f_m(\max) \leq \frac{1}{2\pi RC} \leq f_c. \tag{2.1}$$

An expression for the correct time constant is determined by equating the rate of change of the modulation envelope with the rate of change of the discharging capacitor voltage. The envelope voltage is

$$V_c = E_c(1 + m\cos\omega_m t)\ \mathrm{V}. \tag{2.2}$$

Differentiating (2.2) yields

$$\frac{dV_c}{dt} = -E_c m\omega_m \sin\omega_m t. \tag{2.3}$$

The discharging capacitor voltage is expressed as

$$V_c = E_c e^{-\frac{t}{CR}}\ \mathrm{V}. \tag{2.4}$$

Differentiating this equation yields

$$\frac{dV_c}{dt} = -\frac{1}{CR}E_c e^{-\frac{t}{CR}} = -\frac{V_c}{CR}. \tag{2.5}$$

Substituting V_c from (2.2) into (2.5)

$$\frac{dV_c}{dt} = \frac{-E_c(1 + m\cos\omega_m t)}{CR} = -E_c m\omega_m \sin\omega_m t. \tag{2.6}$$

The time constant is therefore

$$CR = \frac{(1 + m\cos\omega_m t)}{m\omega_m \sin\omega_m t} = \frac{u}{v}. \tag{2.7}$$

We have to differentiate (2.7) and equate to zero, to get an express for a suitable time constant:

$$\frac{dCR}{dt} = \frac{m\omega_m \sin\omega_m t(-m\omega_m \sin\omega_m t) - (1 + m\cos\omega_m t)(m\omega_m^2 \cos\omega_m t)}{(m\omega_m \sin\omega_m t)^2}. \tag{2.8}$$

Equate the differentiated function to zero:

$$\frac{dCR}{dt} = \frac{-(m^2\omega_m^2 \sin^2\omega_m t + m^2\omega_m^2 \cos^2\omega_m t) - m\omega_m^2 \cos\omega_m t}{(m\omega_m \sin\omega_m t)^2} = 0 \tag{2.9}$$

$$\Rightarrow (-m^2\omega_m^2(\sin^2\omega_m t + \cos^2\omega_m t) = m\omega_m^2 \cos\omega_m t. \tag{2.10}$$

Since $\sin^2\omega_m t + \cos^2\omega_m t = 1$, therefore

$$-m^2\omega_m^2 = m\omega_m^2 \cos\omega_m t \Rightarrow \cos\omega_m t = -m. \tag{2.11}$$

Since $\cos^2 \omega_m t = m^2$, therefore, $\sin \omega_m t = \sqrt{1 - m^2}$. A suitable expression for the time constant in seconds is

$$CR = \frac{1 - m^2}{m\omega_m \sqrt{1 - m^2}} = \frac{\sqrt{1 - m^2}}{m\omega_m} = \frac{\sqrt{1/m^2 - 1}}{2\pi f_m}. \tag{2.12}$$

2.2 AUTOMATIC GAIN CONTROL

An AM signal generated by an **ABM1** part with the AM signal equation inputted, is driven by a modulating signal and applied to the diode detector shown in Fig. 2.6. This circuit generates an automatic gain control (*AGC*) signal that is used to stabilize a superhetrodyne receiver (receiver design is discussed in Chapter 4). The AGC signal regulates receiver gain against amplitude variations in the received input carrier signal. The low-pass *CR* filter, formed from *Ragc* and *Cagc*, has a low cut-off frequency and generates a voltage proportional to the input carrier amplitude. This voltage, when fed back to the first IF amplifier bias circuit, controls the overall receiver gain.

The AGC capacitor filter component is calculated by assuming a 1-Hz AGC cut-off frequency and a resistance of 100 kΩ (it should be much larger than the tank circuit resistance to avoid loading it) as

$$f_{\text{agc}} = 1/2\pi R_{\text{a}} C_{\text{a}} \Rightarrow C_{\text{a}} = 1/2\pi f_{\text{agc}} R_{\text{a}} = 1/2\pi 100k = 1.56\ \mu\text{F}. \tag{2.13}$$

The high-frequency ripple is removed by another low-pass filter whose cut-off frequency f_c is 5 kHz. Again, we assume a 100-kΩ large resistance to avoid loading:

$$f_c = 1/2\pi R_1 C_1 \Rightarrow C_1 = 1/2\pi f_c R_1\ \text{kHz}. \tag{2.14}$$

Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 20 ms, and **Maximum step size** = 1 us. Tick **Skip the initial transient** and press F11 to simulate.

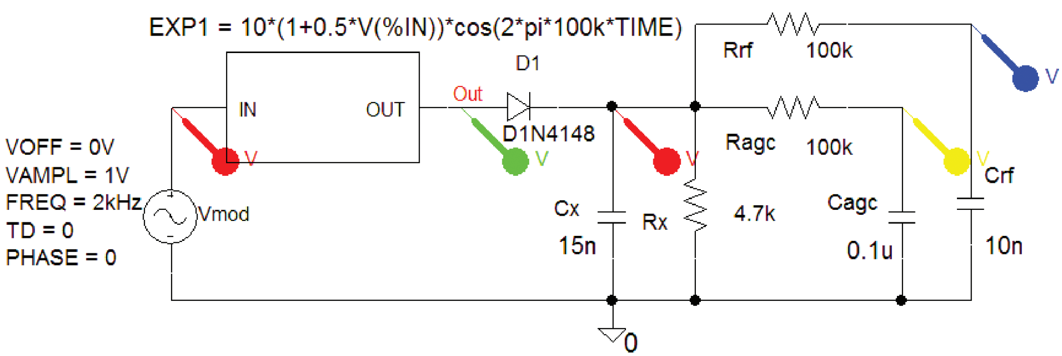


FIGURE 2.6: AM and detection circuit with AGC

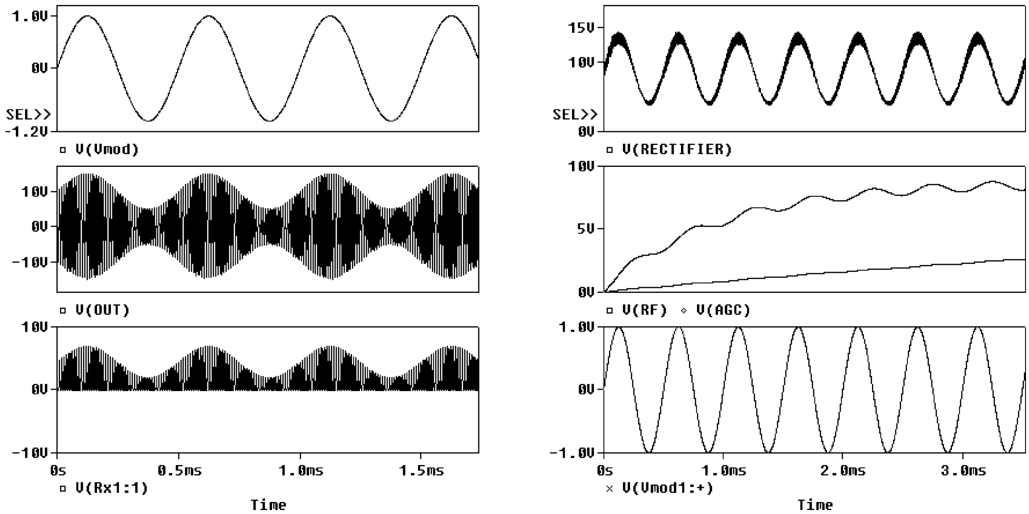


FIGURE 2.7: Note the distortion in the recovered signal

Fig. 2.7 shows the distortion resulting from an incorrect choice of filter time constant.

Increase the magnitude of the input carrier to show how the AGC has a magnitude proportional to the carrier amplitude input.

2.2.1 Probe Log Command

After simulation, many plots may be present in Probe. Each variable should be given a separate sub plot—a tedious and time-consuming effort if you have to repeat the simulation. The **LOG** command, from the **File** menu, creates a text file, or log, of the keyboard keystrokes pressed when separating the waveforms in Probe in the first instance. This log file can be opened from the **File/Run** menu after simulating and avoids having to repeat keystroke sequences each time we simulate for different designs.

Place markers as shown and simulate by pressing **F11**. This creates overlaying plots in Probe, which must be separated into different plots. From Probe, select **File/Log Commands in Probe** to start logging all keystrokes. Selecting this option opens up a further menu where you enter the log sequence file name such as "Fig.2-008.cmd." (This file can be edited in a text editor if so required.) We add multiple plots that are separated for easier interpretation by selecting **Plot** and add a new plot (or shortcut keystroke sequence **alt PP)**. When you select a variable from the signals at the bottom of the Probe screen, it should turn red. Use **ctrl C** and **ctrl V** to place the selected variable into the new plot created by **alt PP**. Repeat for the other variables. You might want to press the **FFT** icon and resize the frequency *x*-axis scale as part of the log commands. When you have finished recording the Probe screen commands, turn off the log recording by selecting **File** and removing the tick from **Log Commands**. To use

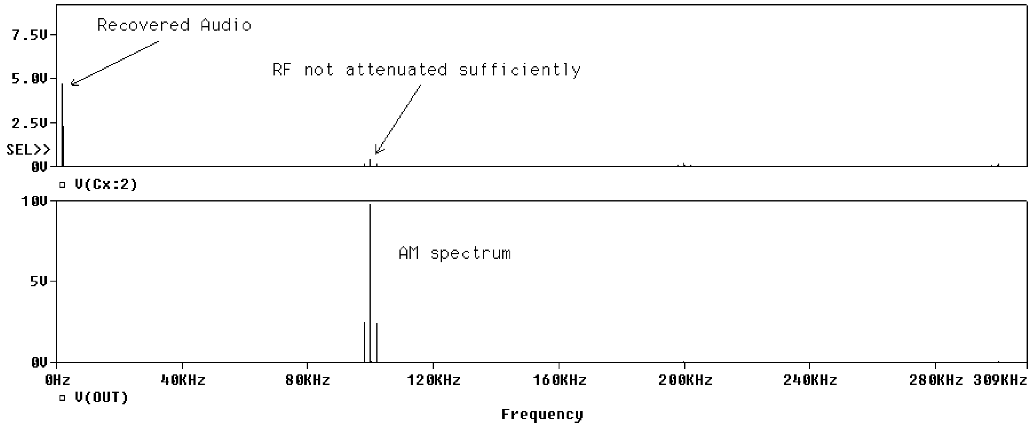


FIGURE 2.8: Use the log command to create multiple screens

this recorded log in Probe simulate the schematic and from the Probe screen, select **File/ Run Commands**. This displays a list of **.cmd** files previously saved. Select "**Fig.2-008.cmd**" file and the screen should automatically display the separated plots. If the **FFT** icon was pressed as a keystroke in the log sequence, then the final screen will show the spectral components. Press the **FFT** icon again if you wish to display in the time domain.

You can open a new window by selecting the **Window/New** menu followed by **Tile Vertical**. Copy the variables into the new window and press the **FFT** icon to display in the frequency domain.

2.3 DOUBLE-SIDEBAND SUPPRESSED CARRIER

Amplitude modulation is inefficient because a high percentage of the transmitted power is wasted in transmitting the carrier and only a smaller percentage is contained in the sidebands. By removing the carrier, we increase transmission efficiency producing double-sideband suppressed carrier (DSBSC). The instantaneous amplitude of a DSBSC carrier modulated by a sinusoidal signal is

$$v_c(t) = E_m \cos 2\pi f_m t E_c \cos 2\pi f_c t \text{ V}. \tag{2.15}$$

Apply $\cos \mathrm{A} \cos \mathrm{B} = 0.5[\cos(\mathrm{A} - \mathrm{B}) + \cos(\mathrm{A} + \mathrm{B})]$ to (2.15):

$$v_c(t) = E_m/2 \cos 2\pi(f_c - f_m)t + E_m/2 \cos 2\pi(f_c + f_m)t \text{ V}. \tag{2.16}$$

This DSBSC AM signal contains two high frequencies (sidebands), but no carrier. Each sideband is expressed in terms of the modulation index m as

$$v_c(t) = 0.5mE_c \cos 2\pi(f_c - f_m)t + 0.5mE_c \cos 2\pi(f_c + f_m)t \text{ V}. \tag{2.17}$$

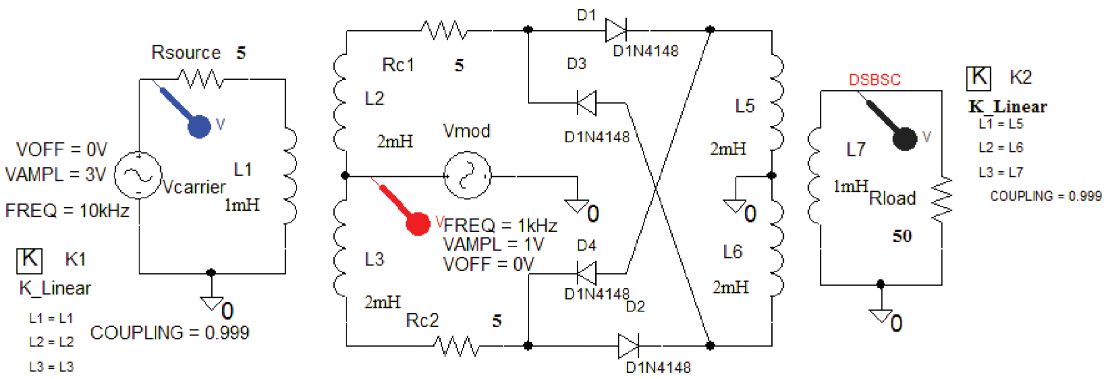


FIGURE 2.9: DSBSC balanced modulator

2.3.1 Double-Balanced Modulator

To generate DSBSC we construct a center-tapped transformer (CTT) in Fig. 2.9 using a **K_Linear** part to link the inductor parts in a double-balanced modulator (BM).

Left click on the **K** part (a component with the square box around it), and enter the parameters as shown. L_1 is the inductor name and the other inductor names for the second center-tapped transformer are ($L_1 = L_5$, $L_2 = L_6$, $L_3 = L_7$) with the coefficient of coupling set to 0.999 in each CTT. Inductors should have resistances (Rc1 and Rc2) placed in series, since PSpice inductors are ideal and do not include coil resistance. Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 10ms, and **Maximum step size** = 1us. Press F11 to display the DSBSC "CUSP" shaped signal outline (The envelope) shown in Fig. 2.10. Select the **FFT** icon to display the signal frequencies. The frequency resolution is increased by making the **Run to time** parameter in the Transient menu larger and/or reducing the **Maximum step size**. The sampled signal spectrum shows how the 10-kHz carrier signal and sidebands are repeated at multiples of the carrier to infinity.

The spectral components are located at 9 kHz and 11 kHz and repeated 29 kHz and 31 kHz (a third harmonic of the carrier). The bandwidth is the lowest side frequency subtracted from the highest side frequency:

$$\text{bandwidth} = (f_c + f_m) - (f_c - f_m) = 2f_m = 2\text{ kHz}. \qquad (2.18)$$

2.3.2 Coherent Detection

Recovering the modulating signal from a DSBSC signal is called coherent detection, where the receiver local carrier must have the same frequency and phase as the original DSBSC carrier. The DSBSC demodulator in Fig. 2.11 is similar to the modulator but the DSBSC is applied to the primary transformer with the local oscillator connected to the secondary transformer.

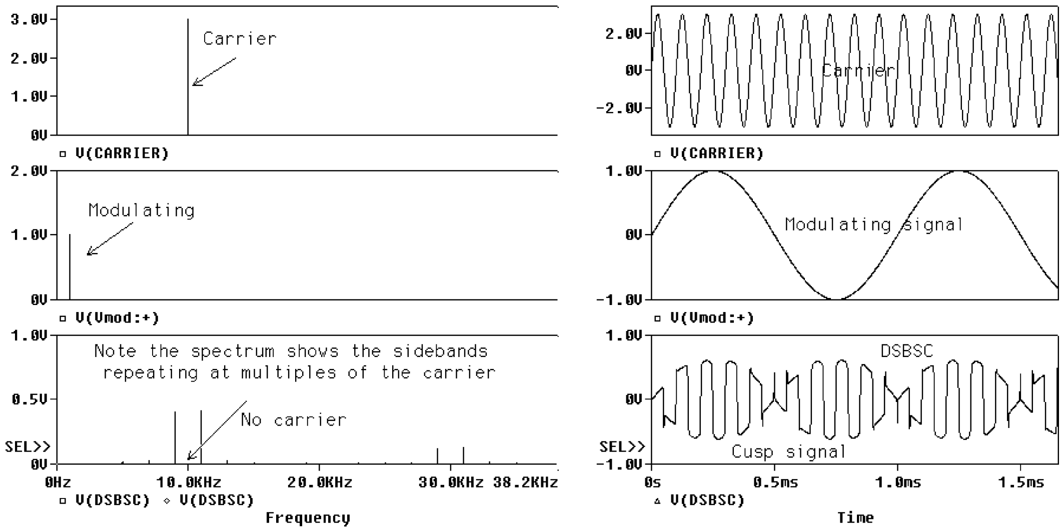


FIGURE 2.10: DSBSC signal and spectrum

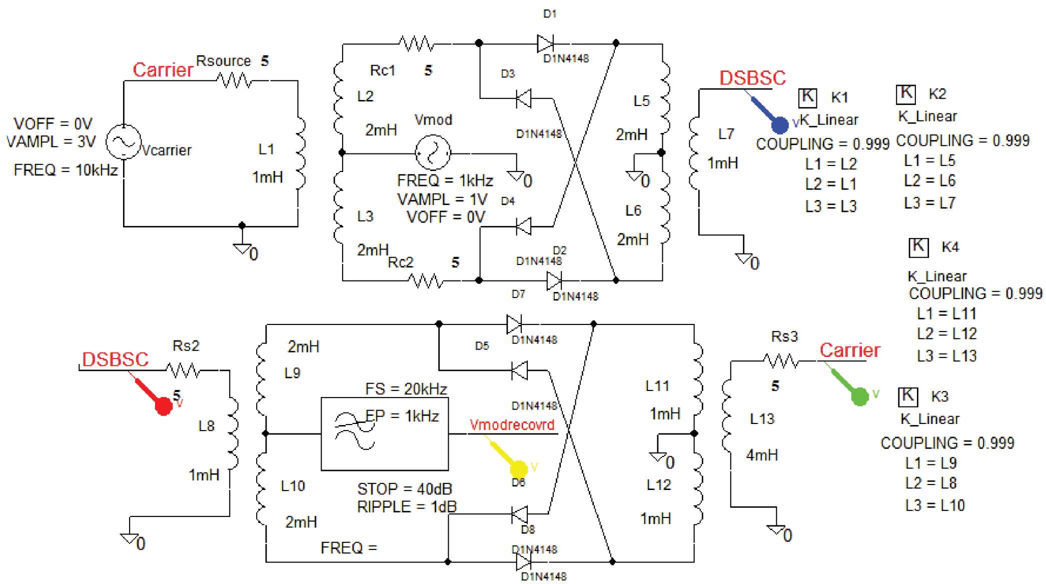


FIGURE 2.11: Coherent detection

The incoming DSBSC signal $m\cos\omega_c t$, is multiplied by a locally-generated carrier that may contain frequency and phase errors, i.e., $f_{LO}(t) = 2\cos[2\pi(f_c + \Delta f)t + \phi]$. An ABM **LOPASS** part low-pass filter attenuates undesirable frequency components to a minimum. To simulate when the local oscillator carrier in the receiver has phase and frequency errors, enter

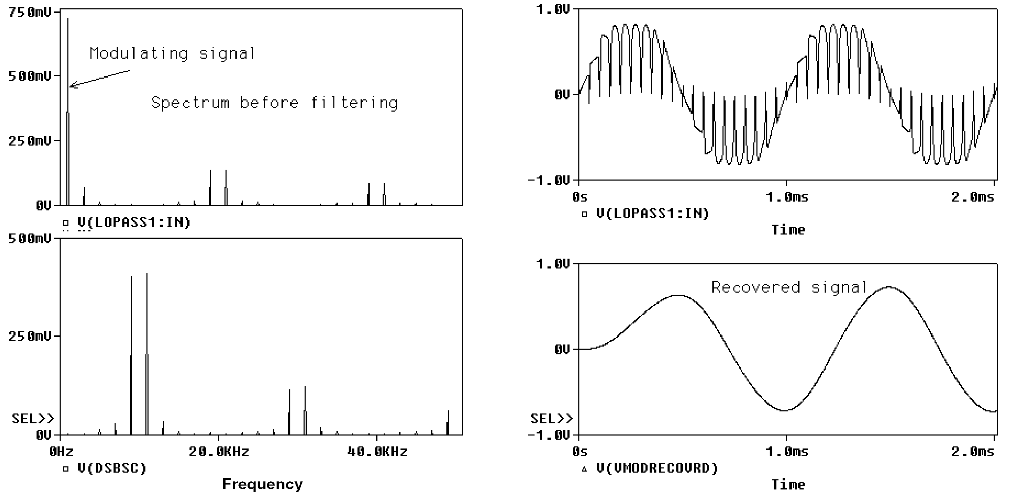


FIGURE 2.12: DSCSC recovery

the following **VSIN** local oscillator parameters and simulate:

- $\Delta f = \phi = 0$,
- $\phi = 0$, $\Delta f \neq 10$ Hz, and
- $\Delta f = 0$, $\phi \neq 30^{\circ}$.

Fig. 2.12 shows DSBSC signals in time and frequency. Repeat for different conditions in the local oscillator.

See exercise (8) on single-sideband (SSBSC) production and carrier recovery.

2.4 DSBSC PRODUCTION USING FOUR-QUADRANT MULTIPLIERS

The AD633 four-quadrant multiplier in Fig. 2.13 produces a DSBSC signal. The schematic will not simulate in the evaluation version (because of the number of components), so instead we simulate the modulator part first and save the output signal in Probe by applying Ctrl C on the DSBSC Probe variable. Paste the copied variable into Notepad and save as an ASCII file.

2.4.1 DSBSC Demodulation

Fig. 2.14 shows the carrier applied via a low-pass filter that changes the phase of the recovered carrier. The DSBSC signal located in **C:\Pspice\Circuits\signalsources\AMFM\DSBSC.txt** was created from the previous schematic by copying the output

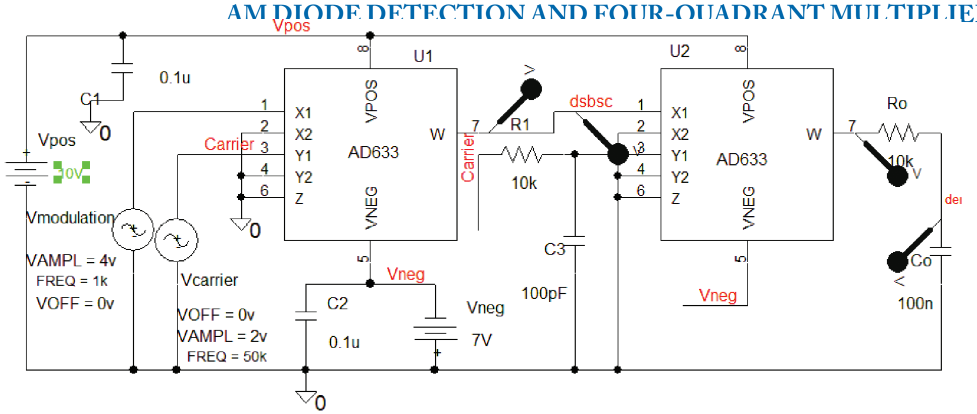


FIGURE 2.13: DSBSC modulator and demodulator

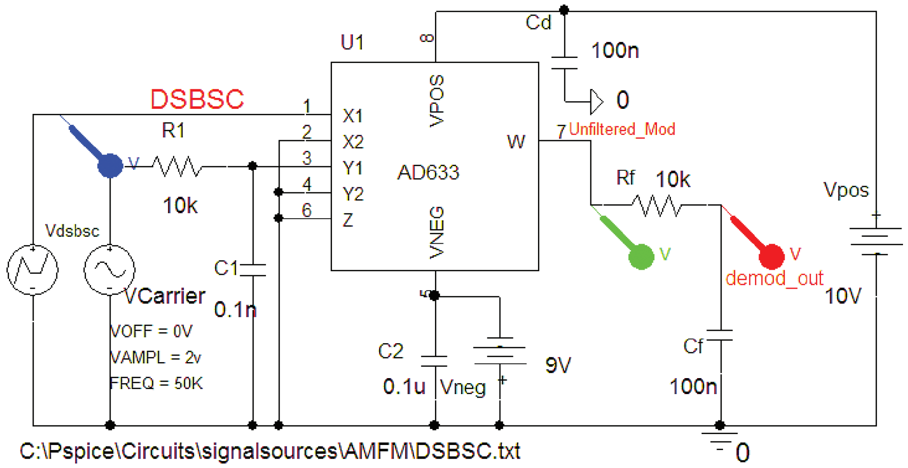


FIGURE 2.14: DSBSC demodulator using a VPWL_F_RE_FOREVER part

variable from Probe. This signal is then applied to pin 1 using a **VPWL_F_RE_FOREVER** generator part.

The power supply is decoupled by connecting a 100 nF capacitor at the pin of the IC (helps with convergence problems). Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 100ms, and **Maximum step size** = 1us. Press F11 to display the filtered demodulated signal shown in Fig. 2.15.

2.5 EXERCISES

(1) Investigate the switching amplitude modulator in Fig. 2.16.
Use the **FFT** and the magnifying icon to examine the largest spectral component and observe the sidebands.

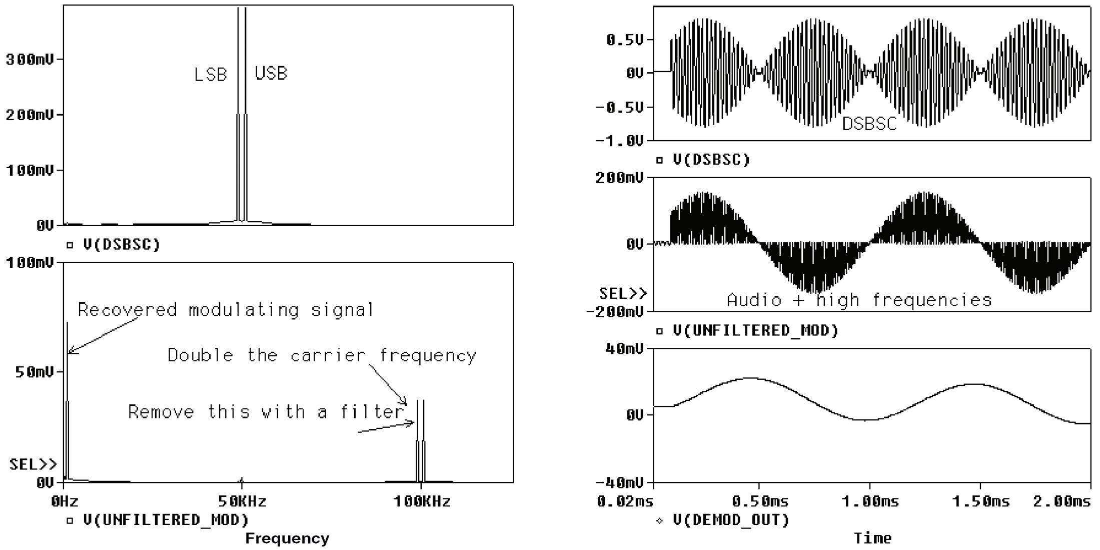


FIGURE 2.15: DSBSC signals and spectra

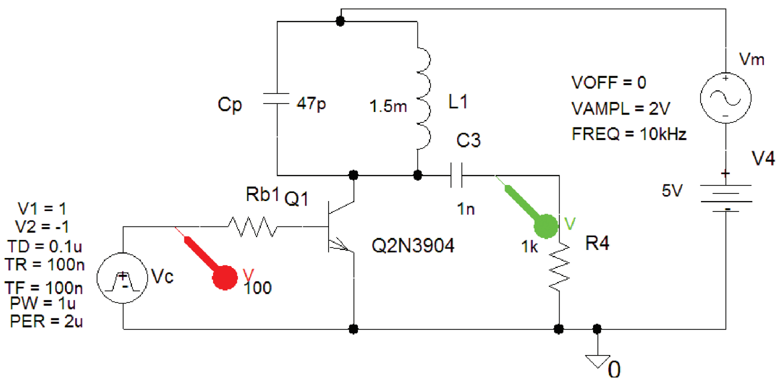


FIGURE 2.16: Amplitude modulation

(2) Investigate the circuit in Fig. 2.17. Here, the AM signal is further modulated by another cosine signal with the same carrier frequency.

(3) From the spectrum in Fig. 2.18, we see the modulation signal but with twice the amplitude. This is due to the "negative frequency" lower side frequency (LSF) being folded back, thus modifying the upper side frequency (USF).

(4) Investigate SSBSC production using the phase-shift method shown in Fig. 2.19. Digital filters can be designed in Matlab using the mfile [b,a] = ellip(4,0.1, 60,100*2/fs; % 4th order LPF@600 Hz.

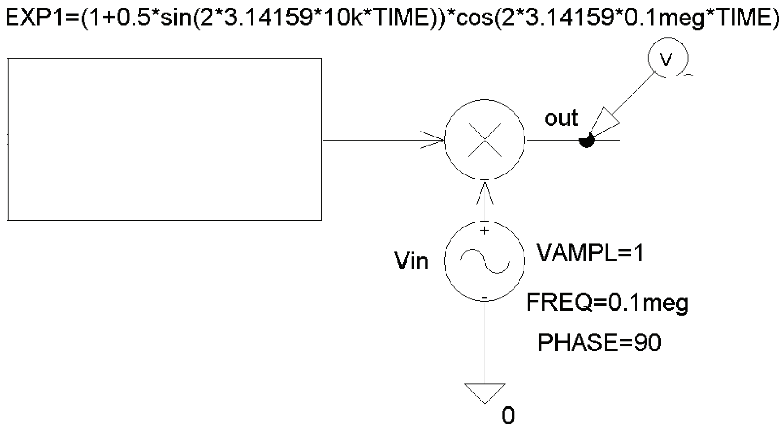


FIGURE 2.17: Mixing the AM signal down to baseband

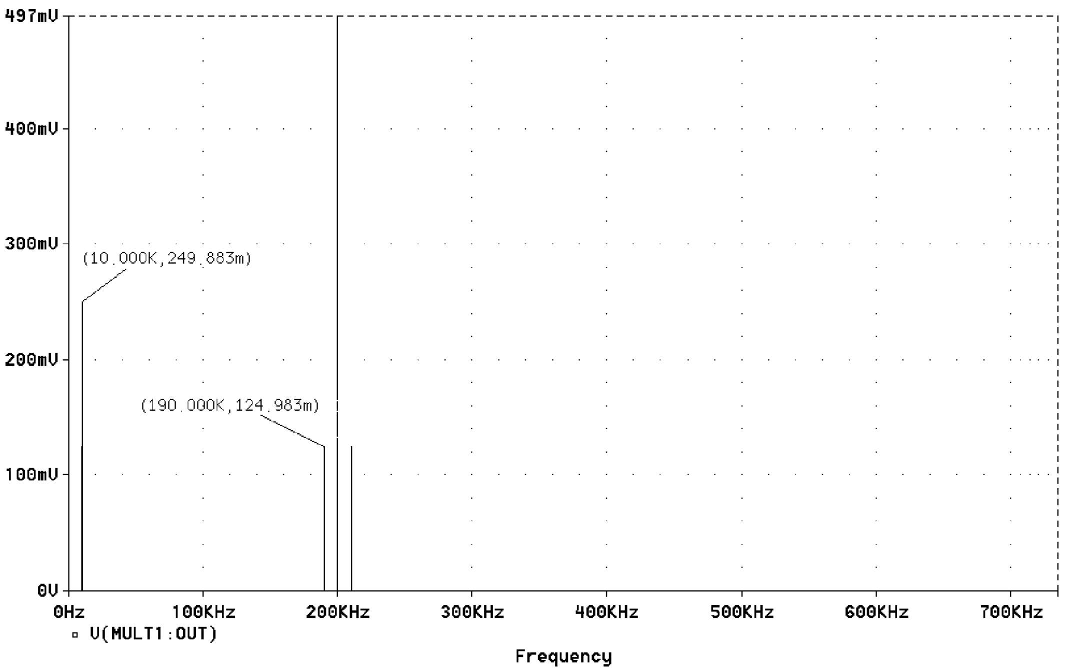


FIGURE 2.18: AM spectrum

(5) Investigate the DSBSC demodulator in Fig. 2.20 and use the PLL from the FM receiver in Chapter 6, section 6.7.
The DSBSC waveforms are shown in Fig. 2.21.

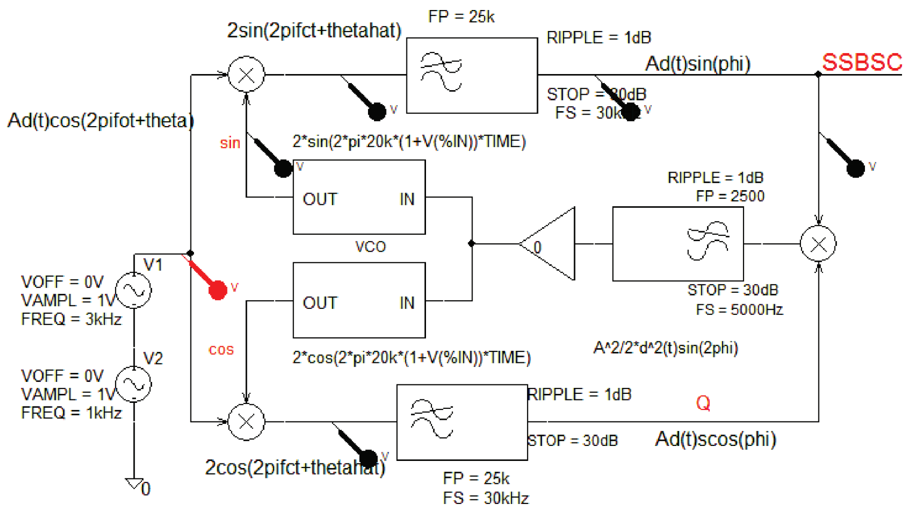


FIGURE 2.19: SSB-phase-shift method

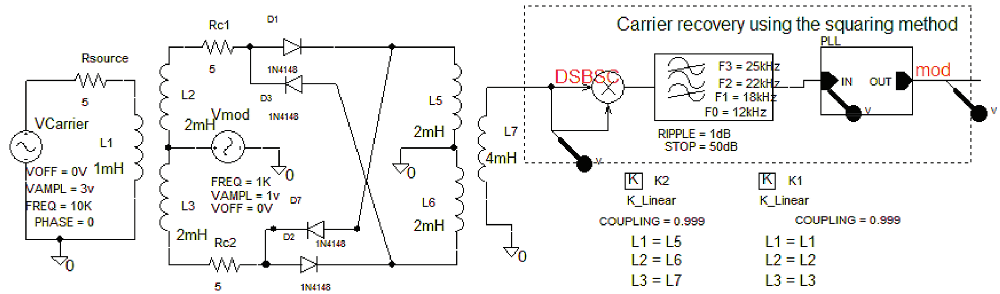


FIGURE 2.20: Carrier recovery from a DSBSC signal using a PLL

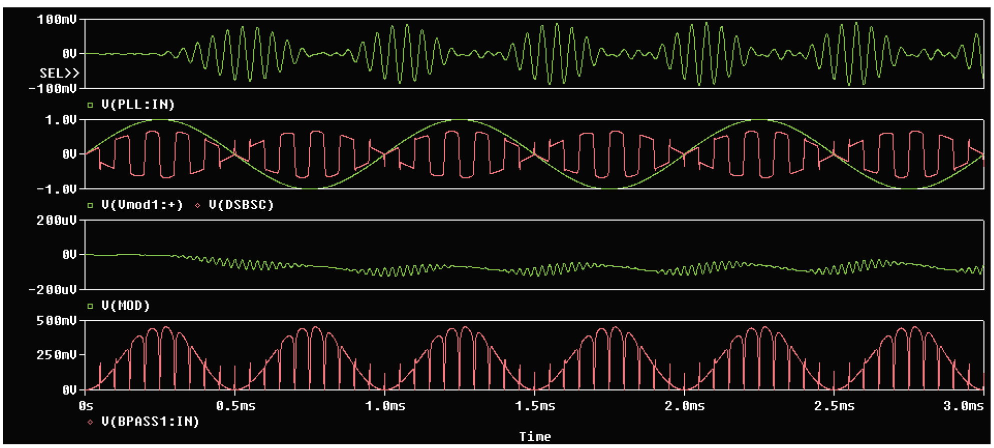


FIGURE 2.21: DSBSC demodulation

CHAPTER 3

System Stability, Nyquist Criterion

3.1 NYQUIST CRITERION

An oscillator is a circuit that produces self-sustained output signals but with no external input signal. Oscillators are used for testing electronic systems, or as the local oscillator for superhetrodyne action in radio receivers. The block diagram in Fig. 3.1 shows the three oscillator parts required to produce sustained oscillations. A portion, β of the output signal, is fed back to the input thus acting as an input signal.

The input signal is a fraction of the output signal expressed as

$$V_{\text{in}} = \beta V_{\text{out}}. \tag{3.1}$$

The output voltage is

$$V_{\text{out}} = AV_{\text{in}}. \tag{3.2}$$

Substitute (3.2) into (3.1)

$$(1 - \beta A)V_{\text{in}} = 0. \tag{3.3}$$

Since V_{in} is not zero, then the nontrivial solution for sustained oscillations is

$$(1 - \beta A) = 0 \Rightarrow \beta A = 1. \tag{3.4}$$

In general, since both A and β are complex, then the loop gain is complex $A\angle\phi\beta\angle\theta = \beta A\angle(\theta + \phi)$. Thus, the two conditions for the loop phase and gain to produce sustained oscillation are called the **Barkhausen criteria** as $\beta A\angle(\phi + \theta) = 1\angle(0^\circ \text{ or } 360^\circ)$ (Heinrich Barkhausen 1881—1956).

3.2 JFET COLPITTS OSCILLATOR

In this example, the FET input impedance is very high, so that the output impedance of the beta network will not load the amplifier input. The Colpitts oscillator in Fig. 3.2 uses a common

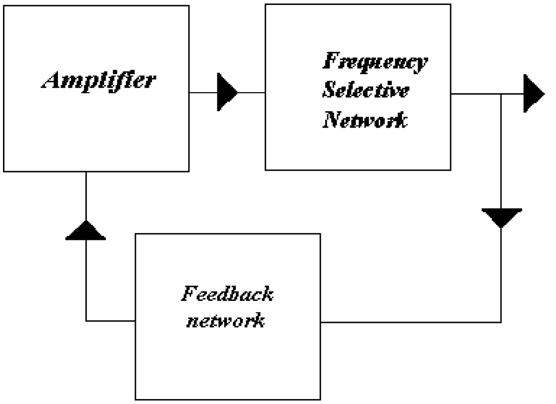


FIGURE 3.1: Closed-loop system

source JFET amplifier and a selective pi feedback network. The JFET has a drain–source resistance r_{ds} that is greater than the resistive load and so the gain A is

$$v_{out} = -g_m V_{gs} R_L \Rightarrow A_v = V_{out}/V_{gs} = -g_m R_L. \qquad (3.5)$$

The voltage gain for $g_m = 5.6$ ms and $R_L = 2.7$ kΩ is

$$A_v = -5.6 \times 10^{-3} \times 2.2 \times 10^{-3} = 12.2. \qquad (3.6)$$

Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 4ms, and **Maximum step size** = 0.1us. Press F11 to produce the signals shown in Fig. 3.2 where you should compare the results to the voltage gain calculation by hand for a 50 mV, 100 kHz input signal.

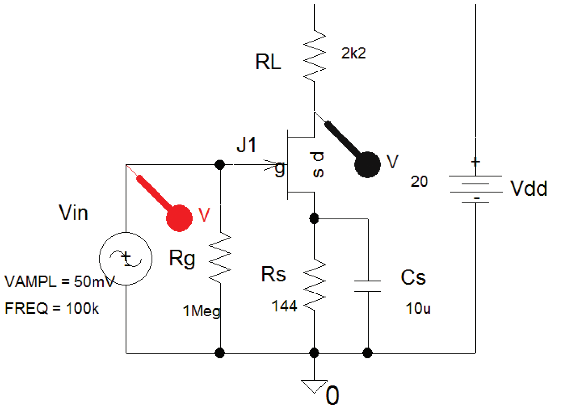


FIGURE 3.2: JFET amplifier stage

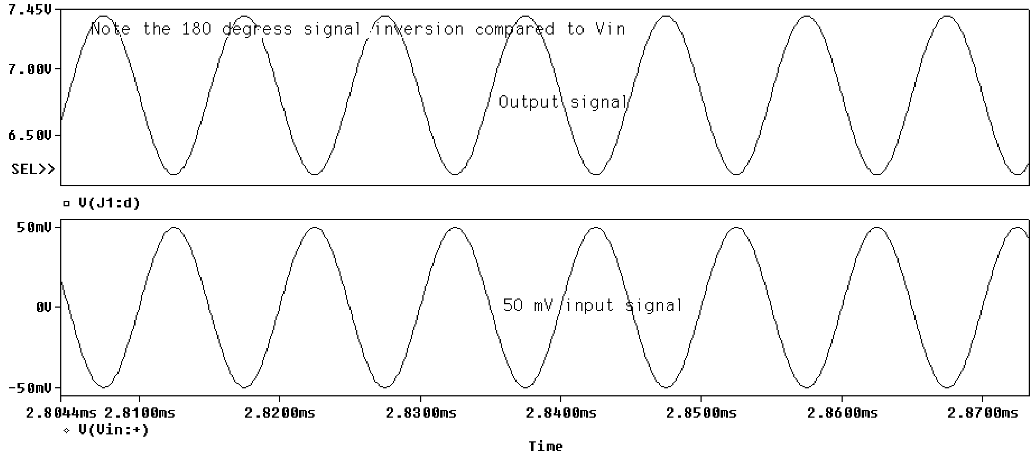


FIGURE 3.3: Signal waveforms at input and output

Confirm the measured gain A is 12.2. Note the 180° phase shift between input and output signals.

3.2.1 The Feedback Network

The beta transfer function is derived by applying the potential divider principle to the network in Fig. 3.4. The coil resistance is assumed much less than the coil reactance at the frequency of oscillation, so we may write

$$\beta \approx \frac{1/j\omega C_2}{j\omega L + 1/j\omega C_2} \times \frac{j\omega C_2}{j\omega C_2} = \frac{1}{j\omega L j\omega C_2 + 1} = \frac{1}{1-\omega^2 LC_2}. \tag{3.7}$$

The resonant frequency $\omega^2 = 1/LC_T$ has a total capacitance $C_T = C_1C_2/(C_1 + C_2)$. Substituting this into (3.7)

$$\beta = \frac{1}{1-\frac{1}{LC_T}LC_2} = \frac{1}{1-\frac{1}{C_1C_2/(C_1+C_2)}C_2} = \frac{1}{1-\frac{C_1+C_2}{C_1}} = \frac{C_1}{C_2}. \tag{3.8}$$

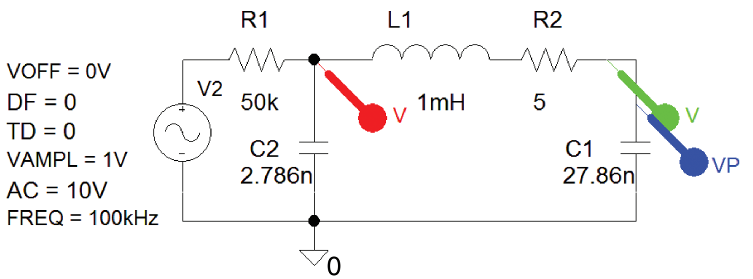


FIGURE 3.4: The beta feedback selective network

Thus, beta is a ratio of C_1 and C_2, but we need to find absolute capacitance values. Throughout the design, unrealistic component values were used so we may compare calculated and simulated values. (In practice, you would select the nearest preferred available component value.) In this design, we could have chosen unity loop gain but you run the risk of the loop gain being less than one because of component tolerances. Assume a loop gain $\beta A = 1.22$ (greater than one to ensure sustained oscillations), since the measured open-loop amplifier gain is 12.2 giving a handy value of 1/10. The capacitor ratio is

$$|\beta A| = 1.22 = \frac{C_1}{C_2} g_m R_L = \frac{C_1}{C_2} 12.2 \Rightarrow C_1 = 0.1 C_2. \tag{3.9}$$

The total capacitance required for sustained oscillations at 100 kHz and a 1-mH inductor is

$$C_T = \frac{1}{\omega^2 L} = \frac{1}{4\pi^2 \times 10^{10} \times 10^{-3}} = 2.533 \text{ nF}. \tag{3.10}$$

The total capacitance for capacitors in series is

$$C_T = \frac{C_1 C_2}{C_1 + C_2} = \frac{0.1 C_2 C_2}{0.1 C_2 + C_2} = 2.533 \text{ nF}. \tag{3.11}$$

Substituting for C_1 from (3.9) into (3.11) yields $C_2 = 27.86$ nF and $C_1 = 2.786$ nF. Draw the schematic in Fig. 3.4 and place voltage and phase markers on the output.

The JFET current source is simulated as a voltage source in series with a high resistance R_1. The beta phase response is obtained by placing a phase marker from the **PSpice/Markers/Advanced** menu. Measure the output voltage and phase at the resonant frequency, and hence show the beta network has a transfer function of 0.1 at the resonant frequency, and the phase is 180°. Set the Analysis tab to Analysis type: AC Sweep/Noise, AC Sweep Type to **Linear, Start Frequency** = 50k, **End Frequency** = 110k, **Total Points** = 1000, and press F11 to simulate and produce the amplitude and phase response as shown in Fig. 3.5.

Measure the phase of the beta network phase response at the resonant frequency f_0. Even though the output voltage is greater at other frequencies, the phase is only 180° at f_0. Measure the output voltage with respect to the input voltage and calculate the feedback factor beta. Examining the phase response shows that the network introduces 180° phase shift at the desired frequency of 100 kHz. Note that a large resistive load value $R_{o/c}$ is required by PSpice to fix any floating errors displayed. This error also occurs when you attempt to simulate with no ground symbol **AGND** connected. Fig. 3.6 shows how to test the amplifier and feedback network on an open-loop configuration, where the **VSIN** generator is set to 100 kHz.

Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 5ms, and **Maximum step size** = 500 μs. Press F11 and measure the open-loop gain verifying that it is 1.2.

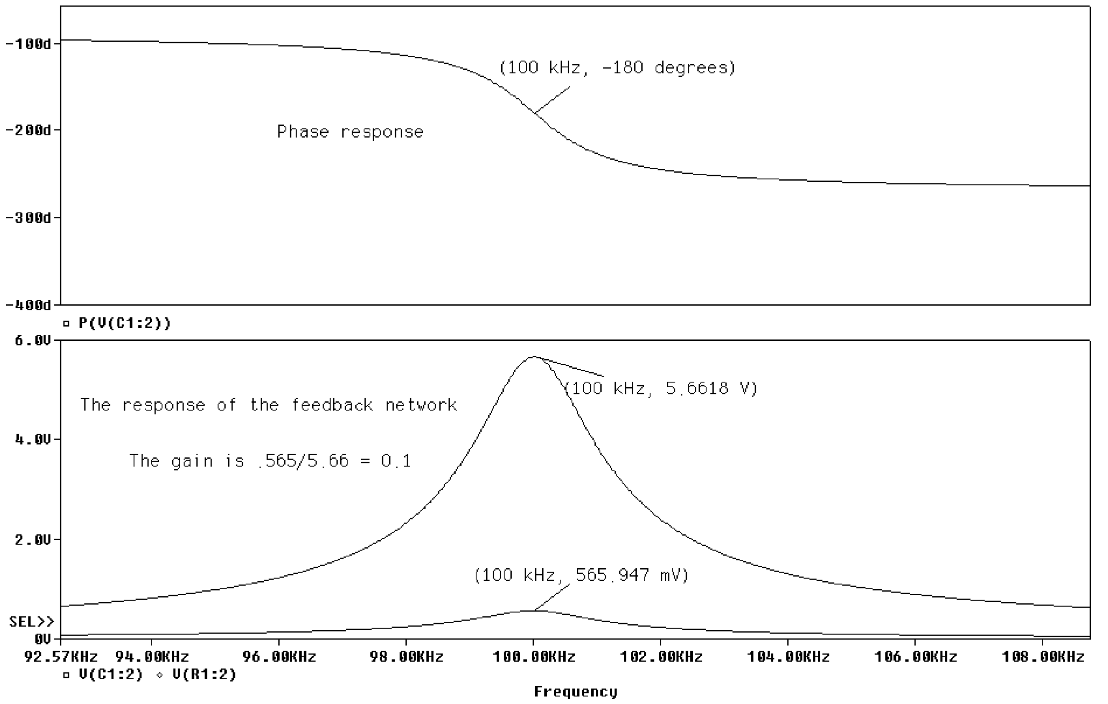


FIGURE 3.5: Frequency response of beta network

3.2.2 Closed-Loop Testing

Remove the AC voltage source and close the loop as shown in Fig. 3.7 by naming the input and output wire segments with the same name. The closed-loop gain does not change when the loop is closed since the input impedance of the JFET is very large, and hence will not load the output of the beta network. *Note*: Make sure the **Skip initial transient solution** is not ticked in **the Analysis Setup/transient menu.**

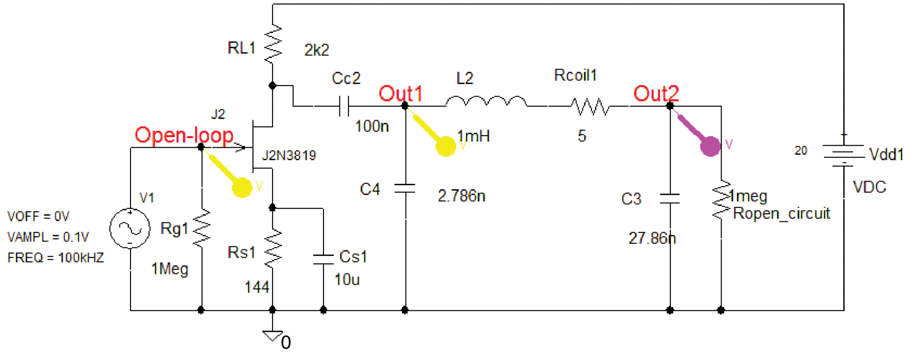


FIGURE 3.6: Open-loop test

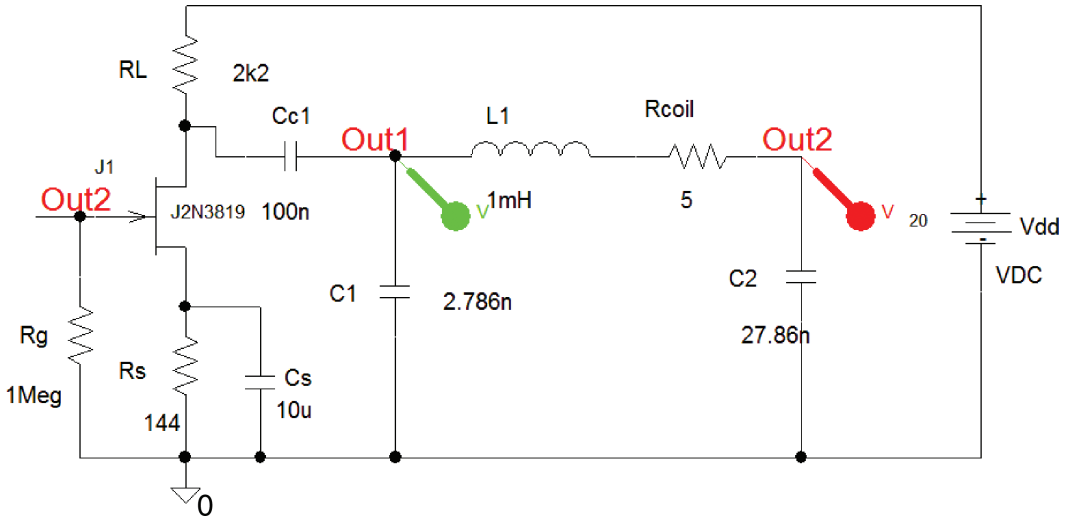


FIGURE 3.7: Closed-loop tests

3.2.3 The Output File

From the Analysis Setup/Transient/Fourier, set the Analysis tab to **Analysis type: Time Domain (Transient)**, **Run to time** = 5 ms, and **Maximum step size** = 500 μs. Press the **Output File Options** tick **Perform Fourier Analysis**, **Center frequency** = 100k, **Number of Harmonics** = 10, and **Output Variables:** = V(out1) V(out2). (You may enter more than one output name provided each name is separated by a space.) Select the net alias icon, type in the names out1 and out2, and press **F11** to simulate.

3.2.4 The Oscillator Output

If the design calculations are correct, then sustained oscillations should occur as shown. From the **FFT**, measure the frequency of the sustained oscillations as shown in Fig. 3.8.

Select **View/Output File** to display the harmonic analysis information at the end of the **.out** text file. Measure the harmonic distortion at each end of the inductor. What is responsible for the different distortion values? The Fourier spectral components should be displayed at the end of the output file as shown in Fig. 3.9. (The output results may also be viewed by selecting **PSpice/View Output File** menu.)

3.3 HARTLEY OSCILLATOR

The Hartley oscillator (Ralph Hartley 1888–1970) is analyzed is a similar manner to the Colpitts oscillator considered previously, where the feedback network transfer function was derived by applying the potential divider principle. Consider the design of a 100-kHz Hartley oscillator in Fig. 3.10, obtain an equation for the beta factor and show it is equal to L_2/L_1.

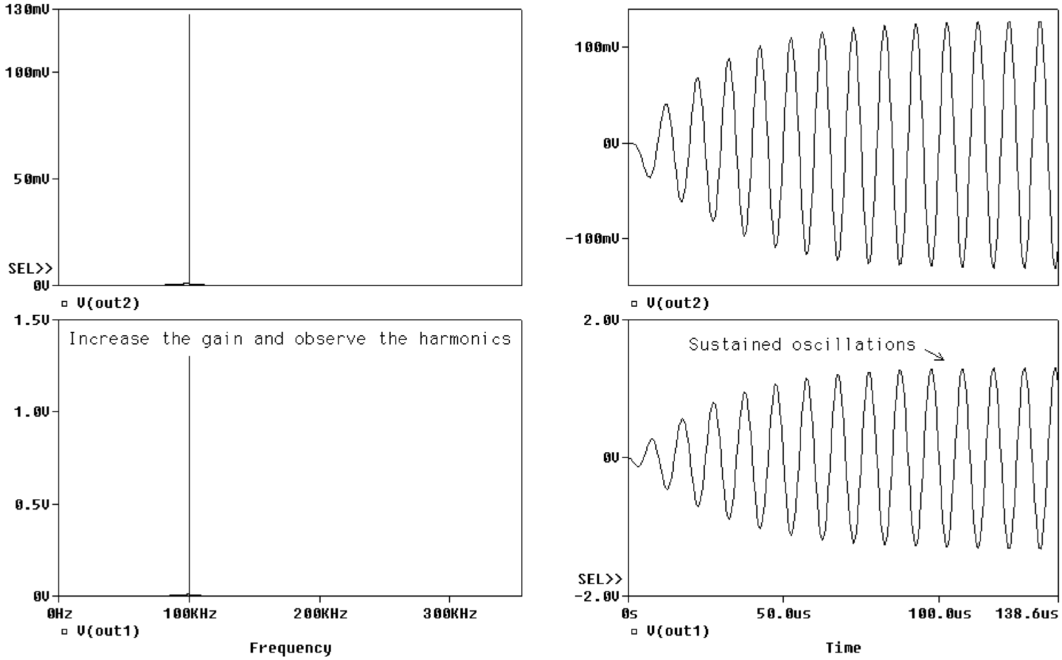
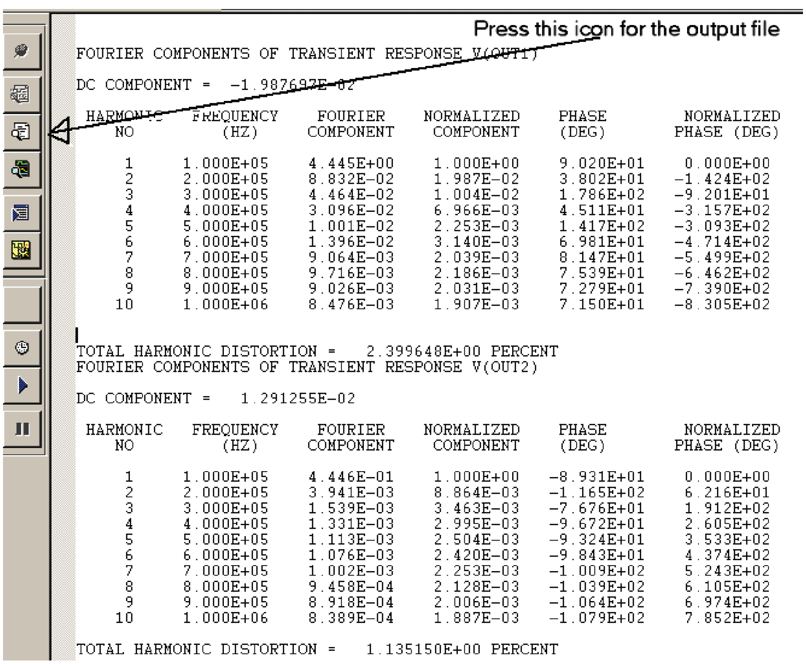


FIGURE 3.8: Spectral content at the output



FOURIER COMPONENTS OF TRANSIENT RESPONSE V(OUT1)

DC COMPONENT = -1.987697E-02

HARMONIC NO	FREQUENCY (HZ)	FOURIER COMPONENT	NORMALIZED COMPONENT	PHASE (DEG)	NORMALIZED PHASE (DEG)
1	1.000E+05	4.445E+00	1.000E+00	9.020E+01	0.000E+00
2	2.000E+05	8.832E-02	1.987E-02	3.802E+01	-1.424E+02
3	3.000E+05	4.464E-02	1.004E-02	1.786E+02	-9.201E+01
4	4.000E+05	3.096E-02	6.966E-03	4.511E+01	-3.157E+02
5	5.000E+05	1.001E-02	2.253E-03	1.417E+02	-3.093E+02
6	6.000E+05	1.396E-02	3.140E-03	6.981E+01	-4.714E+02
7	7.000E+05	9.064E-03	2.039E-03	8.147E+01	-5.499E+02
8	8.000E+05	9.716E-03	2.186E-03	7.539E+01	-6.462E+02
9	9.000E+05	9.026E-03	2.031E-03	7.279E+01	-7.390E+02
10	1.000E+06	8.476E-03	1.907E-03	7.150E+01	-8.305E+02

TOTAL HARMONIC DISTORTION = 2.399648E+00 PERCENT

FOURIER COMPONENTS OF TRANSIENT RESPONSE V(OUT2)

DC COMPONENT = 1.291255E-02

HARMONIC NO	FREQUENCY (HZ)	FOURIER COMPONENT	NORMALIZED COMPONENT	PHASE (DEG)	NORMALIZED PHASE (DEG)
1	1.000E+05	4.446E-01	1.000E+00	-8.931E+01	0.000E+00
2	2.000E+05	3.941E-03	8.864E-03	-1.165E+02	6.216E+01
3	3.000E+05	1.539E-03	3.463E-03	-7.676E+01	1.912E+02
4	4.000E+05	1.331E-03	2.995E-03	-9.672E+01	2.605E+02
5	5.000E+05	1.113E-03	2.504E-03	-9.324E+01	3.533E+02
6	6.000E+05	1.076E-03	2.420E-03	-9.843E+01	4.374E+02
7	7.000E+05	1.002E-03	2.253E-03	-1.009E+02	5.243E+02
8	8.000E+05	9.458E-04	2.128E-03	-1.039E+02	6.105E+02
9	9.000E+05	8.918E-04	2.006E-03	-1.064E+02	6.974E+02
10	1.000E+06	8.389E-04	1.887E-03	-1.079E+02	7.852E+02

TOTAL HARMONIC DISTORTION = 1.135150E+00 PERCENT

FIGURE 3.9: Fourier results in the output file

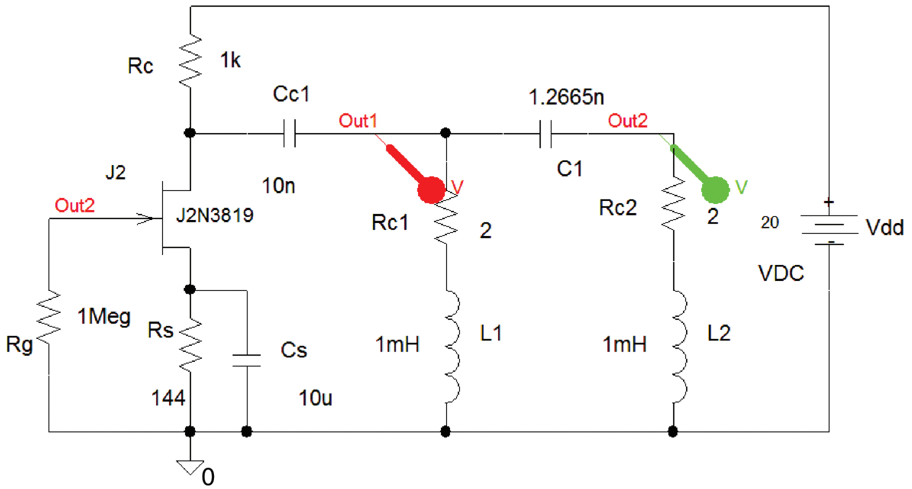


FIGURE 3.10: The closed-loop Hartley oscillator

This design will not oscillate at 100 kHz, even though the π network has a resonant frequency of 100 kHz. Investigate and find out why this is so (Hint: Is the L_2/L_1 ratio too large?). The Hartley oscillator output and spectrum are shown in Fig. 3.11.

Make $L_1 = 0.9$ mH, $L_2 = 100$ µH, and $C_1 = 2.53$ nF and simulate again.

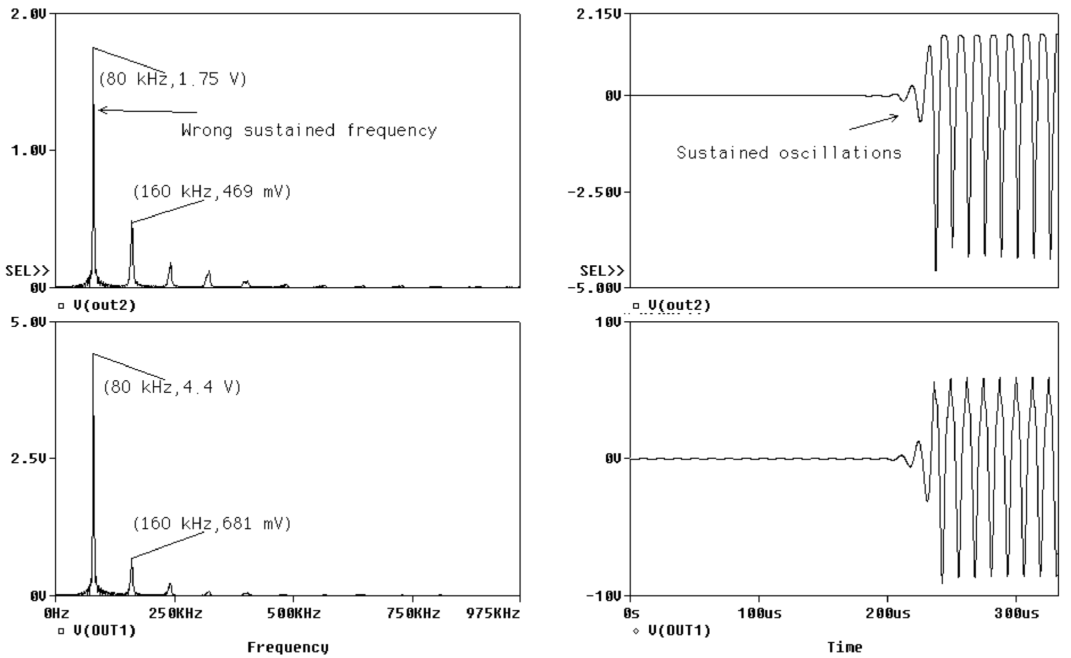


FIGURE 3.11: Hartley output

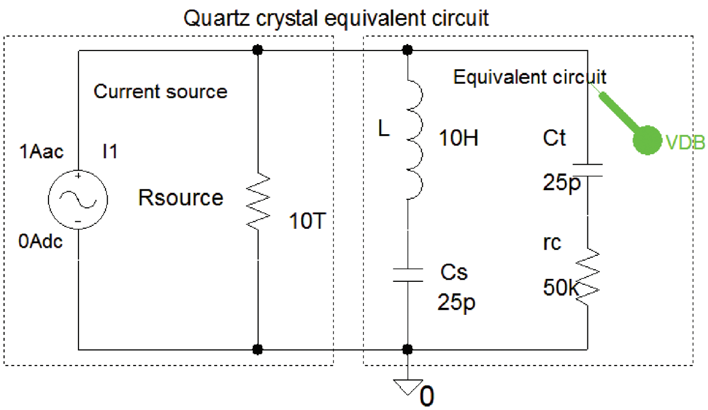


FIGURE 3.12: Equivalent circuit for a crystal

3.4 QUARTZ CRYSTAL AND EQUIVALENT CIRCUITS

A quartz crystal device is modeled as a series-parallel tuned circuit shown in Fig. 3.12. A nonideal current source is formed from a current source **IAC** in parallel with a high source resistance, Rsource, in order for the source not to load the crystal.

Fig. 3.13 is a test schematic for the customized crystal symbol.

3.4.1 Quartz Crystal Response

Set the Analysis tab to Analysis type: AC Sweep/Noise, AC Sweep Type to **Linear, Start Frequency** = 10k, **End Frequency** = 10Meg, **Total Points** = 1000. Press F11 to create the crystal frequency response in Fig. 3.14 which shows how the crystal can be operated in two modes: Series mode, or parallel mode. In the following section, the crystal is used in the series mode to produce a low resistance path at a crystal frequency.

3.4.2 CMOS Colpitts Crystal Oscillator

An expression for the closed loop gain for systems can be expressed as:

$$\mathrm{CLTF} = \frac{A}{1 + \beta A} \tag{3.12}$$

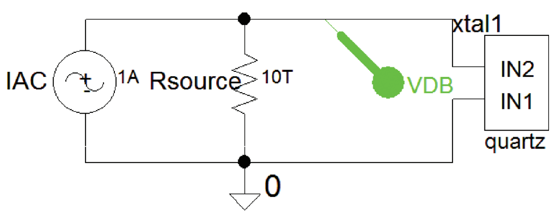


FIGURE 3.13: Testing the quartz device

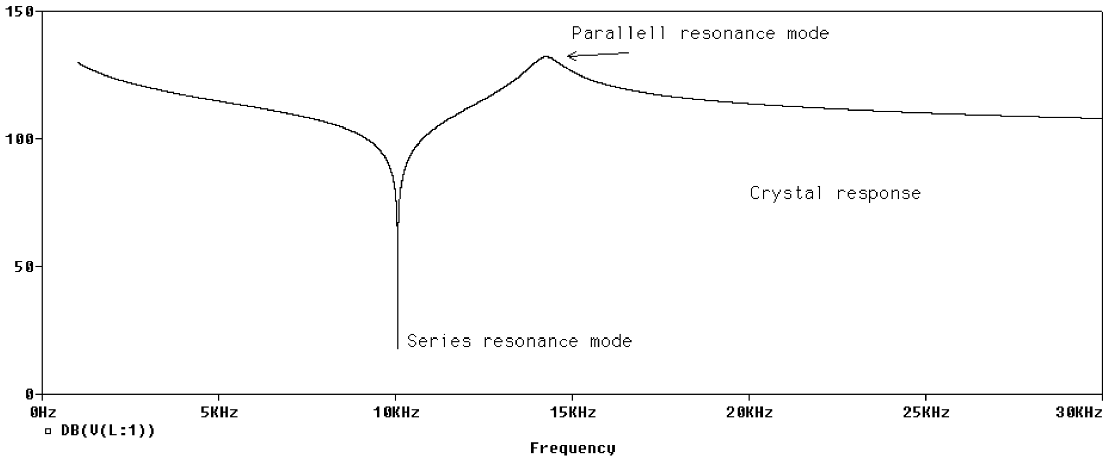


FIGURE 3.14: The frequency response for the crystal

Here A is the open loop gain and beta is the feedback factor. The magical factor, is used in negative feedback systems to produce all sorts of desirable parameter changes such as increased input impedance, bandwidth changes, etc. However, the next few exercises we use positive feedback to deliberately produce marginal stability in the form of sustained oscillations.

Fig. 3.15 shows a Colpitts oscillator using a crystal operated in the series mode to produce sustained oscillations.

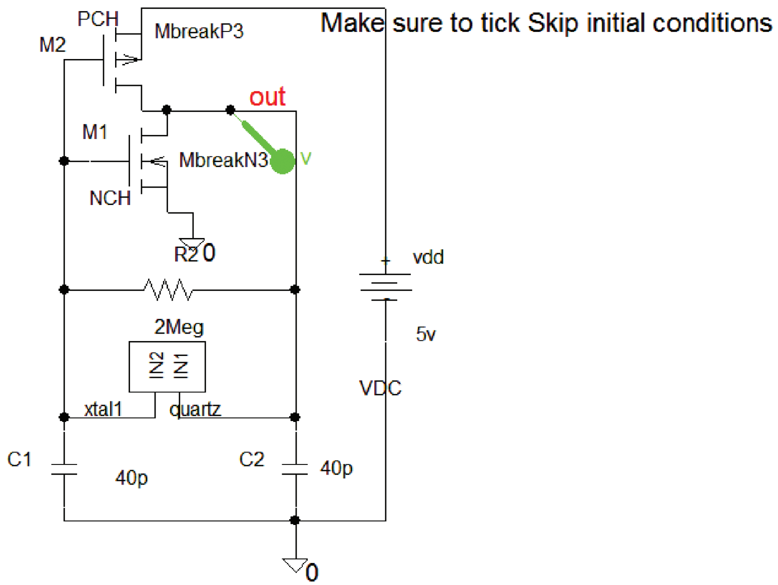


FIGURE 3.15: CMOS oscillator using a quartz crystal

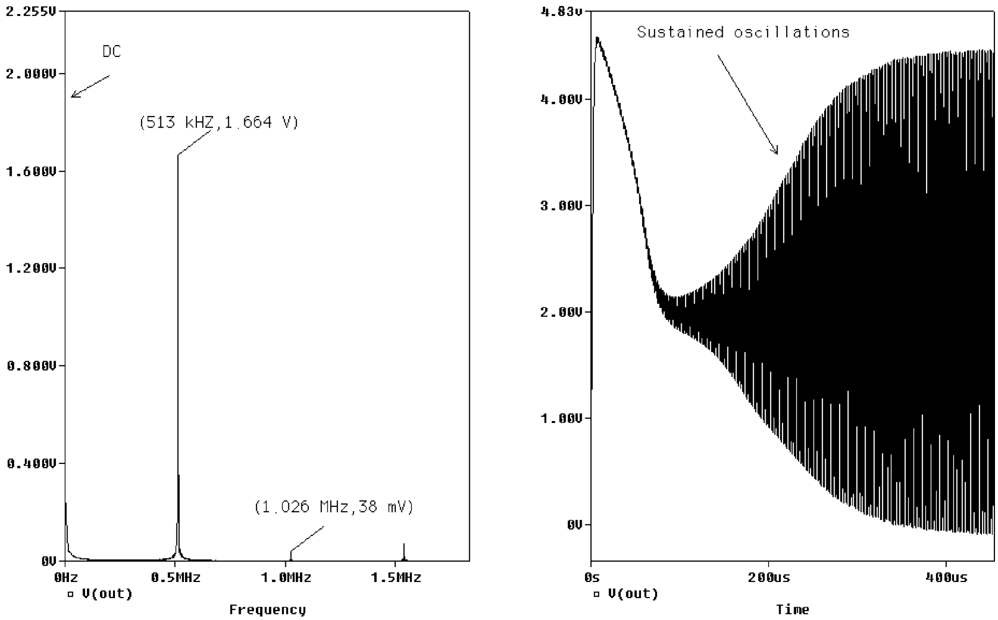


FIGURE 3.16: CMOS oscillator output

Modify the contents of the crystal block and set the inductance to 0.05 H. In the **Analysis Setup/Transient box**, set the parameters to **Print** = 0.1 ms, **Run to time** = 1 ms, **Maximum step size** = 0.1 µs. *Note*: Make sure to place a check mark on **Skip initial transient solution,** otherwise you will get an empty Probe screen. From Probe, press **alt PP** to make a separate plot. Copy the same output variable and paste in the new plot. From the **Plot** menu select **Unsynchronize Plot** to display signals in the time and frequency domain plots as in Fig. 3.16.

3.5 PHASE-SHIFT OSCILLATOR

This oscillator uses a feedback network consisting of three *CR* networks giving a total phase shift of 180° and an inverting opamp to provide the necessary gain and the other 180° of phase shift. We test the phase-shift oscillator in Fig. 3.17 by opening the closed loop and applying a 1-V **VSIN** generator part. (Remember to fill in the other **VSIN** parameters such as **VOFF** = 0.) Rename the input wire as **vsine** in order to connect the **VSIN** part to the input. The three *CR* circuits contribute the other 180° of phase shift necessary to satisfy the Barkhausen conditions for sustained oscillations. (The loss of the phase shifter at the desired frequency of oscillation is an attenuation factor of 1/29.) The frequency at which the phase shift is exactly 180° is given by the following expression:

$$f_0 = \frac{1}{2\pi RC\sqrt{6}}\ \text{Hz} = \frac{1}{2\pi 5 \times 10^3 0.0816 \times 10^{-6}\sqrt{6}} = 159\ \text{Hz}. \qquad (3.13)$$

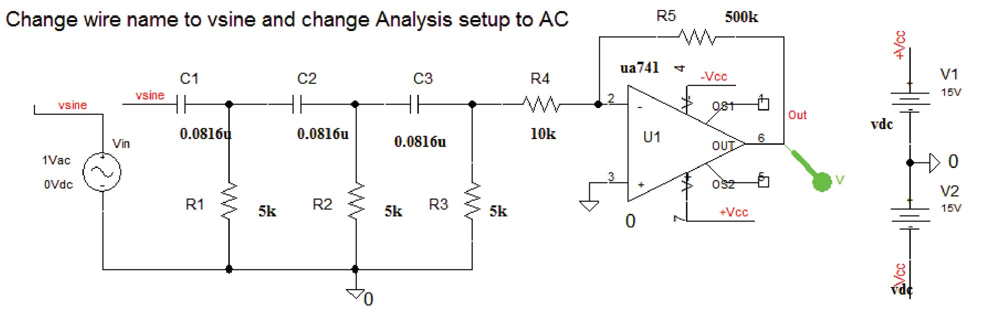


FIGURE 3.17: Phase-shift oscillator on open loop

The attenuation at this frequency is

$$\beta = \frac{1}{29}. \tag{3.14}$$

The operational amplifier has a gain of 50, or 20 log50 $=$ 33 dB, with a 180° phase shift. Test the open-loop response by naming the wire segment to vsine. From the **AC analysis Setup** set the **Total Points:** to 100001, the **Start Frequency** to 0.01, **End Frequency** to 20k, and **Logarithmic.** Place dB and phase markers should result in Fig. 3.18. (Separate the two variables using **alt PP.**)

3.5.1 Gain and Phase Margins

Some means of assessing the stability of closed-looped systems must be considered. One technique defines the **Gain Margin** (**GM**) and is the value of the open-loop gain βA in dB at a frequency where the phase of the output is 180°. The **Phase Margin** (**PM**) is an angle of 180° minus the angle where the open-loop is 0 dB.

Close the loop by renaming the input and output wire segments names to **vout** to connect the output back to the input. Simulate to test for sustained oscillations. Measure the harmonic distortion at the output by entering values in the **Analysis Setup/Transient/Fourier** and ticking off the number of harmonics required in the output file.

3.5.2 Nyquist Plot

The Nyquist plot (Harry Nyquist 1894–1976) is an alternative systems stability test to the Gain and Phase margins techniques discussed in the previous section. After simulation, change the x-axis to the real part of the transfer function. The imaginary part of the transfer function is plotted on the y-axis. We will demonstrate the Nyquist plotting technique using a first-order low-pass *CR* filter. Create/Load, the LPF schematic investigated in book2 (PSpice for Filters and Transmission lines), and set the Analysis AC parameters (1 Hz to 1 MHz, and **Total Points:** = 1000). Simulate, and from Probe, select **Trace/Add trace** and plot the imaginary

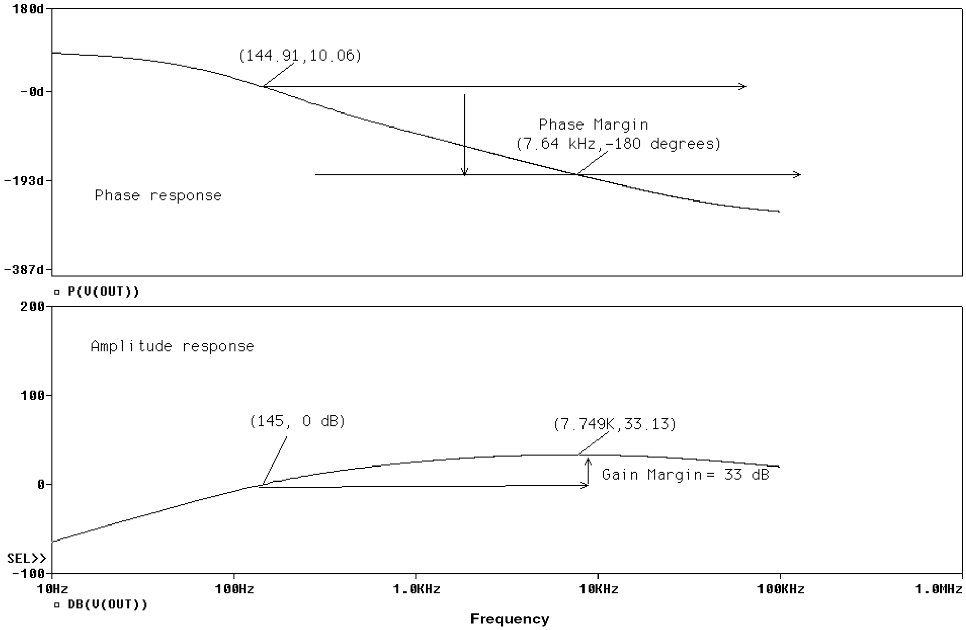


FIGURE 3.18: Gain and phase margins

part of the output by selecting **IMG()** from the right trace list and inserting the output voltage in the parenthesis as **IMG(vout)**. **DLclick** an x-axis number and change the **X-Axis Settings** from **Log** to **Linear**. The easier way of doing this is to select **Plot Windows Templates** from the top right hand menu after pressing the insert button and select **Nyquist()** from the list. Fill in the variable **V(out)** from the list of variable in the left pane.

Select the **Axis Variable** and in the **Trace Expression** box enter **R(vout)**. This changes the x-axis to the real part of the output voltage **R(vout)**. The Nyquist plot for a first-order low-pass *CR* filter is a semicircle starting at the TF real value equal to one (imaginary part = 0) and ending at the origin. The transfer function magnitude is determined from the length of the vector from the origin to the circle. In Fig. 3.19, the vector length at the cut-off frequency is 0.707, i.e., $\sqrt{\text{realpart}^2 + \text{imaginary part}^2}$. The phase at a particular frequency is the angle the vector makes with the real axis. The phase angle at the cut-off frequency is −45° as shown.

3.5.3 Nyquist Diagram and Oscillators

The Nyquist plot assesses the stability of systems and here we apply it to the phase-shift oscillator (a marginally stable closed-loop system). Sustained oscillations are produced when the loop gain $|\beta A\angle\theta| \geq 1\angle 0$. For stability in nonoscillatory circuits, the loop-gain should be less than one, i.e.,$|\beta A\angle\theta| < 1\angle 0$ and is displayed in the Nyquist plot when the *TF* does not

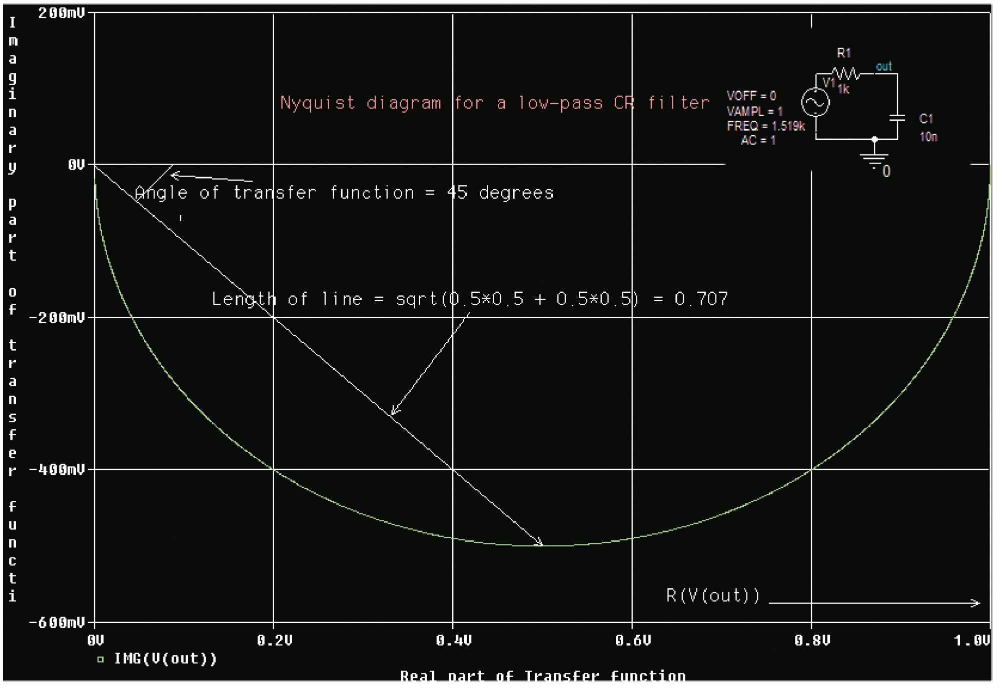


FIGURE 3.19: Nyquist plot for a first-order CR filter

encompassing the "−1" on the real axis. Simulate and observe sustained oscillations at $f = 161$ Hz in Fig. 3.20.

Change the input bubble back to **vsine** and set the **Analysis Setup** parameters: **Start Freq** = 1, **End Frequency** = 1 Meg and **Total Points:** = 1000. Simulate, and from Probe, select **Trace/Add.** From the **Plot Window Template** at the top right select **Nyquist Plot(1)** and replace the 1 with **V(Out)** from the left list. This plots the imaginary part **IMG(V(Out))** of the output (the vertical, or y-axis) and changes the **X-Axis Settings** to the real part of the output voltage **R(vout)**. The open-loop oscillator Nyquist plot illustrated in Fig. 3.21 shows the −1 point is enclosed so the system should produce sustained oscillations.

3.6 EXERCISES

(1) The transfer function for the equal-value component Wien network in Fig. 3.22 is

$$\begin{aligned}\frac{V_2}{V_1} &= \frac{R/(1+j\omega CR)}{R/(1+j\omega CR)+R+1/j\omega C} \\ &= \frac{R}{R+R(1+j\omega CR)+(1/j\omega C)(1+j\omega CR)} \\ &= \frac{R}{3R+j\omega CR^2-j/\omega C}.\end{aligned} \tag{3.15}$$

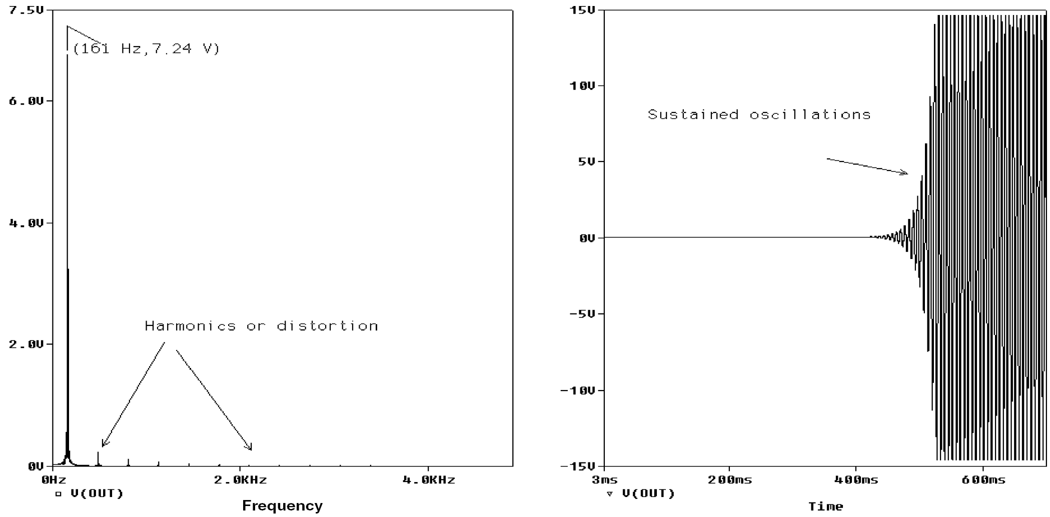


FIGURE 3.20: Sustained oscillations for the phase-shift oscillator

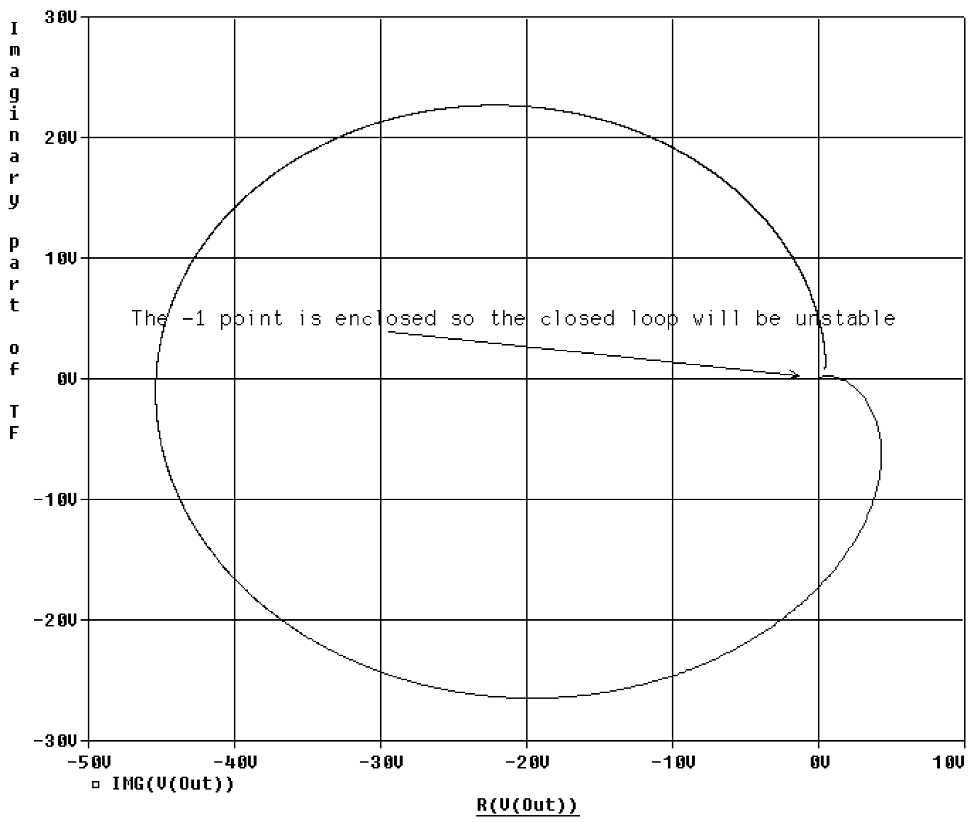


FIGURE 3.21: Nyquist plot for the open-loop phase-shift oscillator

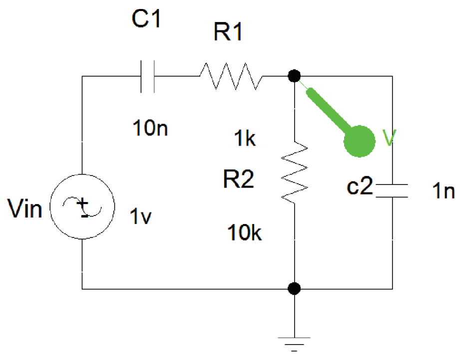


FIGURE 3.22: Wien network

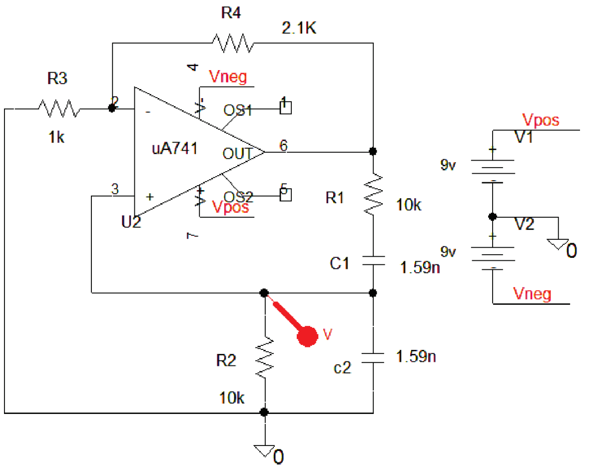


FIGURE 3.23: Wien oscillator

The Wien network phase is zero when the imaginary terms in (3.14) are zero. Equating the imaginary terms to zero gives the condition for oscillation as $\omega_o = 1/CR$. Dividing (3.14) above and below by R yields

$$\mathrm{TF} = \frac{1}{3 + j(\omega CR - 1/\omega CR)} = \frac{1}{3 + j(\omega/\omega_o - \omega_o/\omega)}. \tag{3.16}$$

(2) Plot the frequency response and measure the maximum amplitude at $\omega_0 = 1/CR$. Design a 10-kHz oscillator using the Wien network and opamp. Determine the value for all components (the gain $A_v = 1 + R_3/R_4 = 3$) in Fig. 3.24 to satisfy the Barkhausen criteria for the closed-loop to sustain oscillation. Open the feedback loop and apply a 1-V **VAC** part to the noninverting input and observe the signals across R_2. Is the output equal to 3 V? Produce a Nyquist plot by changing the x-axis, or select the inbuilt macro.

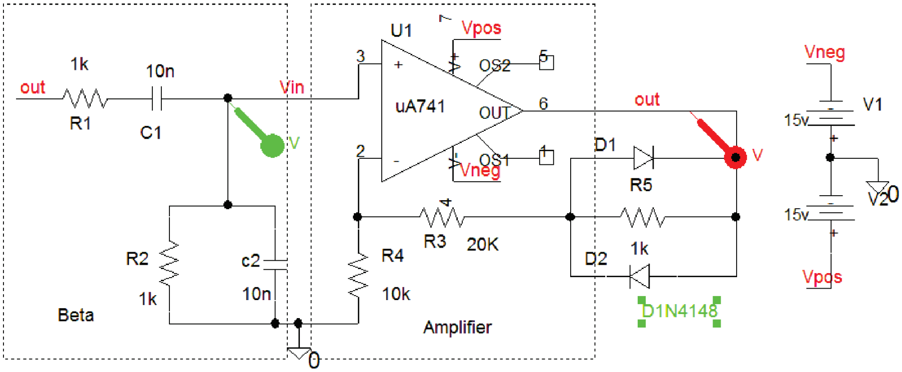


FIGURE 3.24: Wien oscillator with amplitude limiting

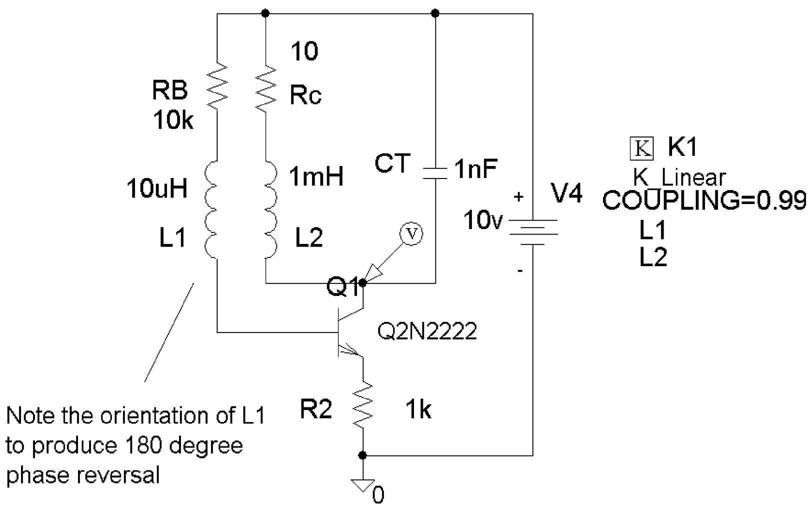


FIGURE 3.25: Tuned-collector oscillator

Does the circuit produce sustained oscillations at the required frequency of 10 kHz?

(3) Place a pair of parallel back-to-back diodes, anode to cathode, in series with R_3 but in parallel with R_5 as shown in Fig. 3.24. Investigate the effect on signal amplitude limiting by changing the loop gain.

(4) Investigate the tuned-collector oscillator shown in Fig. 3.25. This could be used as a local oscillator with the tuning capacitance C_T coupled to input tuning capacitance (Hint: **Param** part). The coil L_1 is rotated in the orientation shown in order to satisfy the phase Barkhausen criterion. (Total phase shift around the loop is zero or 360°.) The common emitter amp provides 180° and the coil reversal the other 180°.

CHAPTER 4

Superhetrodyne Amplitude Modulation Receivers

4.1 NONLINEAR MIXING

A modulator is required to relocate a signal to a different part of the frequency spectrum. This process is also called nonlinear mixing or heterodyning where the signal is combined with a local oscillator or carrier. There are two basic types of mixers: **Multiplicative** and **additive**. The multiplicative mixer uses a multigate JFET device to multiply the two signals to be mixed. The additive type uses the nonlinear properties of an active device, such as a suitably biased BJT or FET, to produce mixing components. A radio frequency (RF) signal applied to a superhetrodyne receiver is frequency shifted downward to an intermediate frequency (IF). A common source connected FET produces nonlinear mixing, and the drain current is related to the gate source voltage as:

$$
\begin{aligned}
I_{ds} &= I_{dss}\left(1-\frac{V_{gs}}{V_{po}}\right)^2 = 12mA\left(1-\frac{V_{gs}}{3}\right)^2 = 0.012(1-0.6V_{gs}+0.09V_{gs}^2) \\
&= 0.012 - 0.0072V_{gs} + 0.001V_{gs}^2 + \cdots . \qquad (4.1)
\end{aligned}
$$

For the JFET J2n3819 device, the pinch-off voltage, V_{po}, is −3 V, and the saturation drain–source current is $I_{dss} = 12$ mA. The gate–source voltage has two voltages applied to the gate–source terminals. What we are trying to do is to mix the “outside” RF signal with a “local” RF signal. The local signal has a much higher frequency and amplitude and attempts to shift the incoming RF signal to a different part of the RF spectrum. Thus, the terminal gate–source voltage is $V_{gs} = V_{rf}\cos\omega_m t + V_{lo}\cos\omega_c t$. The drain–source voltage is defined as

$$
\begin{aligned}
I_{ds} = 0.012 &- 0.0072(V_{rf}\cos\omega_{rf} + V_{lo}\cos\omega_{lo}) \\
&+ 0.0011(V_{rf}\cos\omega_{rf} + V_{lo}\cos\omega_{lo})^2. \qquad (4.2)
\end{aligned}
$$

The squared term results in amplitude modulation and this is shown by expanding into sum and difference frequencies, i.e., cosAcosB = 0.5cos(A − B) + cos(A + B). We will shortly see how a parallel-tuned circuit, resonant at the intermediate frequency (IF) of 455 kHz, filters out all undesirable frequencies produced in the mixing process, but passes the IF.

4.2 FET NONLINEAR MIXER CIRCUIT

The circuit in Fig. 4.1 shows a FET amplifier whose voltage gain is

$$V_{out} = -g_m V_{gs} R_5 \Rightarrow V_{out}/V_{gs} = A_v = -g_m R_5. \qquad (4.3)$$

The drain–source resistance does not load R5 since it is much larger [ref: 1 Appendix A].

RF and local oscillator signals are applied as shown using a **VSIN** generators, with parameters as shown.

4.2.1 Mixer Output

Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 10 ms, and **Maximum step size** = 100 µs, press F11 to show the output in Fig. 4.2 is distorted because of the extra frequencies generated in the mixing process. Press **alt PP** for a new plot and copy the variable into it. From **Probe/Plot** select **unsynchronize** and press the **FFT** to display the mixing frequency components. We can extract the 455 kHz difference frequency in the mixed signal using a resonant amplifier circuit resonant at this frequency and rejects other mixing frequencies.

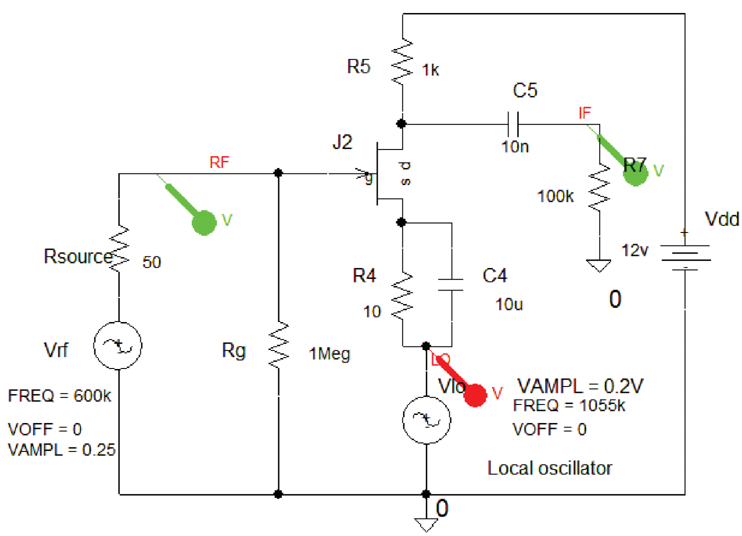


FIGURE 4.1: Mixing circuit

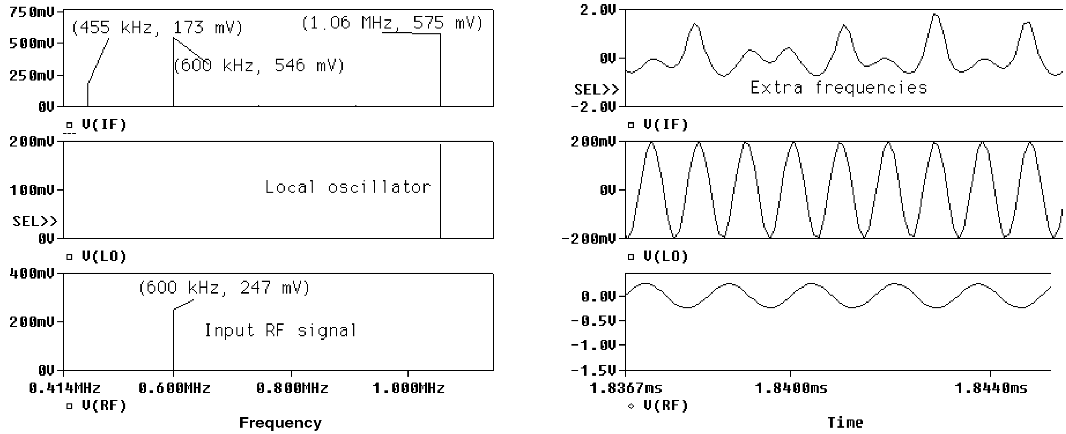


FIGURE 4.2: The three RF signals

4.2.2 Superhetrodyning

The superhetrodyne principle involves modulating/mixing an external RF source with the receiver local oscillator source. We simulate superhetrodyning in PSpice using a **PARAM** part to mimic ganging of the input RF tuned capacitance and the local oscillator capacitance. Consider the tuned IF mixer schematic in Fig. 4.3.

Ganging refers to the older technique of mounting two capacitors on the one spindle, thus ensuring the two capacitors change in step together (although it is no longer done like this but uses varactor diodes instead). A **VSIN** part represents the input (the desired RF frequency) and has a frequency {**frf**} that is referenced by the **PARAM** part. The local oscillator achieves ganging using a **Param** part to define the part name and value. From the **Analysis Setup** menu, tick **Transient** and **Parametric.** Set the **Swept Var Type** to **Global parameter**, and enter **Name** = frf with **Start Value** = 500k, **End Value** = 900k, and **Increment** = 200k. **Sweep Type** is **Linear**. This sweeps the frequency in step with the local oscillator, i.e., {**frf + 455k**}. If the input signal f_{rf} is swept from 500 kHz up to 900 kHz in steps of 200 kHz, then the local oscillator will be swept from 955 kHz to 1355 kHz in steps of 200 kHz. The frequency spectrum in Fig. 4.4 demonstrates superhetrodyne action showing how the RF signal is always relocated to 455 kHz irrespective of what the input frequency is.

4.2.3 Image Frequency

A major disadvantage of superhetrodyning is that it introduces an **image frequency** that is always present no matter what RF signal is selected. In a superhetrodyne receiver, the local oscillator (*LO*) frequency is at a higher frequency than the desired *RF* frequency because of limited capacitor ratio values available. An image frequency is greater than the *LO* frequency by an amount equal to the intermediate frequency. This will mix to give another frequency

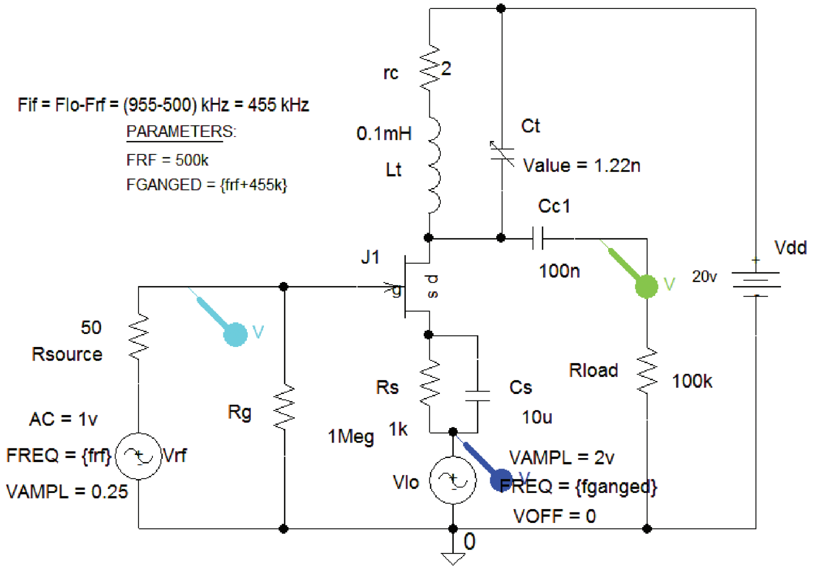


FIGURE 4.3: Tuned IF mixer

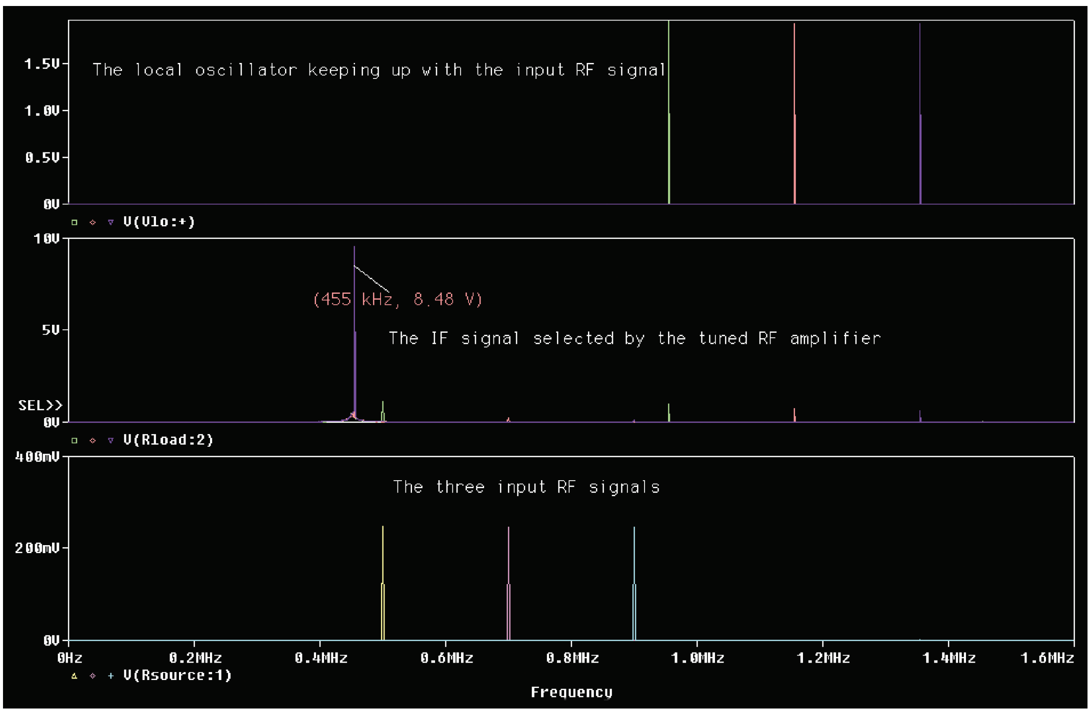


FIGURE 4.4: Spectrum of IF mixer

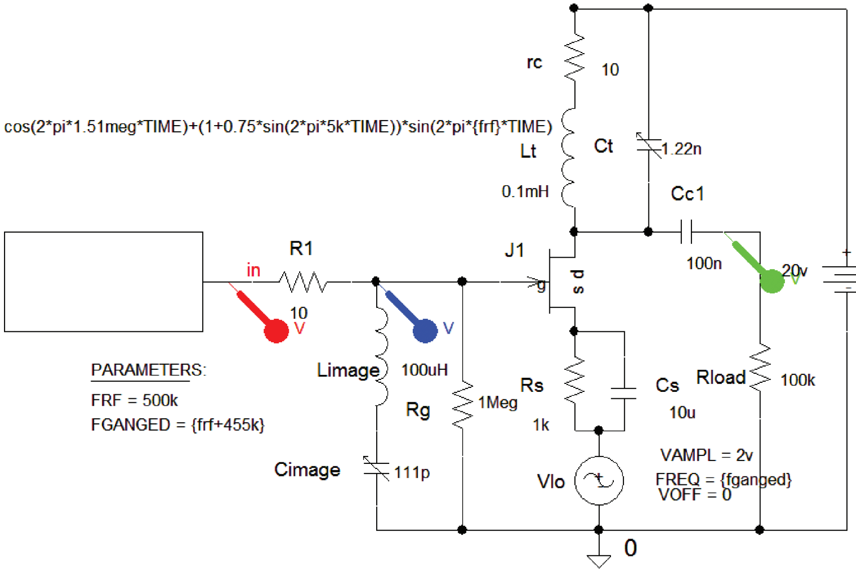


FIGURE 4.5: Image rejection

coincident at the IF at the output of the mixer. The image signal must therefore be attenuated at the input tuned circuit before it reaches the mixer. The image frequency is defined as

$$f_{\text{image}} = f_{\text{LO}} + f_{IF}. \tag{4.4}$$

But the local oscillator is defined as

$$f_{\text{LO}} = f_{RF} + f_{IF}. \tag{4.5}$$

Hence, the image frequency is

$$f_{\text{image}} = f_{RF} + 2f_{IF}. \tag{4.6}$$

Any image frequency signal, after down conversion to IF, will appear at same RF location as the desired signal, and will be heard as an interfering signal so image frequencies must be rejected before they reach the mixing stage. To test image frequency rejections, we add an extra cosine term to the input signal as shown in Fig. 4.5. The image signal is cos(2*pi*151000*TIME), where 1510000 is the desired RF plus twice the IF (600 kHz + 2*455 kHz = 1.51 MHz). The image channel rejection ratio (ICRR) is defined as

$$ICRR = \sqrt{1 + Q^2\,\rho^2} \tag{4.7}$$

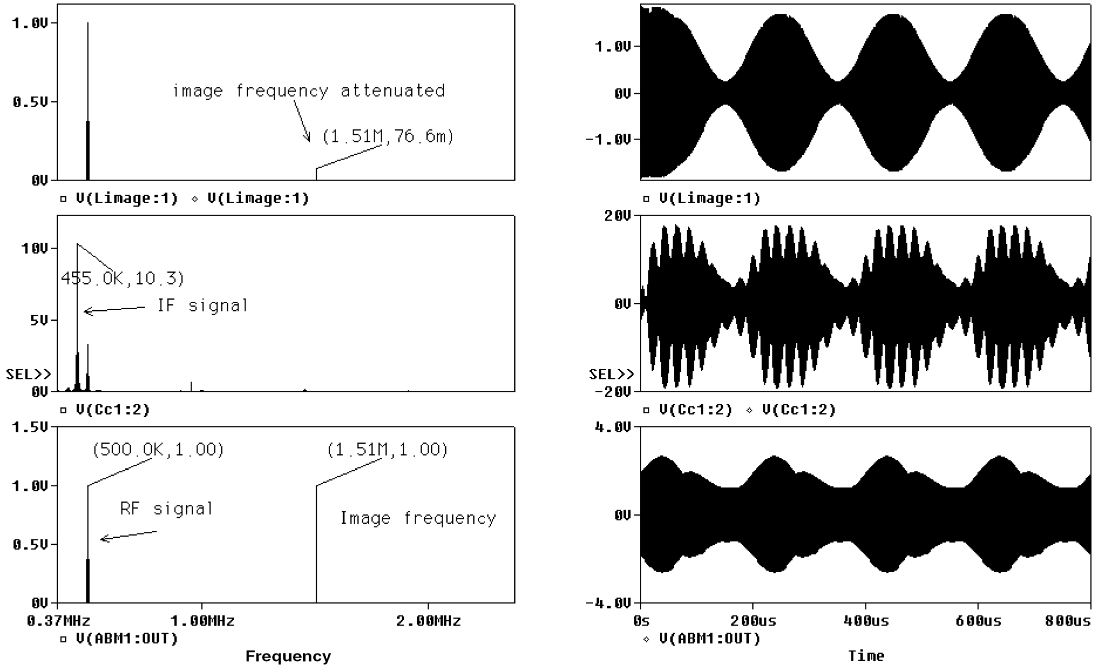


FIGURE 4.6: Image frequency rejection

where $\rho = f_{\text{image}}/f_{RF} - f_{RF}/f_{\text{image}}$ and Q is the Q-factor of the input stage. Image frequency rejection must take place before the mixing stage and here a bandstop tuned input circuit is used to achieve image rejection [ref: 1 Appendix A].

4.2.4 Superhetrodying and Image Frequencies

Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 10 ms, and **Maximum step size** = 10 µs, press F11 to produce Fig. 4.6 which shows the image signal is attenuated.

The three overlapping displays were separated using **alt PP** to give three new plots, **ctrl X** to delete a selected variable, followed by **ctrl V** to paste it into a new plot area. Select **Window/New/Tile Vertical** for a new window and copy the three variables across. Press the **FFT** icon to display the signals in the frequency domain.

4.3 AM SUPERHETRODYNE RECEIVER

Mixing two frequencies to produce a third frequency, the difference of the two frequencies, is called heterodyning. Amplitude modulation is also a heterodyne process, where the modulating information signal is mixed with the carrier signal to produce sidebands on either side of the

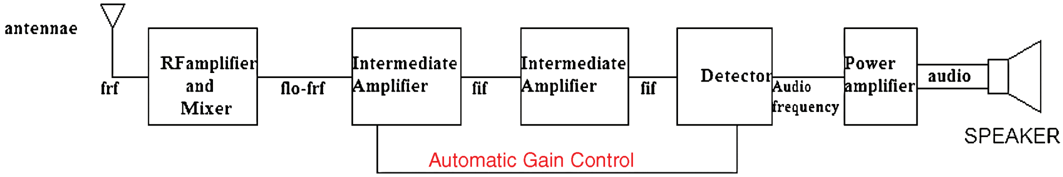


FIGURE 4.7: AM superhetrodyne receiver

carrier. The superhetrodyne principle was created by an American engineer Edwin Armstrong (Edwin Armstrong 1890–1954) and repositions a desired RF signal to a new frequency location called the intermediate frequency, f_{IF}, defined as

$$f_{IF} = f_{\mathrm{LO}} - f_{RF}. \tag{4.8}$$

The desired RF signal is mixed with a locally generated carrier in the receiver so that when you tune to any RF station it is relocated to a **fixed** lower frequency, the intermediate frequency, or IF. Fig. 4.7 shows the main processes in the receiver.

This overcomes one of the major problems associated with the fixed tuned radio frequency (*TRF*) receivers, where each amplifier had to be tuned by the operator (yes, up to four dials on very old radios!). The desired *AM* signal is now at a fixed intermediate frequency where the IF amplifiers are aligned at the manufacturing stage to a fixed value of 455 kHz and does not require any further tuning by the operator. Thus, a number of prealigned amplifiers give the required selectivity and sensitivity. The local oscillator tuning capacitor, *Ct*, is ganged to the tuning capacitor in the *RF* input tuning section so that varying the tuning capacitor in the *RF* stage also changes the LO tuning capacitor simultaneously. The two capacitors do not track each other perfectly but we can achieve acceptable tracking using small capacitors, called trimming and padding capacitors. This project is reasonably complex so we use the hierarchical method by breaking the project into manageable blocks. Fig. 4.8 shows an AM with adjacent channel signals being injected for testing the ability of the receiver to reject adjacent signals. However, set the adjacent channel signal to zero and test the functionality of each stage for a simple modulating signal first.

The receiver schematic comprises a major block and contains a further five blocks as shown in Fig. 4.9. The five blocks are a mixer stage, an IF stage, a detector, a preamplifier, and finally, a power amplifier.

4.3.1 The Input/Mixer Stage

One of the main problems with cascading stages is the problem of loading. The second stage input impedance is quite low and thus will load the first stage selective circuit. We get over this here using transformer coupling with the coupling selected for impedance matching. The first

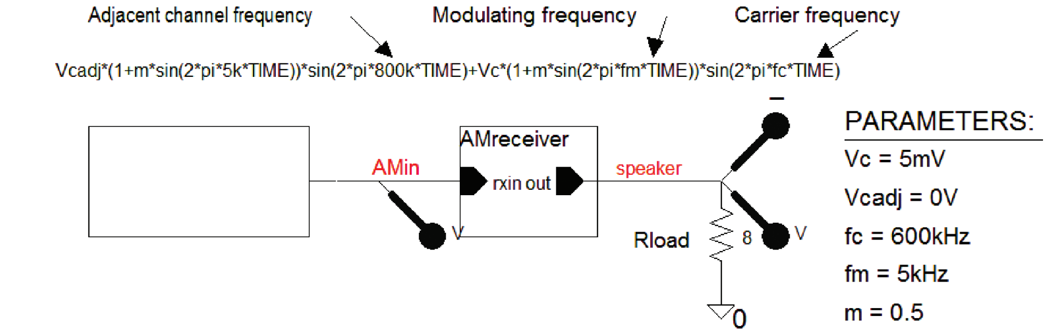


FIGURE 4.8: Adding an adjacent channel signal to the input signal

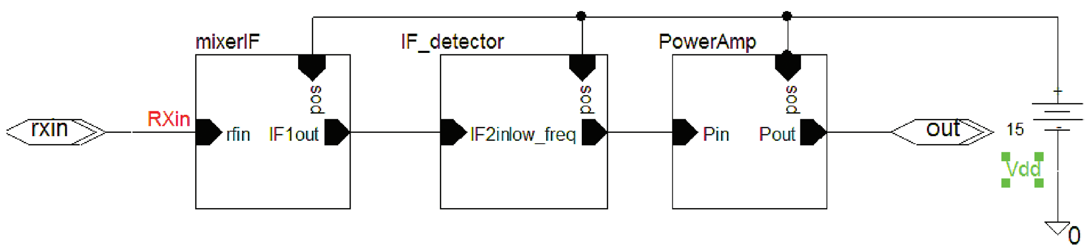


FIGURE 4.9: Individual stages

stage in Fig. 4.10 is both a mixer and a tuned RF amplifier with the center frequency being the intermediate frequency of 455 kHz. Mixing is achieved by using the nonlinear properties of the BJT biased in the nonlinear region. This produce mixing frequencies, of which one is the IF. The local oscillator (LO) signal is injected into the emitter using capacitor coupling C_2. The local amplitude must be higher than the RF signal because of losses in the mixing process. The tuned load is set to the IF and the output is transformer-coupled to the next stage.

4.3.2 The Local Oscillator: Arithmetic Selectivity

The local oscillator is implemented here using a **VSIN** generator part with **Ampl** = 1 V and **FREQ** = 1055 kHz. Selectivity is the ability to select and amplify desired signals but reject unwanted RF frequencies close to the desired carrier frequency. Adjacent channel selectivity occurs in the band-pass IF amplifier stages, where the amplifier selects the desired signal from **adjacent** signals and amplifies them to a level so that the detector operates correctly. (The detector needs at least a 600 mV input for a silicon diode to operate, but only 200 mV for a germanium diode.) Test receiver selectivity by injecting a signal comprising an adjacent signal added to the AM signal. The single-tuned RF stage in Fig. 4.11 provides amplification, selectivity against adjacent signals and provides automatic gain control (AGC) which is required

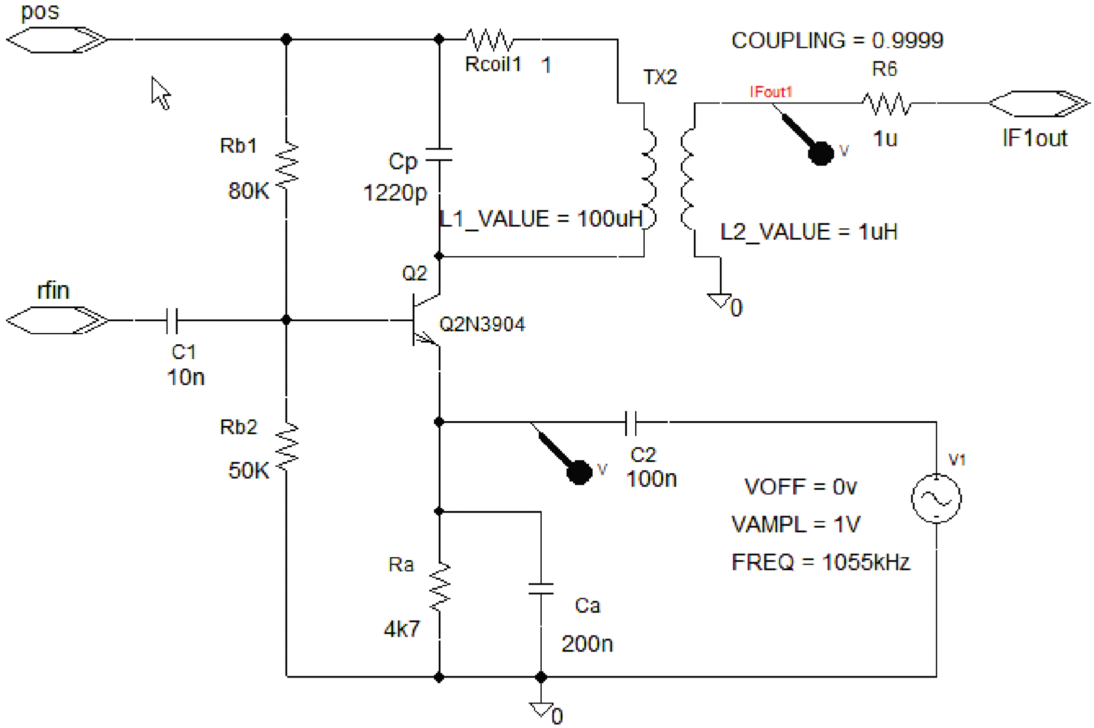


FIGURE 4.10: The mixer stage containing the local oscillator

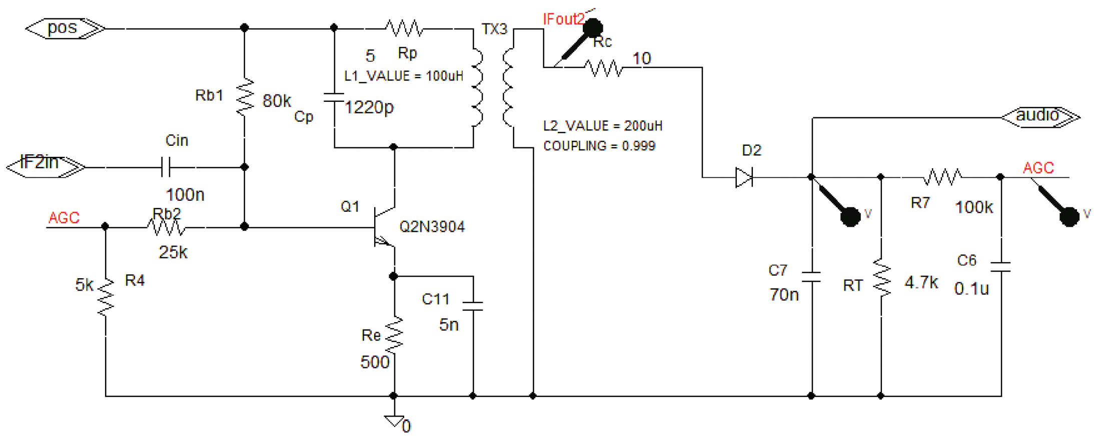


FIGURE 4.11: The first IF amplifier

to stabilize the receiver gain against a varying input RF signal. A feedback signal from the detector is injected via the base bias resistance, R_{b2}, to stabilize the output audio power against input RF signal fluctuations. Before we simulate the complete receiver let us look at the different parts of the system in order to appreciate how the receiver produces an audio signal from an RF signal. We have already looked at mixing so consider the RF amplifiers first.

4.3.3 JFET-Tuned Radio Frequency Amplifier

Single tuned RF amplifiers select and amplify weak RF signals and are the main parts of a receiver. We need to calculate unloaded and unloaded Q-factors and tuning capacitance C_T, if the circuit is to achieve maximum amplification at 100 kHz. You may assume an inductance of 1 mH. What is the −3 dB bandwidth and maximum voltage gain? The **VSIN** is set to 0.1 V at the resonant frequency f_0, and is applied to the circuit in Fig. 4.12 by renaming the input wire segment to **vin**. Set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 10 ms, and **Maximum step size** = 10 µs, press F11.

4.3.4 RF-Tuned Amplifier Measurements

Separate the input and output waveforms as shown in Fig. 4.13 using **alt PP** to open up a new plot. Copy the variable you want moved using **ctrl X** and then paste with **ctrl V**. Measure the maximum gain.

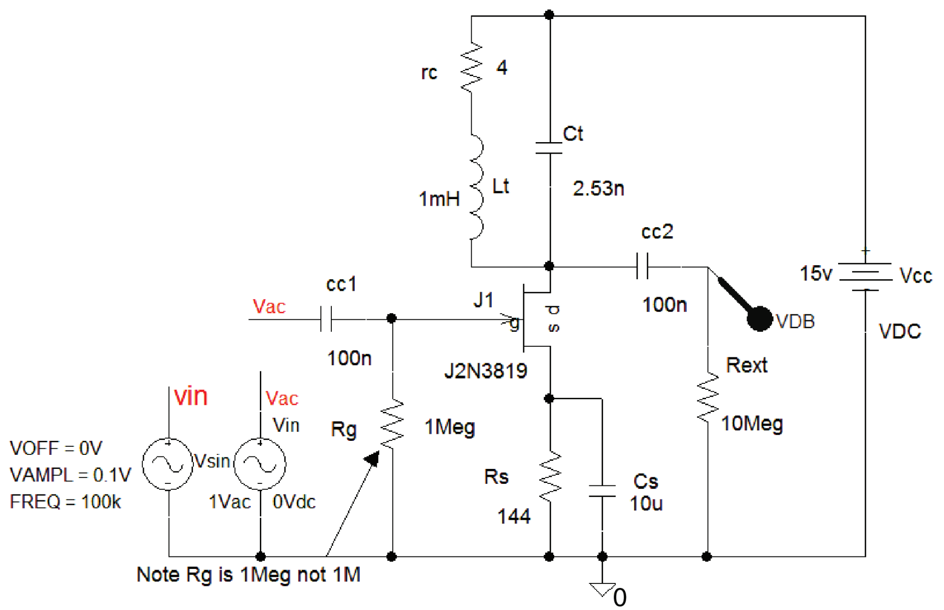


FIGURE 4.12: Selective RF amplifier tuned to 100 kHz

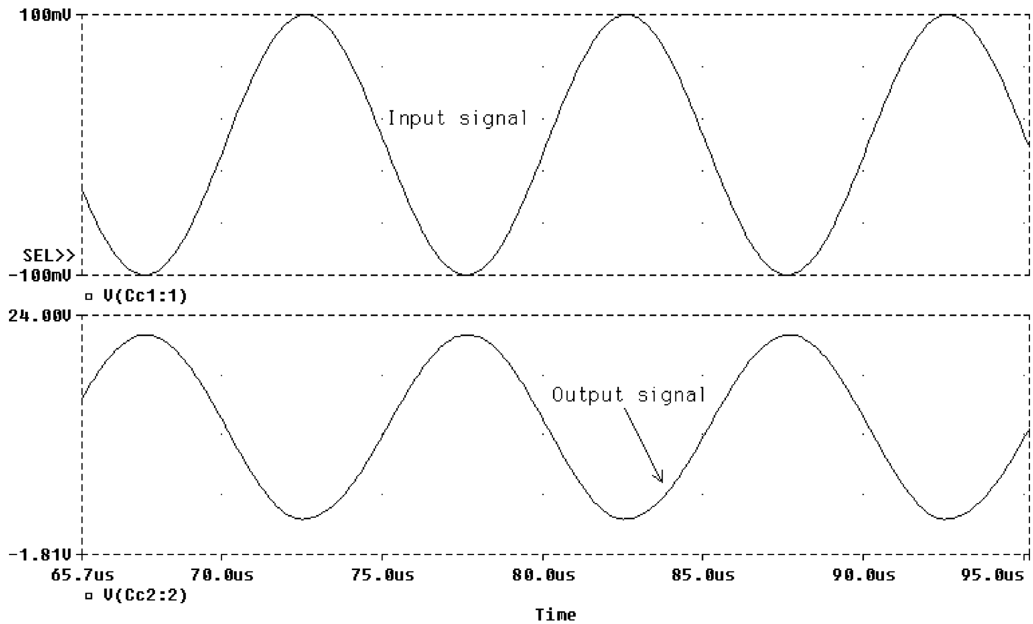


FIGURE 4.13: Input and output signals

This little symbol beside the variable name at the bottom is for locating the cursor on a trace. Name the input line segment vac to connect the **VAC** generator with amplitude set 1 V (20log 1 = 0 dB) to the input, and carry out an AC analysis centered at the resonant frequency. The **VSIN** generator may be used for an AC frequency response, but you cannot use the **VAC** generator for transient analysis. Change the output marker to vdB. From the **Analysis Setup,** select **AC Sweep** and **Linear, Points/Decade** = 1001, **Start Frequency** = 95k, and **End Frequency** = 105k. Simulate with **F11** to produce the response shown in Fig. 4.14. Measure the resonant frequency, the −3dB bandwidth, and the maximum gain. Determine the loaded Q-factor from these measurements, and compare the theoretical and simulated values. Check the DC conditions by selecting the **V** and **I** icons.

The maximum gain at the resonant frequency is

$$A_v|_{\text{dB}} = 20\log(g_m R) = 47.7 \text{ dB} \tag{4.9}$$

where R is the parallel combination of the dynamic impedance $R_p = L/Cr_c$ and the FET output source impedance r_{ds} (the drain–source impedance is measured from the inverse of the output FET characteristic). The bandwidth, resonant frequency, and loaded quality factor are related as

$$Q_{\text{loaded}} = \frac{f_o}{BW} = \frac{100 \text{ kHz}}{1.47 \text{ kHz}} = 68. \tag{4.10}$$

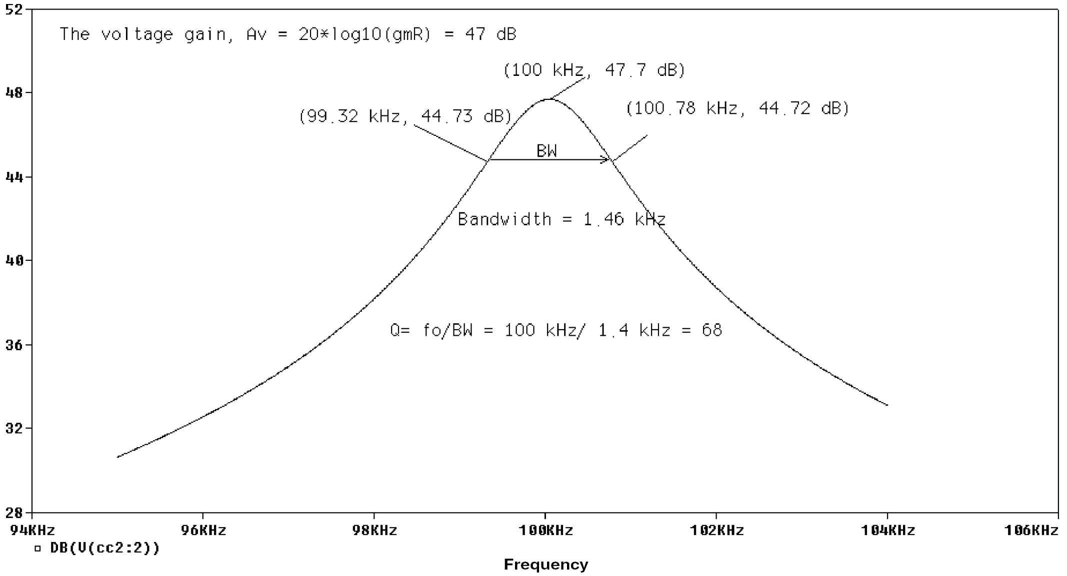


FIGURE 4.14: RF amplifier frequency response

Determine a suitable value for an external resistance R_{ex} when placed across the tuned circuit doubles the bandwidth (half the Q-factor). Remember the loaded Q-factor is defined as

$$Q_L = \frac{R}{X_{LP}} \approx \frac{R}{X_{Ls}} = \frac{r_{ds}R_p/(r_{ds}+R_p)}{2\pi f_o L} = \frac{R_p}{2\pi f_o L}\left(\frac{r_{ds}}{rds+R_p}\right) = \frac{Q_{UL}}{1+R_p/r_{ds}}. \tag{4.11}$$

(Clue: two 10-kΩ resistors in parallel are 5 kΩ.) Measure the bandwidth and gain for the new loading conditions [ref: 1 Appendix A].

4.4 MEASURING THE OUTPUT IMPEDANCE OF AN RF AMPLIFIER

This is a useful measurement technique that can be applied in different applications. Apply a 1-V **VAC** generator **vout** across the output, as shown in Fig. 4.15. The input signal generator is replaced by a source impedance (a short-circuit representing an ideal voltage source). A very large capacitance will do the same job. A coupling capacitor Cc_2 avoids changing the bias conditions by the low impedance of the **VSIN** generator when connected. Another reason for this capacitor is that PSpice will replace all sources with their internal resistance when doing a bias point calculation. If a current source is used instead, then the coupling capacitor is not required, as the source impedance is very large.

From the **Analysis Setup**, select **ACSweep** and **Linear, Total Points** = 1001, **Start Frequency** = 95k, and **End Frequency** = 105k. Simulate with **F11** to plot the the response

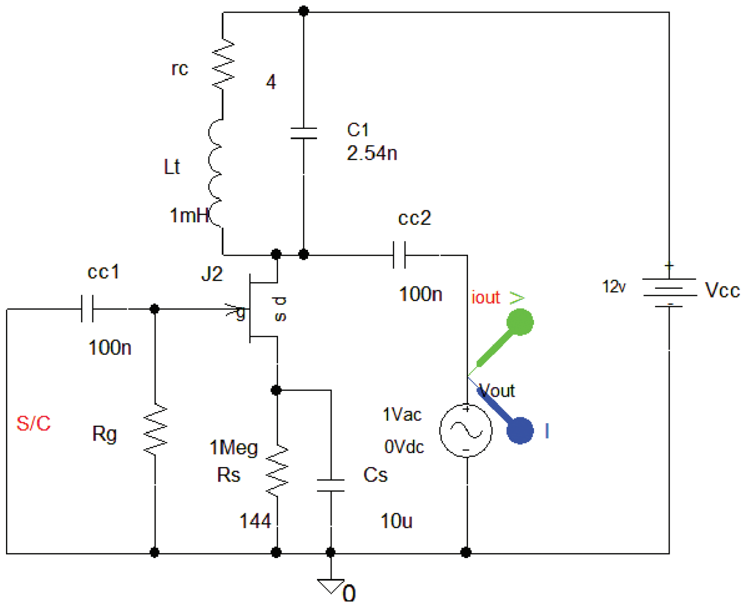


FIGURE 4.15: Measuring the output impedance

shown in Fig. 4.16. Select the **Trace Add** facility (alternatively, press the insert button on your computer) and add the two variables **vout/iout** from the list in the **Trace Expression** box to plot the impedance. Compare the measured 42.78 kΩ maximum impedance to the parallel combination of the dynamic impedance and the output drain–source impedance.

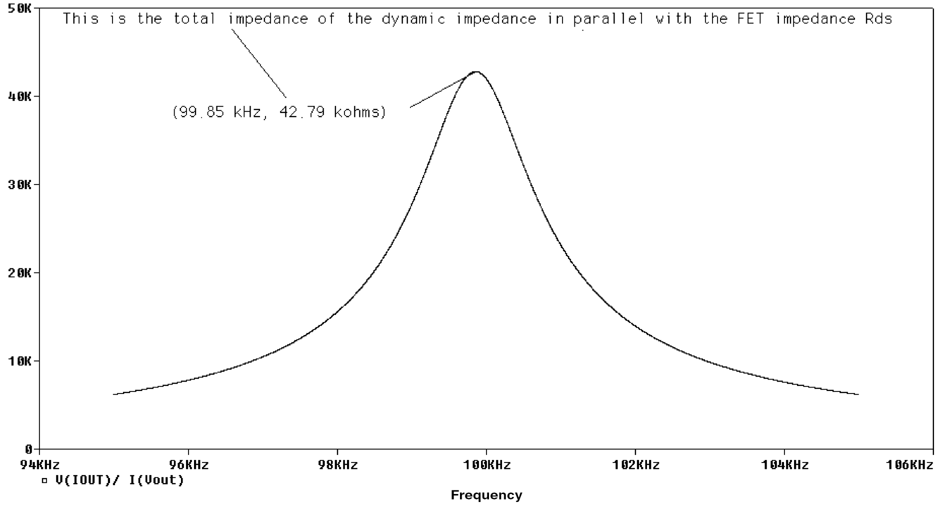


FIGURE 4.16: Plot of the output impedance

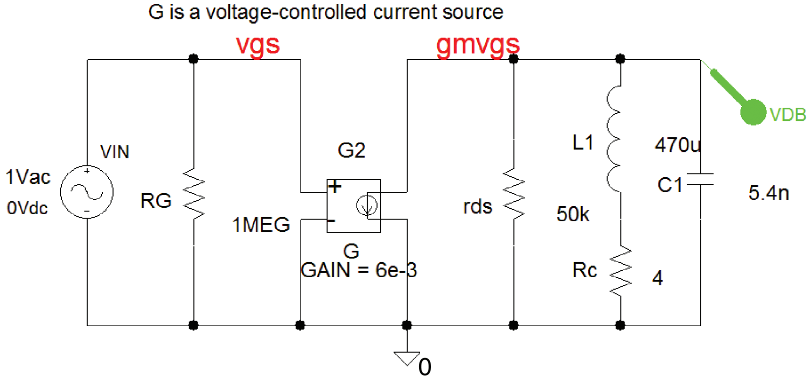


FIGURE 4.17: AC equivalent circuit

4.4.1 AC Equivalent Circuit

The JFET equivalent tuned RF amplifier in Fig. 4.17 is constructed from the parameters measured from the FET characteristics [ref: 1 Appendix A].

The voltage-controlled current source **G** has the gain set to the transconductance g_m from the transfer characteristic ($g_m = 6$ mS). The amplifier output resistance is the FET output resistance, r_{ds}, measured as 50 kΩ from the inverse of the slope of the output characteristic. From the **Analysis Setup**, select **ACSweep and Linear, Points/Decade** = 1001, **Start Frequency** = 95k, and **End Frequency** = 105k. Simulate by pressing **F11**, or press the little triangle. Measure the resonant frequency, gain, BW and Q-factor from the frequency response shown in Fig. 4.18.

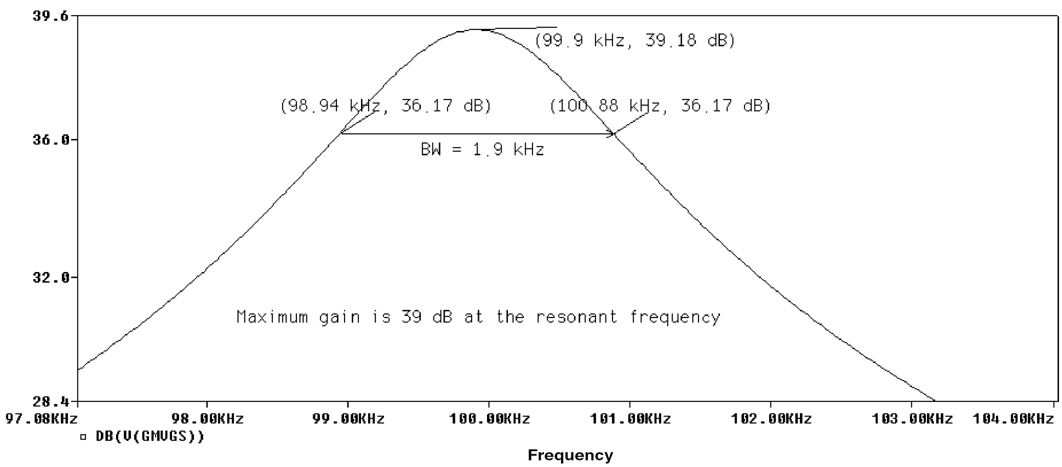


FIGURE 4.18: Frequency response of equivalent circuit

4.5 BJT BANDPASS AMPLIFIER

RF amplifiers generally use BJT devices, rather than FET devices, because they have a much higher gain. From the **Analysis Setup**, select **ACSweep** and **Linear, Total Points** = 1001, **Start Frequency** = 95k, **End Frequency** = 105k and simulate by pressing **F11**. Measure the tuned RF amplifier DC conditions shown in Fig. 4.19.

Measure the maximum gain and bandwidth from the bandpass frequency response in Fig. 4.20 and compare the results to those from the tuned JFET amplifier response.

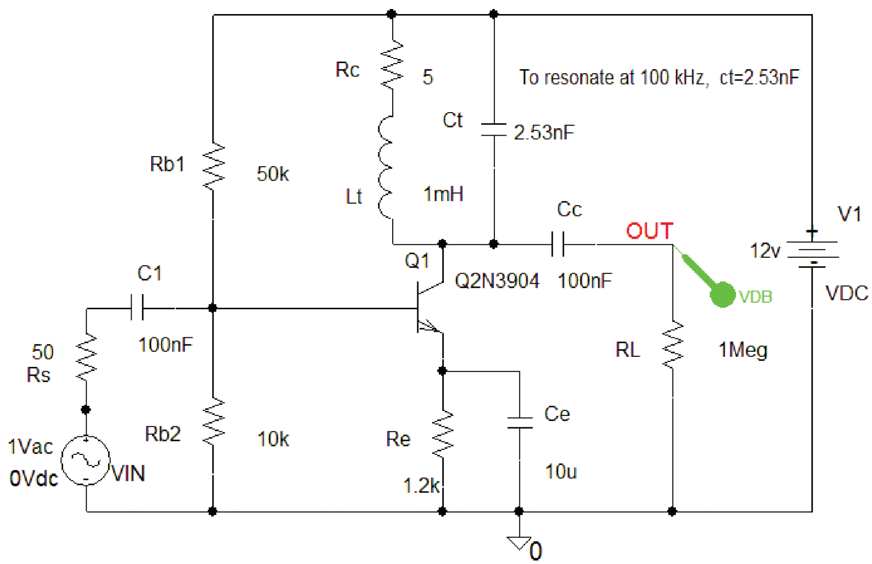


FIGURE 4.19: RF amplifier using an NPN device

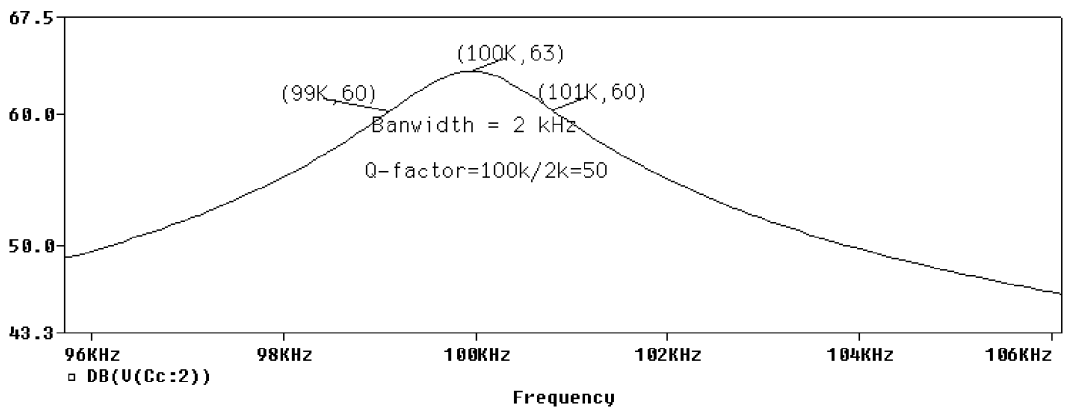


FIGURE 4.20: Amplifier response

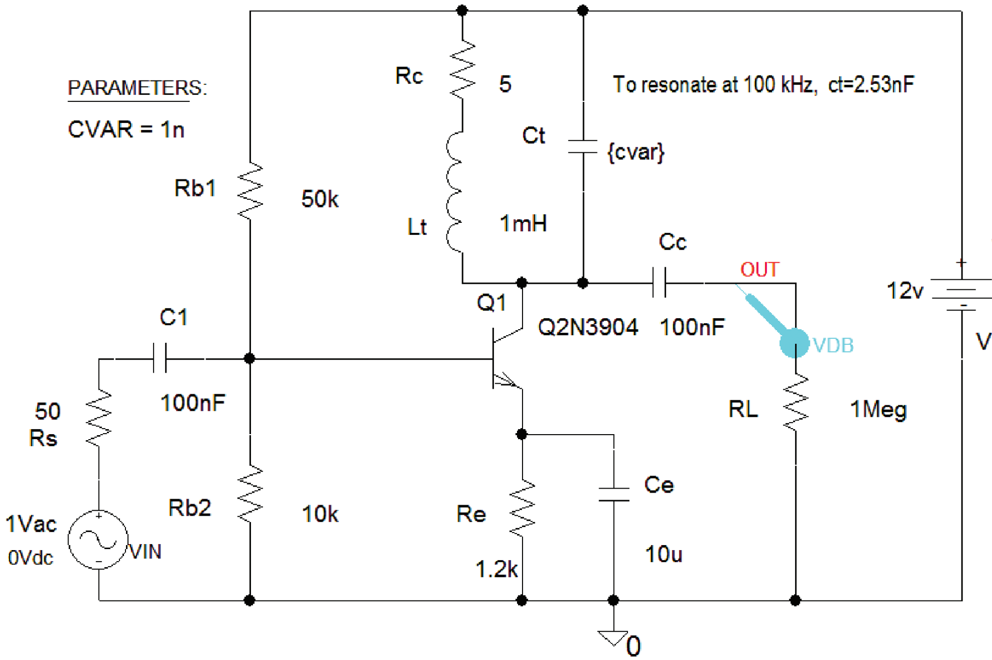


FIGURE 4.21: The PARAM part defines the tuning capacitance C_t

4.5.1 Tuning Capacitance

This schematic in Fig. 4.21 demonstrates how the tuning capacitance Ct changes the resonant frequency. Select the **Param** part, **Rclick,** and select **Edit Properties**. **Press New Row** and in the **New Row** and fill in **Name** = **cvar** and **Value** = 1 nF. The tuning capacitance value is replaced with {**var**} (you must use the curly brackets). From the **Analysis Setup/Parametric** menu set **Name** equal to cvar (no curly brackets), **Start Value** = 0.1 nF and **End Value** = 50 nF, **Increment** = 5 nF. From the **Analysis Setup**, select **ACSweep** and **Linear, Total Points** = 1001, **Start Frequency** = 95k, and **End Frequency** = 105k. Simulate with **F11** to plot the the response shown in Fig. 4.22.

The resonant frequency shifts for each tuning capacitor value as shown in Fig. 4.22. The Q-factor is expressed as

$$Q_L = \frac{R}{X_L} = \frac{R}{\omega_o L} \approx \frac{R}{L/\sqrt{LC}} = R\sqrt{\frac{C}{L}}. \tag{4.12}$$

The gain is also dependent on the tuning capacitance, since the dynamic impedance changes ($R_d = L/R_c C_t$).

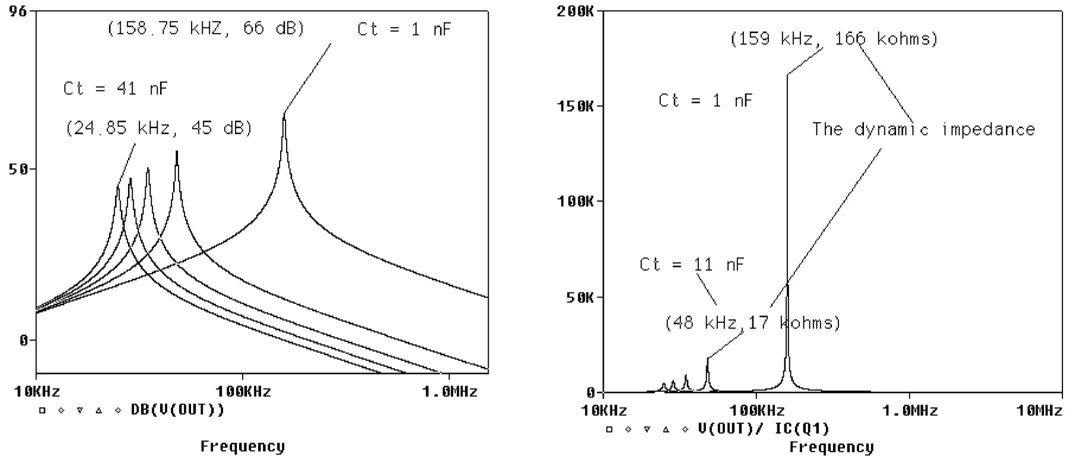


FIGURE 4.22: Variable tuning capacitance C_t

4.6 DIODE DETECTION AND AUTOMATIC GAIN CONTROL

The detector circuit, besides recovering the modulating signal, also provides a means of controlling the receiver gain automatically. The AGC regulates the output level against input carrier amplitude changes due to transmission attenuation. (For example, driving away from the transmitter.) The detector circuit investigated in Chapter 2, Fig. 2.6, showed how the AGC signal was generated by a LPF with a very low cut-off frequency (almost DC = 1 Hz). The AGC filter output is a controlling voltage, which, when applied to the first IF amplifier bias circuits, controls the gain of this stage. If the input carrier signal becomes weaker, then a reduced AGC voltage fed back increase the IF amplifier gain, and thus compensates for the reduced signal level. Of course, when the input signal is too strong the opposite happens, and the gain is reduced. However, in this schematic we did not include AGC but in Chapter 6 we will show how AGC can be applied back to the IF amplifier circuits to stabilise the gain.

4.7 POWER AMPLIFIER STAGE

The power amplifier in Fig. 4.23 is a standard push-pull configuration with diodes to avoid crossover distortion [ref: 1 Appendix A]. The power transistors used here are TIP31 and TIP32 types but you may use any transistor for the initial investigation.

4.8 AUDIO OUTPUT SIGNALS

The audio time and frequency signals are shown in Fig. 4.25 with a transient part at the start of the simulation. The root mean square (RMS) of the output power is plotted as **RMS(I(Rload)*V2(Rload))**. We would need to include a voltage preamplifier to increase the output power but the demo version will not work if we include more components.

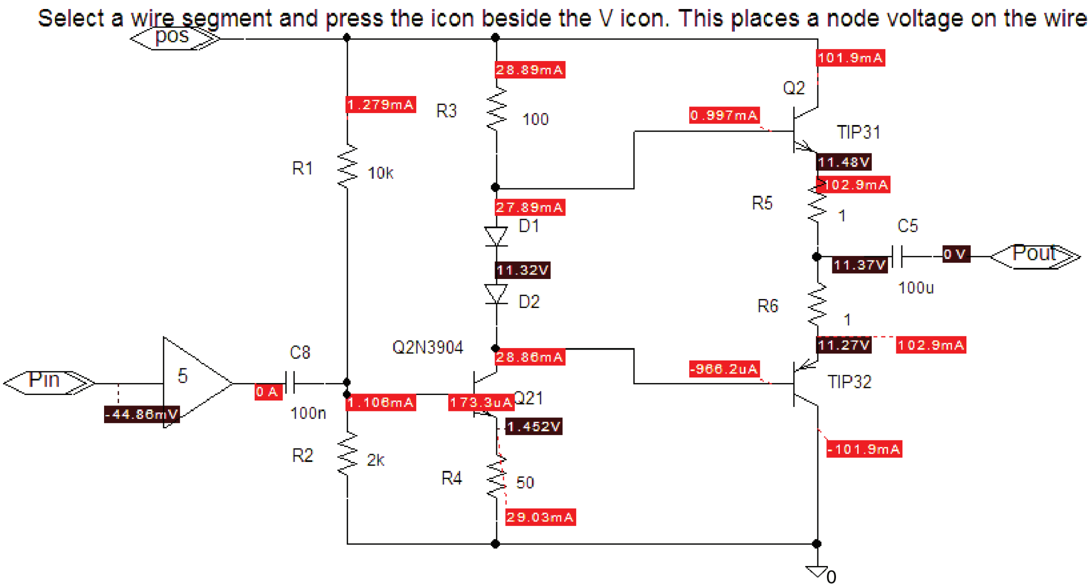


FIGURE 4.23: The power amplifier

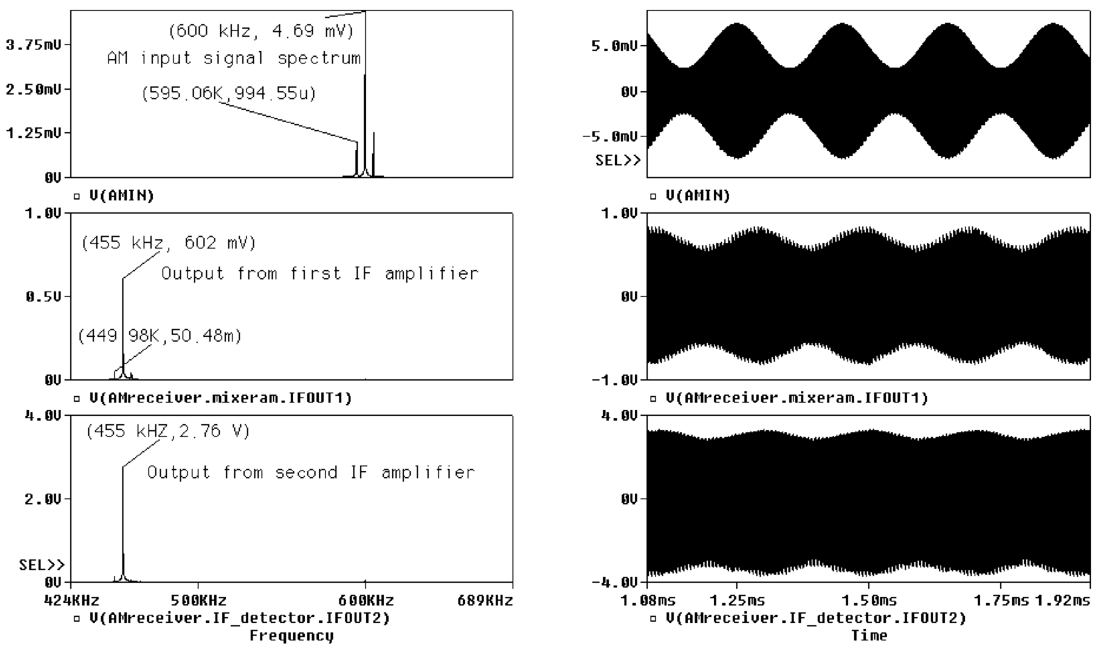


FIGURE 4.24: Superhetrodyne waveforms

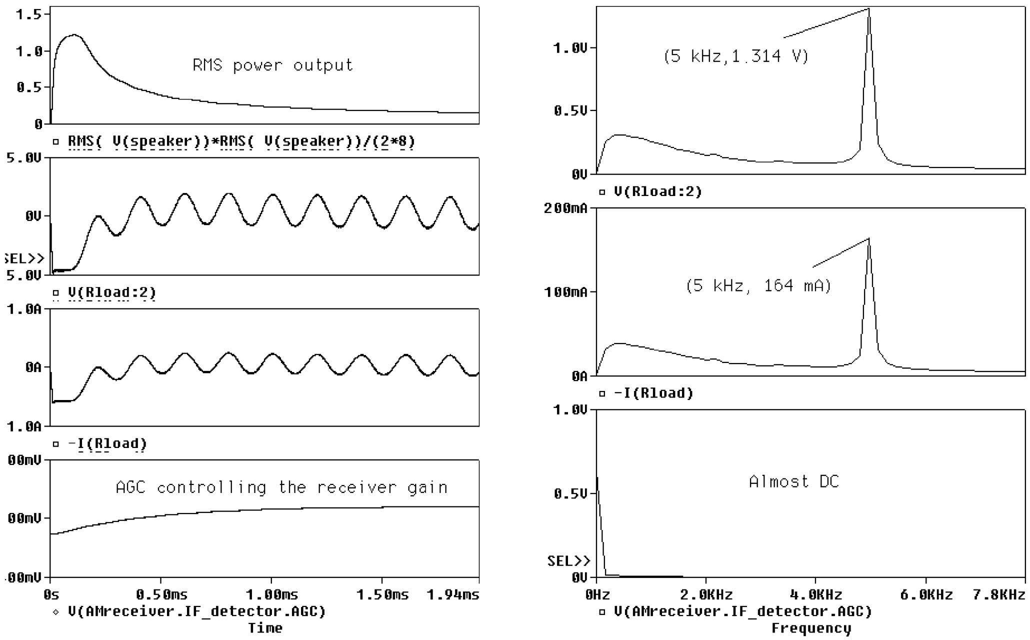


FIGURE 4.25: Receiver audio signals

4.9 RF SIGNALS

We are now in a position to appreciate what goes on inside the receiver, so set the Analysis tab to Analysis type: **Time Domain** (Transient), **Run to time** = 2 ms, and **Maximum step size** = 10 μs and press F11 to simulate. The RF receiver signals shown in Fig. 4.24 are plotted using separate windows, and the log command for logging Probe keystrokes. This is a useful technique when you need to repeat a simulation for different design parameters.

4.10 SPEECH SCRAMBLING

A simple speech scrambling modulation technique is shown in Fig. 4.26. The signal to be scrambled is modulated and then filtered by a high-pass filter to remove the low sideband. It

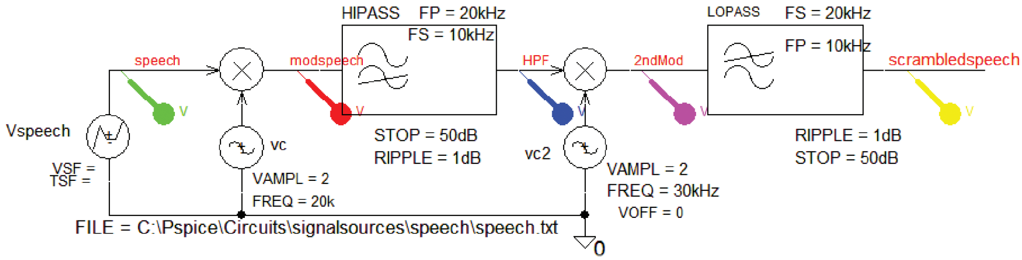


FIGURE 4.26: Speech scrambling

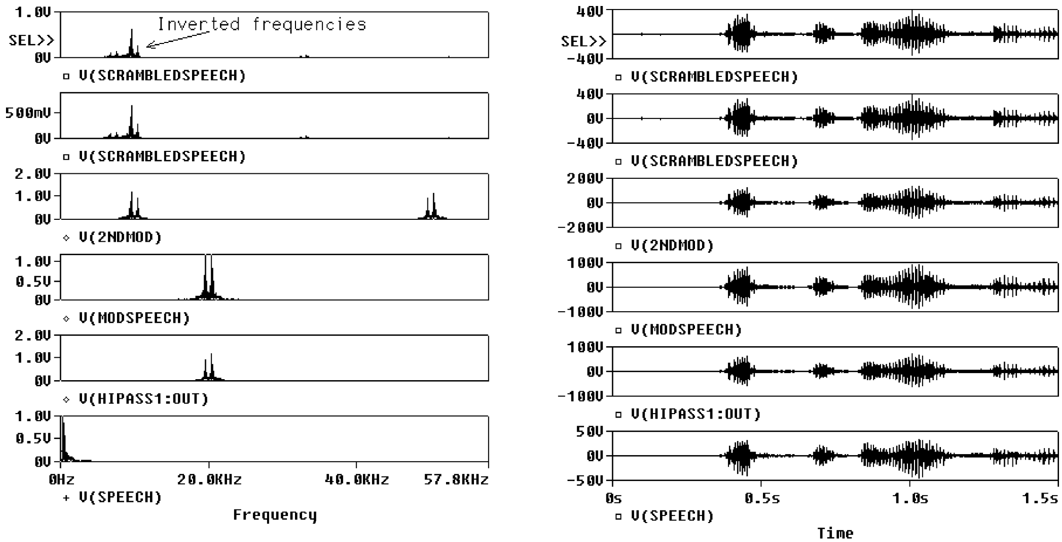


FIGURE 4.27: The original signal is now frequency inverted

is then further modulated and a low-pass filter removes the upper sideband. What you are left with is an inverted lower sideband in the audio band that is, hopefully, unintelligible. Use the executable program **wav2ascii.exe** to listen to the scrambled speech by selecting the output variable in Probe and **Ctrl C**. Alternatively, you may export the scrambled signal to Matlab and reconstruct the speech (consult [ref: 2 Appendix A] on how to use **wav2ascii.exe**).

The inverted speech spectrum is evident from the last plot in Fig. 4.27.

4.11 EXERCISES

(1) Draw the simple receiver schematic in Fig. 4.28 and investigate AGC applied to the FET input.

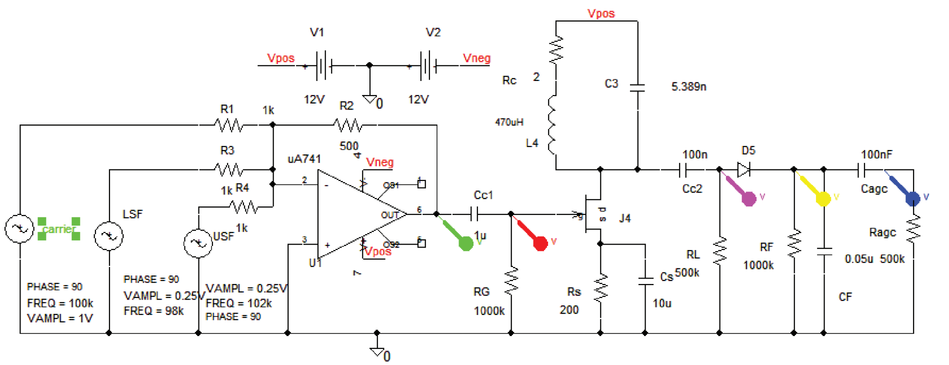


FIGURE 4.28: AGC application

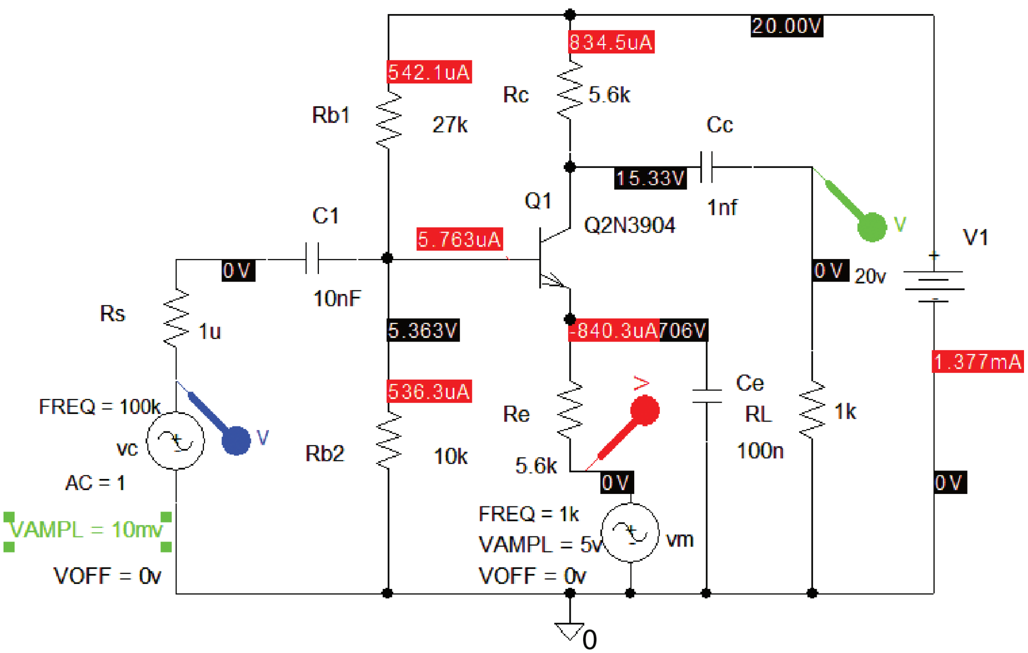


FIGURE 4.29: AM production using a BJT

(2) Investigate AM generation in Fig. 4.29 using a BJT biased in the nonlinear region. The FFT, or the output file, should show the distortion in the signal and the modulation products. Investigate the effect on the AM waveform for different RF and modulating signal amplitudes.

(3) The product detector in Fig. 4.30 recovers the modulation signal from an SSBSC signal.

(4) The time and frequency domain signals are shown in Fig. 4.31. (Press the **FFT** icon in Probe to observe the frequency domain signals.)

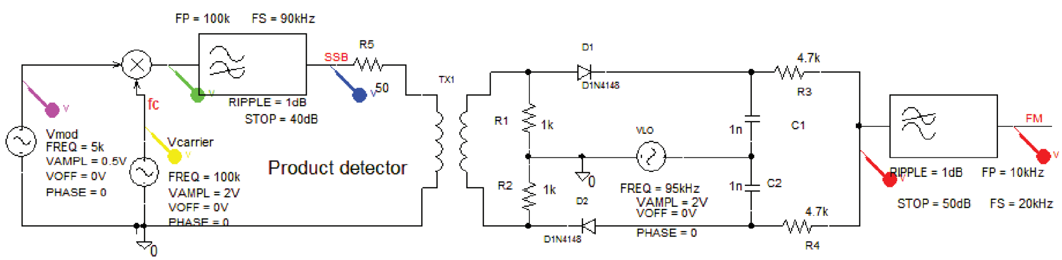


FIGURE 4.30: SSBSC and product detection

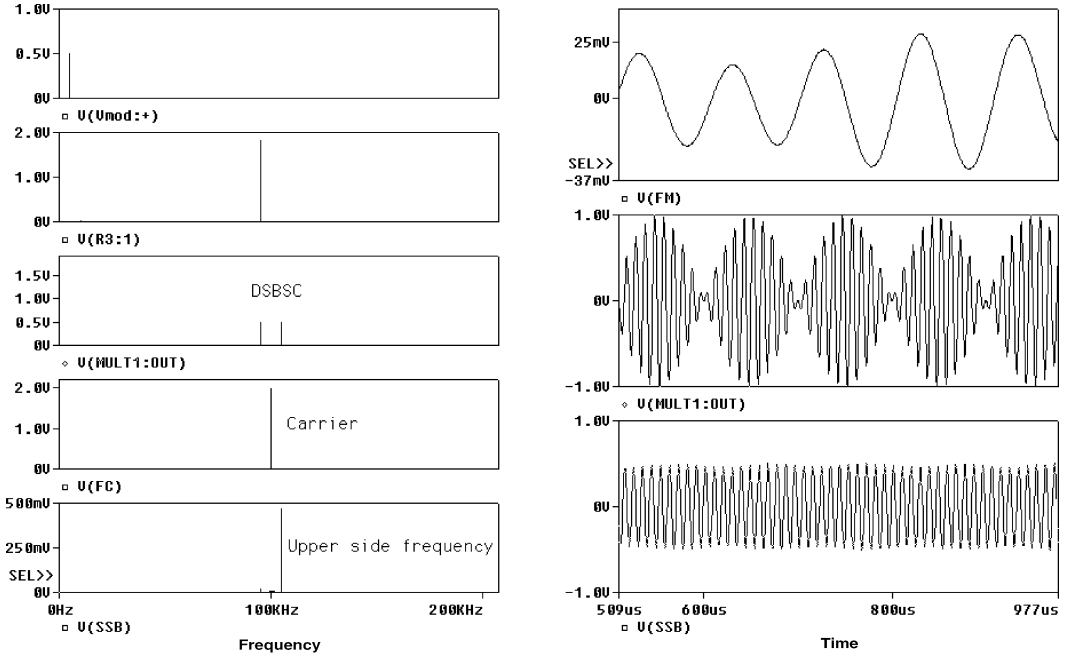


FIGURE 4.31: SSBSC signals in time and frequency

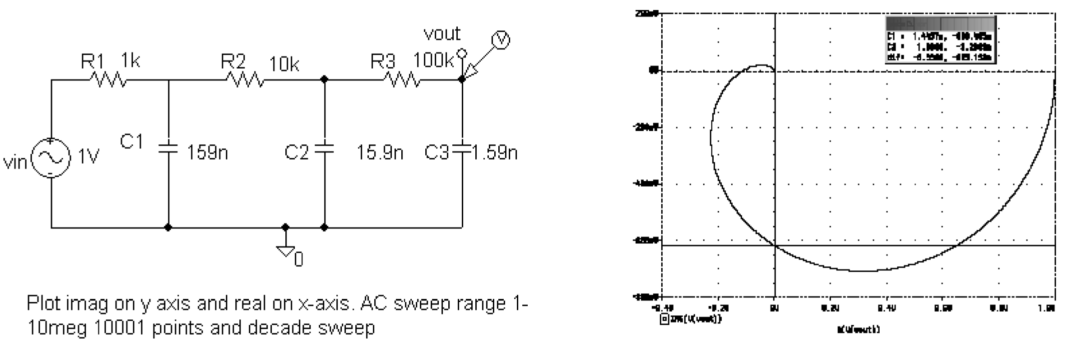


FIGURE 4.32: Placing a bitmap on the schematic area

Investigate the recovered speech modulation when the local oscillator has frequency and phase errors.

(5) Figure 4.32 shows how a bitmap is placed on a schematic using the **Place/Picture** menu. Make sure that the bitmap is in the same location as the original schematic, otherwise you might not be able to run a simulation, i.e., it looks for the bitmap and generates an error if it cannot locate it. Investigate the network and plot the imaginary

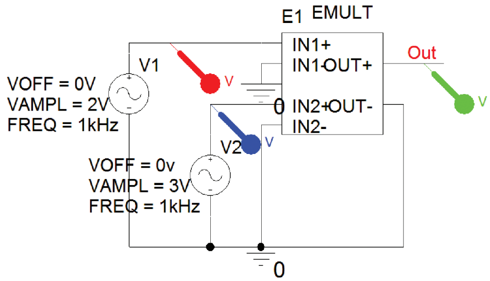


FIGURE 4.33: EMULT part

component, **IMG(V(vout))** of the output variable **vout,** on the y-axis and change the x-axis to the real part of the output **R(V(vout))** and **Scale** to **Linear**. The frequency range is 1 Hz to 10 MHz, and **Points per decade** = 10001.

(6) **EMULT** part may be used to multiply two inputs as in Fig. 4.33.

CHAPTER 5

Frequency Modulation Principles

5.1 MODULATION INDEX

The instantaneous value of a carrier signal is $v_c(t) = V_c \cos(2\pi f_c t + \phi)$. In amplitude modulation, the carrier amplitude, V_c, is changed by the modulating signal, but in frequency modulation the carrier amplitude is kept constant and the carrier frequency is varied by the modulating signal. Changing the phase, ϕ, but keeping the carrier frequency constant, produces phase modulation. A frequency-modulated carrier is produced when the frequency of a voltage-controlled oscillator (VCO) is changed by a modulating signal but the amplitude is kept constant. We need to derive an expression for the instantaneous voltage of an FM signal in order to understand the basic principles. The definition of angular frequency (radians per second) is the rate of change of phase with time:

$$\omega = \frac{d\theta}{dt}. \tag{5.1}$$

Integrating both sides of (5.1) yields an expression for the phase of a carrier, f_c, as

$$\theta = \int_0^t \omega_c dt = 2\pi \int_0^t f_c dt. \tag{5.2}$$

The carrier frequency is deviated by Δf_c and has a peak value Δf_{cpk} for a maximum value of the modulating amplitude. Thus, the carrier frequency is

$$f_c = f_c + \Delta f_{cpk}. \tag{5.3}$$

The modulator sensitivity, k_0, is defined as the change in frequency per volt change in the input voltage as

$$k_0 = \Delta f_{cpk} / V_m \text{ kHz V}^{-1}. \tag{5.4}$$

The change in the carrier frequency is the **peak frequency deviation** $\Delta f_{cpk} = k_o V_m$. Substituting (5.4) into (5.3) and then into (5.2) gives the phase as

$$\theta = 2\pi \int_0^t (f_c + \Delta f_{cpk}) dt = 2\pi f_c t + 2\pi \int_0^t (\Delta f_{cpk}) dt = 2\pi f_c t + 2\pi k_o \int_0^t V_m(t) dt. \tag{5.5}$$

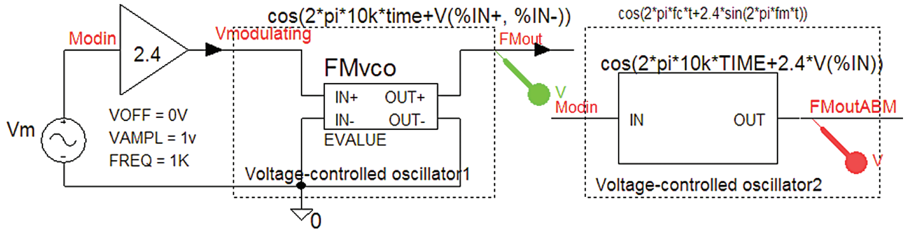


FIGURE 5.1: Two ABM voltage-controlled oscillators

Substitute a cosine modulating signal $v_m(t) = V_m \cos \omega_m t$ into (5.5) and integrate to yield

$$v_{FM}(t) = V_c \cos\left\{2\pi f_c t + \frac{2\pi k_0 V_m \sin 2\pi f_m t}{2\pi f_m}\right\} = V_c \cos\left\{2\pi f_c t + \left(\frac{k_0 V_m}{f_m}\right) \sin 2\pi f_m t\right\}. \tag{5.6}$$

Define the FM modulation index beta as

$$\beta = \frac{k_0 V_m}{f_m} = \frac{\Delta f_{cpk}}{f_m}. \tag{5.7}$$

Thus, the instantaneous voltage of an FM carrier signal, modulated by a sinusoidal voltage, V_m, is

$$v_{FM}(t) = V_c \cos\left(2\pi f_c t + 2\pi k_o \int_0^t V_m dt\right) = V_c \cos(2\pi f_c t + \beta_m \sin 2\pi f_m t). \tag{5.8}$$

5.2 FM SPECTRUM

An FM spectrum consists of a carrier component J_0 at ω_c, and an infinite number of spectral lines at $\omega_\mathrm{c} \pm n\omega_\mathrm{m}$ whose amplitude are computed using **Bessel** functions (Fredrich Bessel 1784–1846). An FM signal is defined as

$$v_c(t) = V_c \cos(\omega_c t + \beta \sin \omega_m t) \text{ V}. \tag{5.9}$$

Expand (5.9) using Bessel functions:

$$\begin{aligned} v_c(t) = V_c\{&J_o(\beta)\sin(\omega_c t + J_1(\beta)[\sin(\omega_c + \omega_m)t - \sin(\omega_c - \omega_m)t] \\ &+ J_2(\beta)[\sin(\omega_c + 2\omega_m)t + \sin(\omega_c - 2\omega_m)t] \\ &+ J_3(\beta)[\sin(\omega_c + 3\omega_m)t - \sin(\omega_c - 3\omega_m)t] + \cdots\}. \end{aligned} \tag{5.10}$$

The sidebands are contained in the table of Bessel functions in the appendix and are computed down to 1% of the carrier. Fig. 5.1 shows two methods for producing an FM signal using ABM **Evalue** and **ABM1** parts:

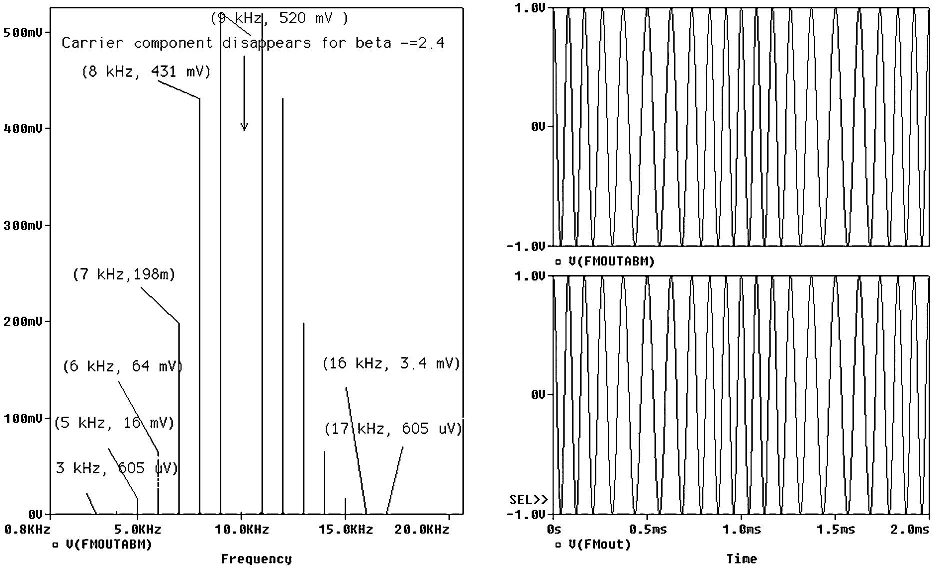


FIGURE 5.2: VCO signal and spectrum

Set the **Analysis Set up/Transient** parameters: **Output File Options/Print values in the output file** = 100 ms, **Run to time** = 200 ms, maximum step size = 1 µs. Press **F11** and select the FFT icon in Probe to show that the FM spectrum spectrum in Fig. 5.2 has no carrier component for beta = 2.4.

5.3 FM PRODUCTION USING THE VSFFM GENERATOR PART

Draw the schematic shown in Fig. 5.3. (The current source part is **ISFFM**.) The generator parameters—carrier **FC**, modulation index **MOD**, modulating signal **FM,** and carrier amplitude **VAMP—**are set to the values shown.

A modulating frequency f_m produces a maximum change in the carrier $\Delta f_{cpk} = \beta f_m$ so the carrier frequency, $f_c + (\Delta f_{cpk}/f_m)$, varies between $f_c - \Delta f_{cpk}$ and $f_c + \Delta f_{cpk}$. The modulation index β is a measure of the peak frequency deviation, Δf_{cpk}. Make the transient **Maximum step size** value at least one tenth of the carrier period. Set the transient analysis

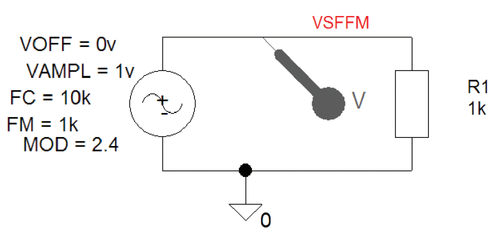


FIGURE 5.3: VSFFM parameters

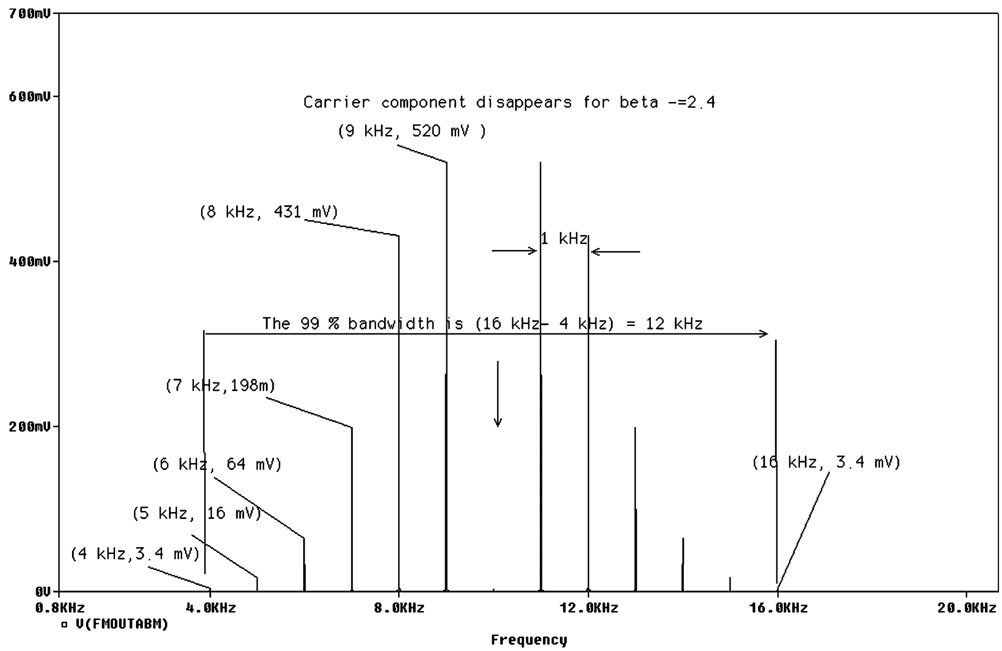


FIGURE 5.4: FM spectrum

parameters: **Run to time** = 20 ms, and **Maximum step size** = 1 μs. Press **F11** to simulate. The FM signal and corresponding spectrum in Fig. 5.4 should appear. Spectral components for a 10-kHz carrier, modulated by a 1-kHz signal and with a modulation index = 2.4, are shown in Fig. 5.4.

Note that the carrier disappears for this particular value of beta. The 99% bandwidth is defined as the frequency range formed by considering the frequency of the spectral component which is 1% of the J_o carrier component, to the frequency of the upper spectral component which is also 1% of the J_o carrier component. We see from the previous example, how the FM spectrum has a 99% bandwidth equal to 12 kHz. However, a modified form of Carson's rule is used in practice [ref: 5 Appendix A] and is given as

$$\mathrm{BW} = 2\ \mathrm{fm}(2+\beta) = 2.10^3(2+2.4) = 8.8\ \mathrm{kHz}. \tag{5.11}$$

5.3.1 Power in an FM Signal

The total power in an FM signal developed across a resistance $R1 = 50\ \Omega$ is plotted by pressing the insert key in Probe and entering the expression **RMS(V2(R1))*RMS(V2(R1))/(50)** in the **Trace expression** box. Information power, **rms(V2(R1))*rms(V2(R1))/(50) − J(o)* J(o)/100**, is plotted as total power–carrier power shown in Fig. 5.5. The carrier component $J(0)$ is zero for

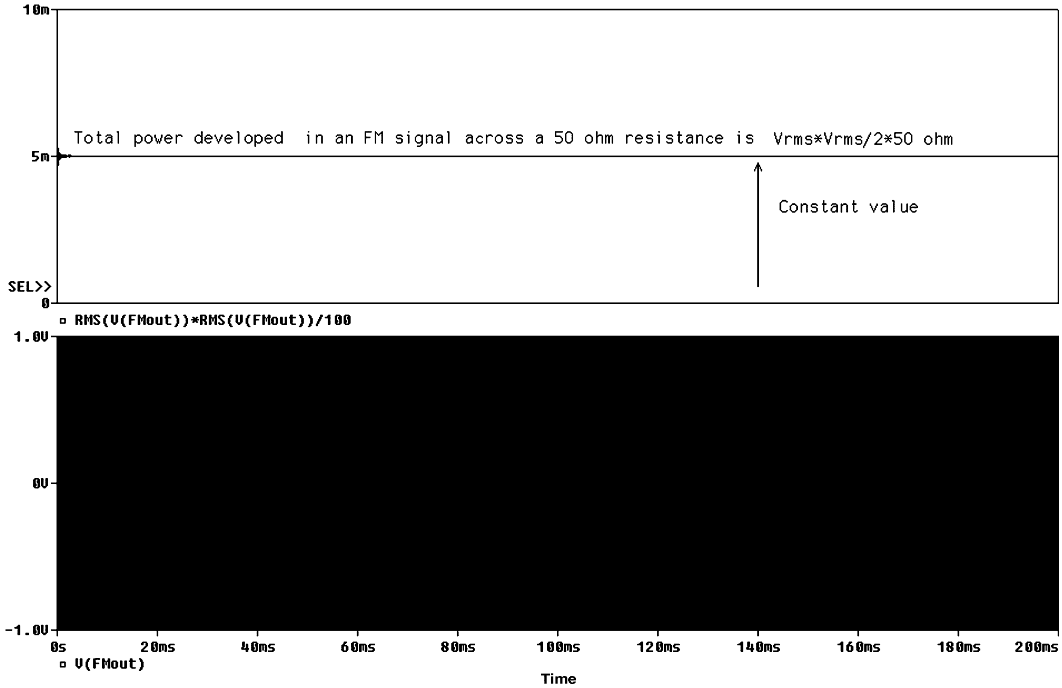


FIGURE 5.5: FM spectrum

a modulation index = 2.4; hence, we need to add all the power components for each sideband. The FM signal power is also shown having a constant value.

5.4 VARACTOR DIODE

To understand how a varactor diode controls the frequency in a VCO, we must investigate the capacitor/voltage characteristics. The diode capacitance varies when the reverse-biased voltage, connected as shown in Fig. 5.6, is swept over a range of voltage values. From the **Analysis**

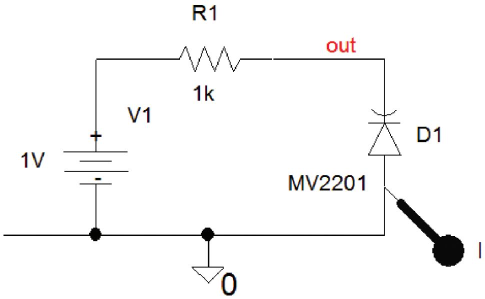


FIGURE 5.6: A reverse-biased varactor diode

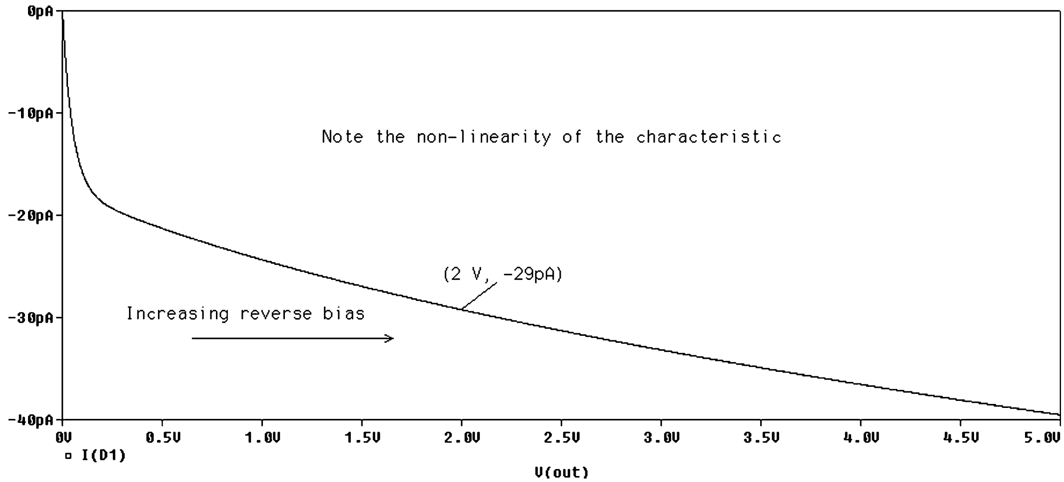


FIGURE 5.7: Vin versus diode capacitance

Setup menu, select **DC Sweep** and place a tick on **Swept Var type** and in the **Name box** enter **Vin**, which is the voltage to be swept from 0 V to 5 V in increments of 0.001 V. The **Sweep Type** is **Linear**.

The relationship between the zero bias capacitance C(0), and transition width capacitance $C_T(V_R)$ is:

$$C_T(V_R) = \frac{C(0)}{[1 + \frac{V_R}{V_k}]^n} \tag{5.12}$$

V_R is the reverse bias voltage, V_k is the knee voltage, and $n = 0.5$.

The schematic in Fig. 5.8 contains a **PARAM** part to produce a range of bias voltages Vin (defined as vcap in the **Param part** which is short for varicap), which in turn varies the capacitance of the Varactor diode. To determine a suitable AC analysis range, we must calculate the resonant frequency using the relationship:

$$\omega_o^2 \approx 1/LC \Rightarrow f_c \approx 1/(2\pi\sqrt{LC}). \tag{5.13}$$

Substituting values for L, and the Varicap capacitance C, yields

$$f_c \approx 1/(2\pi\sqrt{LC} \approx 1/(2\pi\sqrt{0.162\text{ mH} \times 15.88\text{ pF}} = 3.3\text{ MHz}. \tag{5.14}$$

A suitable AC linear frequency range is 3 MHz to 5 MHz. A current source avoids loading the tuned circuit. Of course, in PSpice, sources are ideal but it is good practice to include a source resistance.

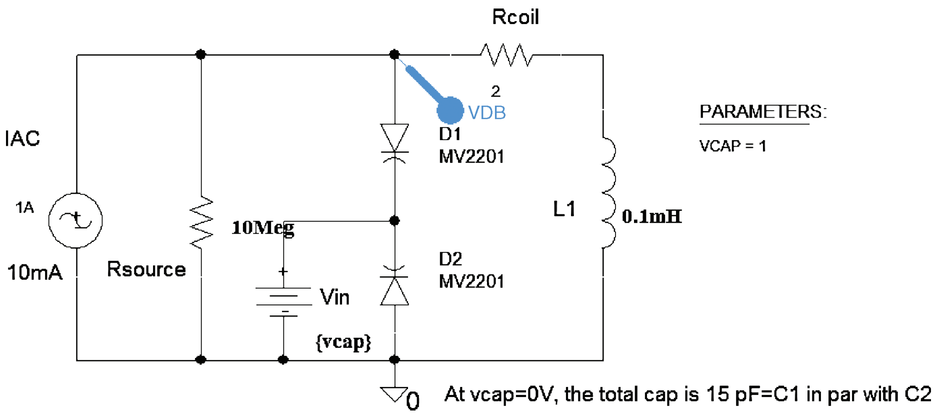


FIGURE 5.8: AC analysis with Vin as a swept variable

Fig. 5.9 shows the variation in the resonant frequency for a range of varicap voltages. From the **Analysis Setup**, select **Parametric** and set the **vcap** from 0 to 5 V, in steps of 1 V.

5.4.1 FM Oscilloscope Display

The schematic in Fig. 5.10 produces the type of display you would get on an oscilloscope, when an FM signal is connected to the x-input and with certain "scope time-base settings." A saw-tooth time-base signal is produced using a **VPULSE** part but with the rise time parameter **TR** set to the **PER** value.

The percentage carrier frequency deviation can be measured from the time display in Fig. 5.11. After simulation, change the Probe window x-axis, by selecting a time-base signal from the parameter list (see index for the change of the x-axis variable procedure). The signal is then wrapped from the right edge of the display to the left edge, and shows a number of overlapping cycles and simulates how an oscilloscope would display an FM signal.

5.4.2 FM Preemphasis and Deemphasis

Good noise performance in FM is achieved by preemphasizing the higher modulating frequencies before modulation. To see this we must consider the FM modulation index:

$$\beta = \frac{\Delta f_{cpk}}{f_m} = \frac{k_o V_m}{f_m} = \frac{2\pi k_o V_m}{2\pi f_m}. \tag{5.15}$$

The instantaneous value of an FM signal is

$$v_{FM}(t) = A_c \cos(2\pi f_c t + \beta_m \sin 2\pi f_m t). \tag{5.16}$$

A maximum modulating frequency, f_m, produces a peak frequency deviation $\Delta f_{cpk} = \beta f_m$, thus the modulation index is inversely proportional to the modulation frequency and directly proportional to the modulation signal amplitude. Hence, higher modulation frequencies produce smaller modulation index values so we need preemphasize the higher frequencies in the modulating signal before modulating the carrier. Preemphasizing the higher modulating frequencies is carried out using the modified high-pass *CR* filter shown in Fig. 5.12. Select components to produce two cut-off frequencies at 1.59 kHz and 15 kHz.

The preemphasis network transfer function is

$$TF = \frac{R_2/(R_1 + R_2)(1 + j\omega C R_1)}{1 + j\omega C R_2 R_1/(R_1 + R_2)} = R_2/(R_1 + R_2)\left(\frac{1 + j\omega/\omega_{c1}}{1 + j\omega/\omega_{c2}}\right). \tag{5.17}$$

The European time constant τ_1 is $CR_1 = 50\ \mu s$ (75 μs in the USA). We make the approximation that $CR_2 \gg CR_1$ and $R_1 = 1.5\ k\Omega$, $C = 33$ nF, and $R_2 = 320\ \Omega$. From the **Analysis Setup**, select **AC Sweep** and **Decade**, **Pts/Decade** = 1001, **Start Freq**uency = 10, and **End Frequency** = 10 Meg. Press **F11** to simulate and measure the cut-off frequencies on the amplitude response in Fig. 5.13.

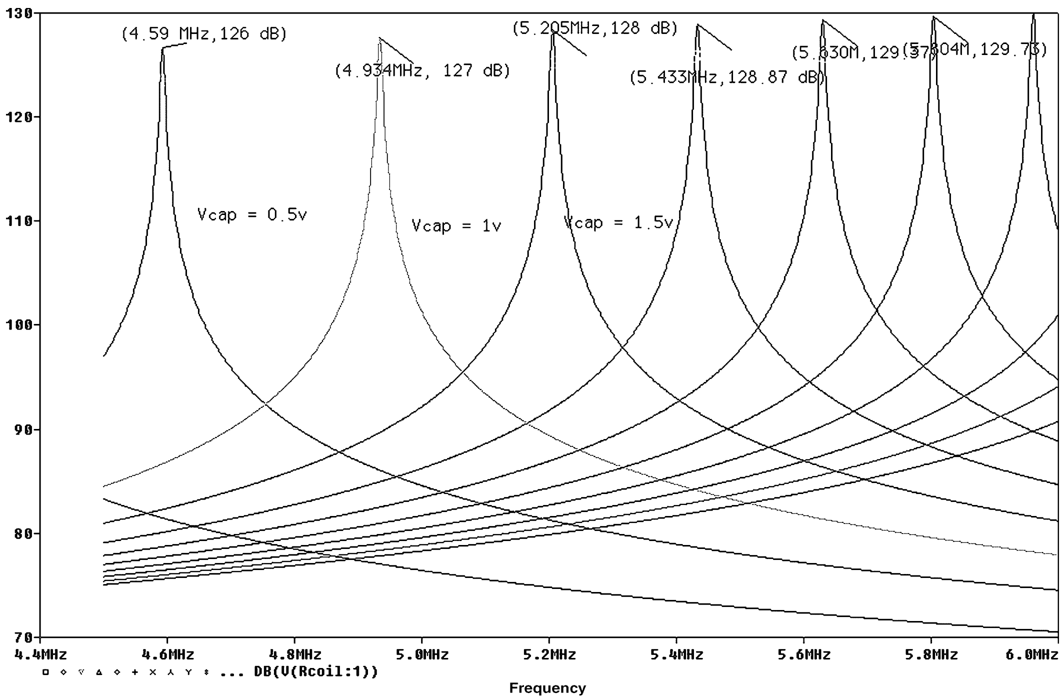


FIGURE 5.9: Sweeping the varicap bias voltage

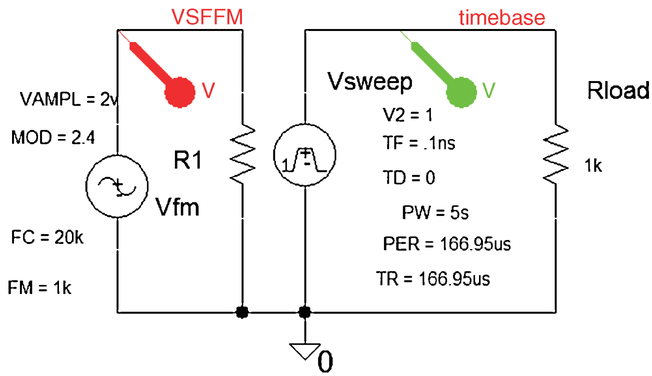


FIGURE 5.10: Multiple display of FM signal

To test the preemphasis circuit, apply two generators with frequencies of 1 kHz and 15 kHz as shown in Fig. 5.14.

From the spectrum in Fig. 5.15, we see the 15-kHz component is now larger than the 1-kHz signal (i.e., it is emphasized). A low-pass filter in the receiver deemphasizes the received signal.

5.5 FM STEREO GENERATION

An FM baseband signal has a bandwidth 50 Hz to 15 kHz, and comprises a left + right signal (***L*** + ***R***) necessary for receiving a complete FM signal by mono receivers, a 19-kHz **pilot** carrier, and the left–right signal (***L*–*R***). Since the $L + R$ and the $L - R$ signals occupy

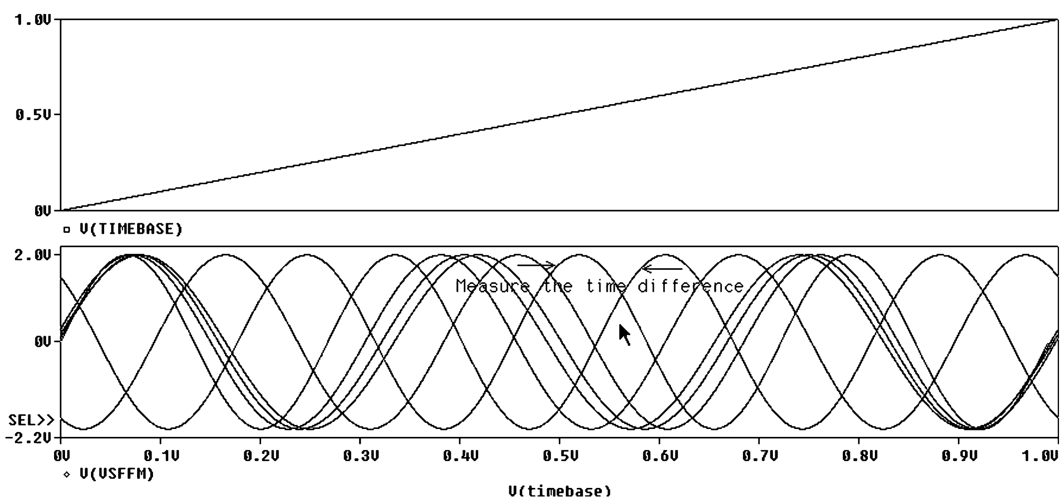


FIGURE 5.11: Change the x-axis to the time-base signal

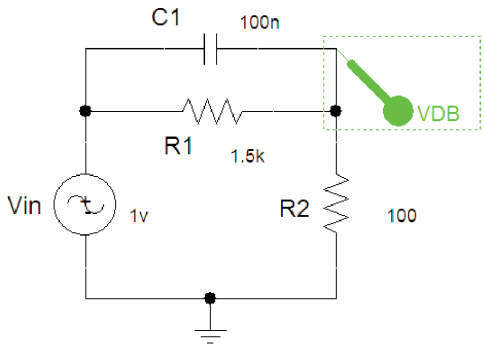


FIGURE 5.12: Preemphasis circuit

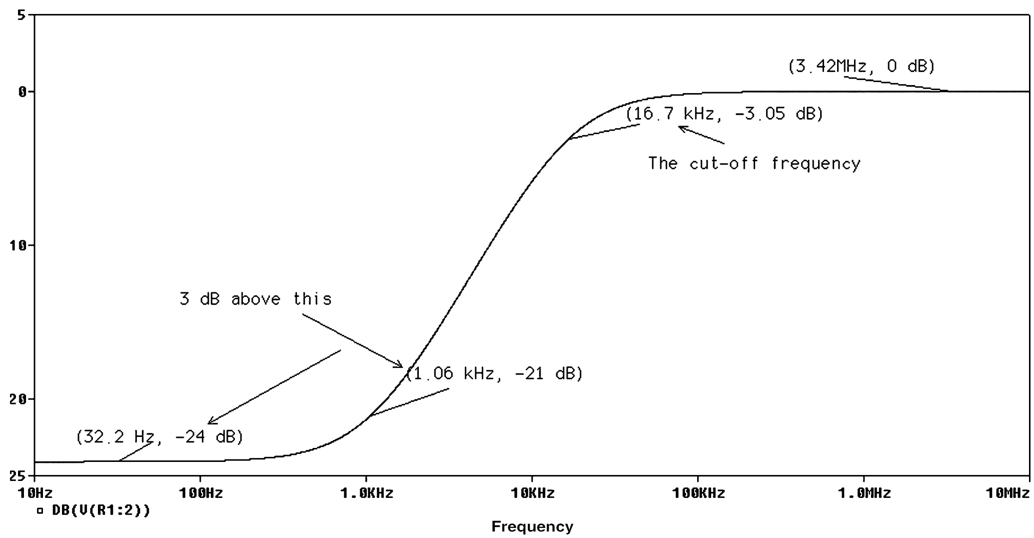


FIGURE 5.13: Amplitude response

the same audio band, we need to separate the two signals using double sideband-suppressed carrier (DSBSC). The balanced modulator shifts the $(L - R)$ signal to a higher frequency band, and produces sidebands centered at 38 kHz, but with the carrier suppressed. The total stereo frequency spectrum occupies a BW from 23 kHz to 53 kHz (actually higher than this because of supplementary information such as traffic and station identification). This necessitates reducing the modulation index to compensate for the increase in baseband bandwidth.

5.5.1 FM Baseband Stereo Signals

Speech files are applied to the two inputs in Fig. 5.16 using the **VPWL_F_RE_FOREVER** part. The two signals in the **signalsources** directory are: “This is the left channel” and the right

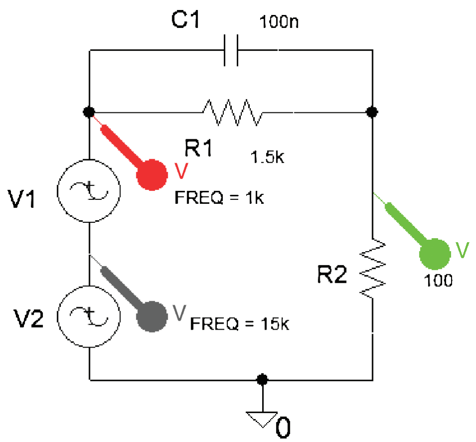


FIGURE 5.14: Two VSIN generators applied to the input

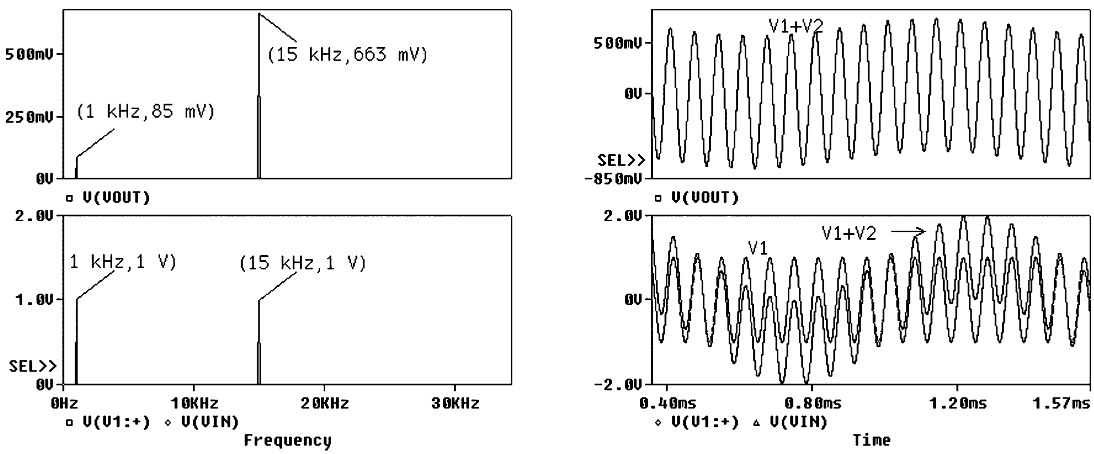


FIGURE 5.15: Input and output signals

input speech signal "This is the right channel." These audio signals from the left and right sources are the connected to a **DIFF** part to produce the L–R signal. The same signals are also fed to a **SUM** part to produce the L + R signal. We need to separate these two signals using a modulator. Balanced modulation uses a **MULT** part with a **VSIN** 38-kHz carrier signal to locate the L–R sidebands at 38 kHz. For easier carrier extraction using a PLL in the receiver, we need to send a pilot tone with the other signals. A 19-kHz pilot tone fits nicely between the lower sideband and the audio signal. The customized three input summer combines the L + R, the L–R, and the 19-kHz carrier signal, but you may use two, two-input **SUM** parts instead.

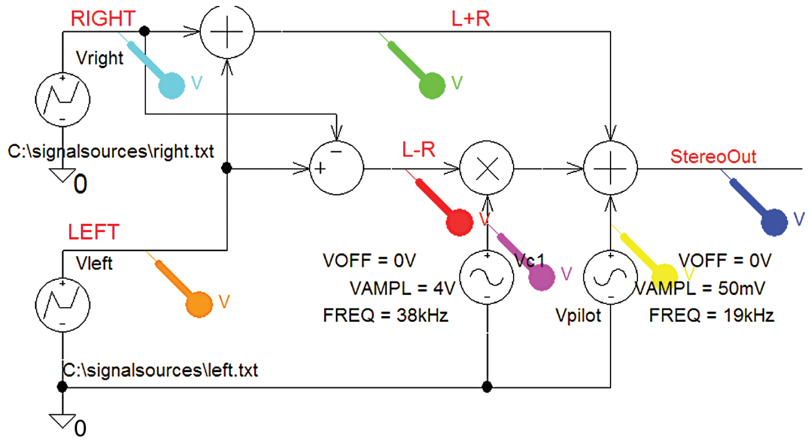


FIGURE 5.16: Stereo FM production

The input ASCII speech files **right.txt** and **left.txt** are created using the m file **right.m** in the miscellaneous directory, or the **Wav2ascii** program.

```
[sound,fsample] = wavread                  % returns samples and the sample rate, and
  ("C:\Pspice\Circuits\                      reads in the speech file sampled
  signalsources\speech\speech.wav");         at fsample = 22050
soundsc(sound,fsample);                    % lets you hear the sound
time = (1:1:length(sound))/fsample;        % Creates time column
time = time';                              % converts the row vector to column form
plot(time,sound)
right = [time sound];                      % creates the time amplitude vector
save C:\Pspice\Circuits\signalsources\
  speech\speech.txt -ascii;                % save file in ASCII with time in first column
```

The composite stereo and input signal spectra are shown in Fig. 5.17.

5.6 REPLACING ABM PARTS WITH CIRCUITRY

Redesign the stereo circuit and replace ABM parts with real circuits. For example, the multiplier, or balanced modulator (BM) could be a four-quadrant multiplier AD633 multiplier. The 38-kHz carrier is produced by applying a 19-kHz to a frequency doubler shown in Fig. 5.18.

The transmitted signal contains a 19-kHz pilot tone for coherent detection in the receiver. Run a transient analysis and use the **FFT** on the output to see if the 19-kHz input signal has been doubled to 38 kHz. The **Run to time** and **Maximum step size** values must be carefully chosen so that the **FFT** has sufficient resolution. Always check the time signals for any triangulation

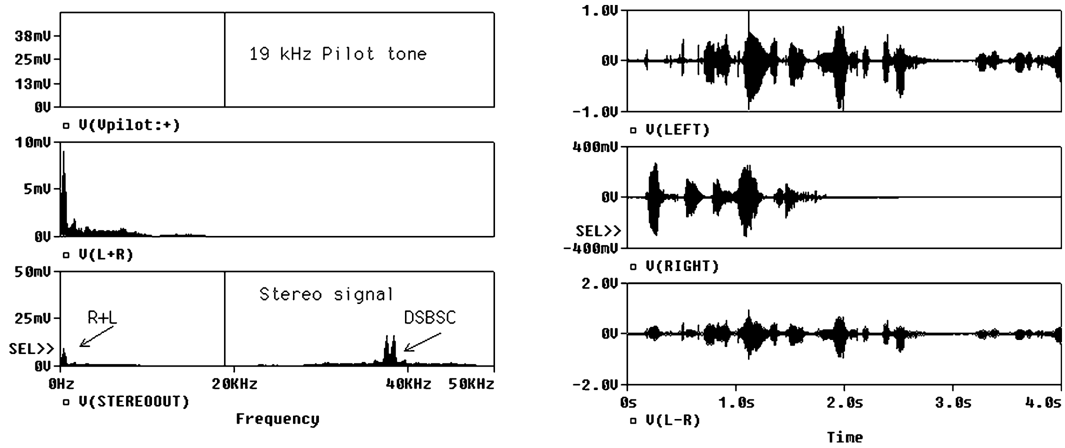


FIGURE 5.17: Spectra of FM stereo signal

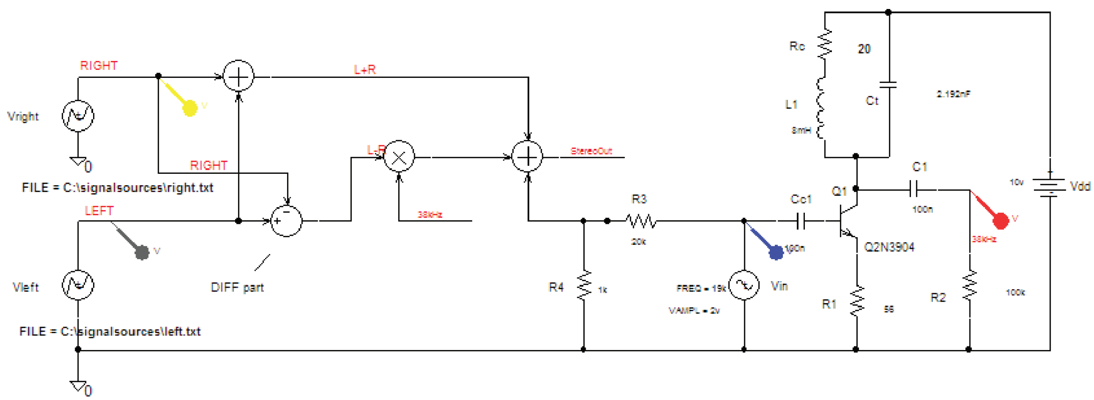


FIGURE 5.18: Stereo generation

present, which is indicative that the **Maximum step size** was too large (not enough points) in the analysis simulation profile and PSpice just "joined the dots." The solution is to decrease the **Maximum step size** parameter, where a good rule of thumb is to set it to one-tenth the smallest event in the observed signal. The potential divider formed from *R*3 and *R*4 attenuates the 19-kHz pilot carrier signal.

5.7 FM STEREO RECEPTION

A simple stereo decoder is shown in Fig. 5.19. Simulate the previous stereo FM signal generator using the left and right input speech signals.

The top **LOPASS** ABM filter extracts the baseband signal and the bottom **BAND-PASS** filter extracts the passband signal. The PLL extracts the pilot 19 kHz and a frequency

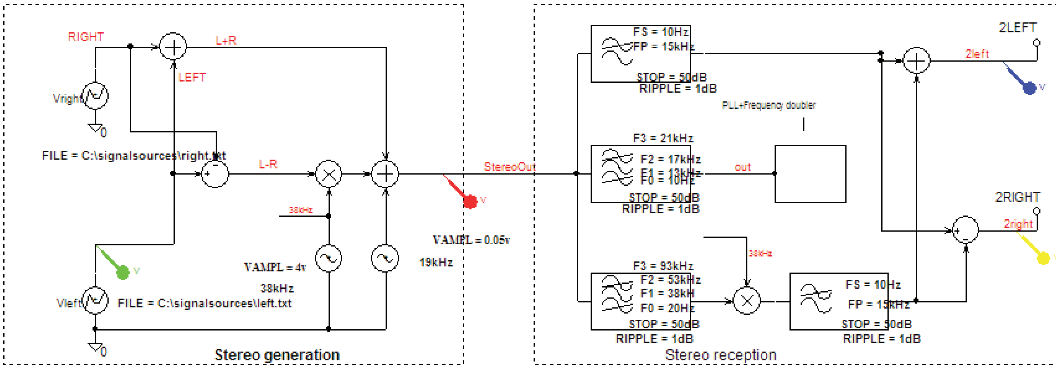


FIGURE 5.19: Stereo transmission and reception

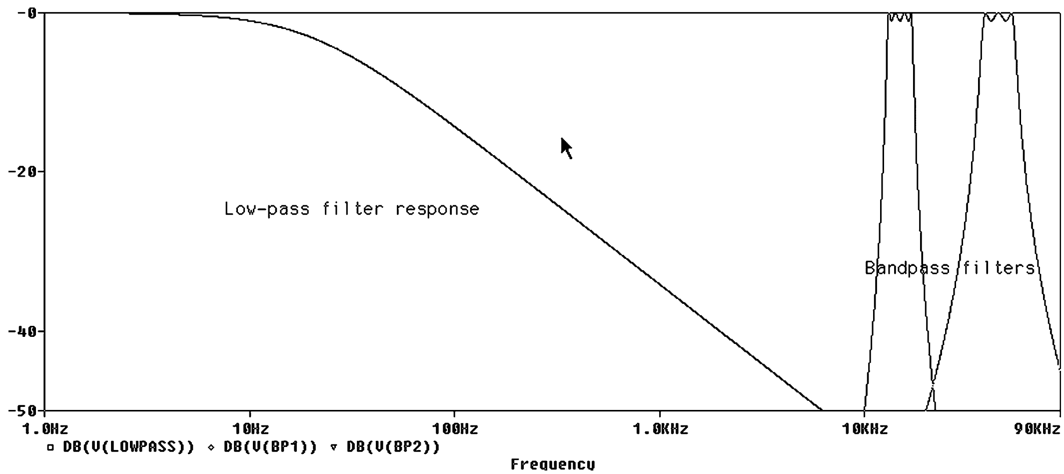


FIGURE 5.20: Frequency response for the three filters

doubler, identical to the previous circuit, produces the local oscillator for the DSBSC balanced modulator. The three filters responses are shown in Fig. 5.20.

Terminate each filter with a wire named using the **Net Alias** icon. The PLL uses the model from the PLL superhetrodyne receiver in the next chapter.

5.8 EXERCISES

(1) Investigate the frequency-modulated square-wave carrier generator using the 555 IC in Fig. 5.21.

(2) Investigate the varactor diode FM VCO in Fig. 5.22. The reverse-biased diode capacitance is varied by the modulating signal. This in turn changes the resonant frequency.

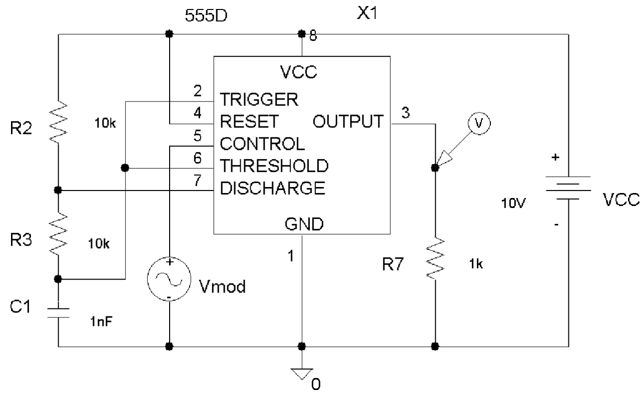


FIGURE 5.21: Frequency modulator

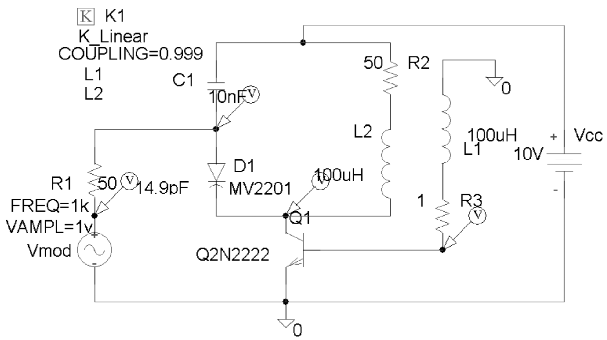


FIGURE 5.22: Varactor diode VCO

(3) The VCO in Fig. 5.23 uses two reverse-biased Varicap diodes and a **Param** part to define the reverse bias voltage name and value. Investigate frequency variation when the voltage is varied.
Open the loop and attach a **vsin** generator to the input to produce the frequency response in Fig. 5.24. Tick **Parametric** and vary rbias from 1 V to 4 V, in steps of 1 V. Selecting the **FFT** icon shows the four carrier frequencies for the swept varactor bias. Apply a modulating **VSIN** source in series with the varactor diode.

(4) Class C amplifiers are very efficient and located in the output stage of mobile transmitters where efficiency is important because we need the batteries to last as long as possible in hand-held equipment such as mobile phones. Fig. 5.25 shows a very basic class C amplifier configured to produce amplitude modulation and the output signals are shown in Fig. 5.26.
The transistor operates more as a switch, and hence, produces much more distortion when compared to a linear power amplifier. The carrier signal biases the transistor

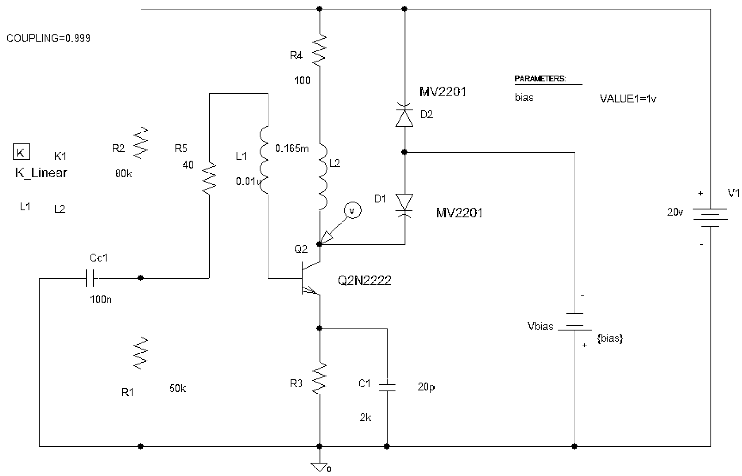


FIGURE 5.23: A tuned-collector VCO

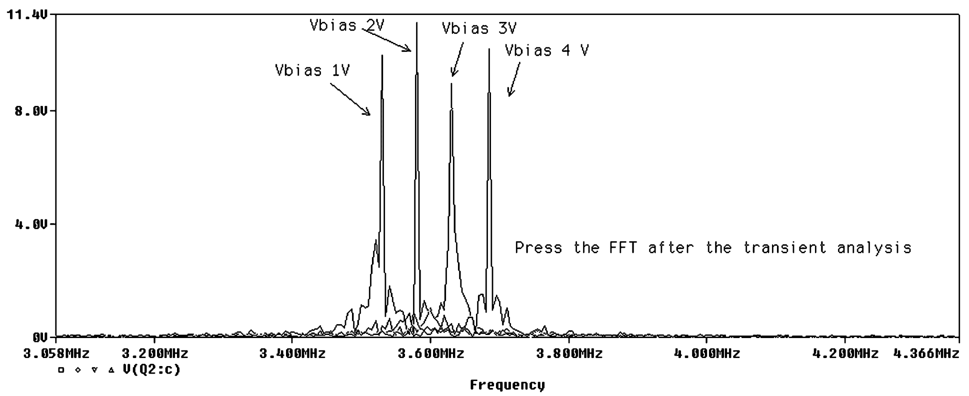


FIGURE 5.24: Testing the varactor diode in a tuned RF amplifier

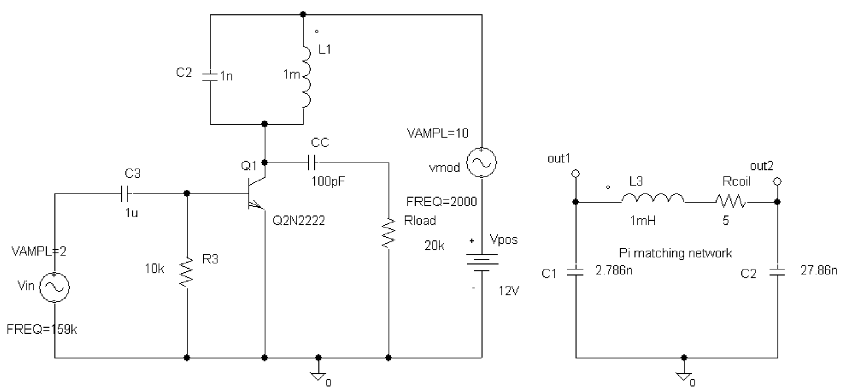


FIGURE 5.25: Class C amplifier and amplitude modulator

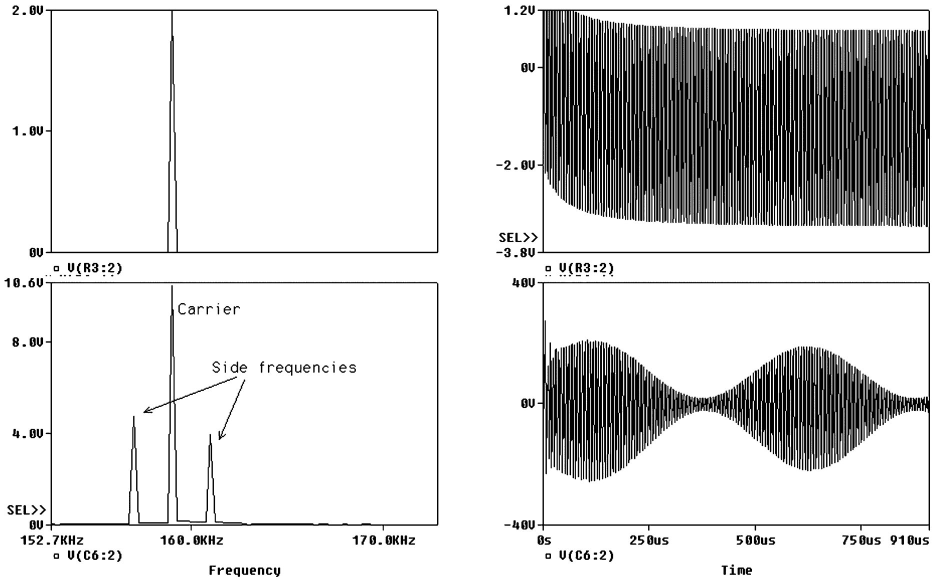


FIGURE 5.26: Class C waveforms

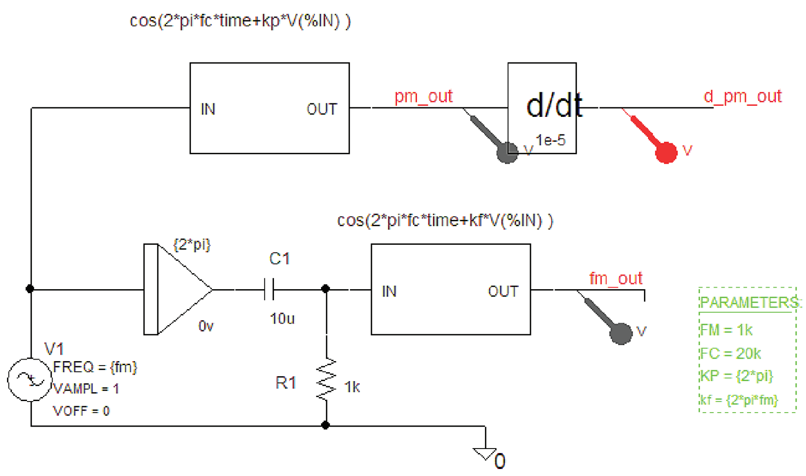


FIGURE 5.27: FM, PM, and AM

because there is no DC bias as required in class A operation. The transistor conducts for less than half a cycle but the distortion is eliminated using a selective network.

(5) Investigate the class E MOSFET transistor amplifier shown in Fig. 5.31, with the output signals displayed in Fig. 5.32.

(6) Fig. 5.27 and Fig. 5.29 are two interesting exercises for investigating FM and PM and AM, where the output signals are displayed in Fig. 5.30.

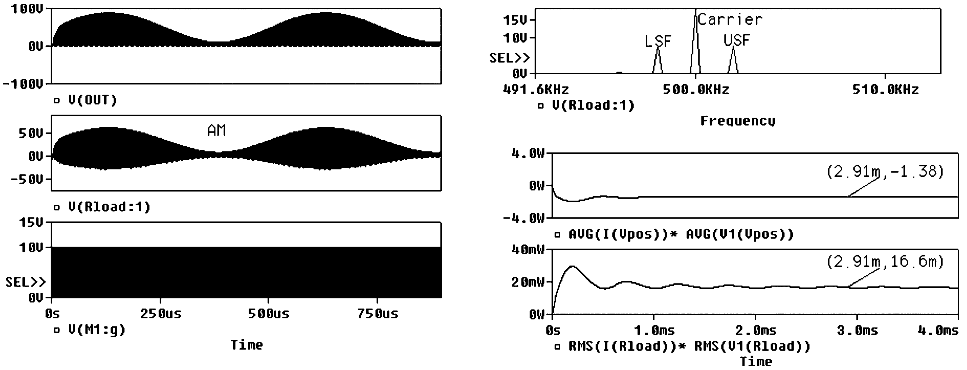


FIGURE 5.28: Output time and frequency domain signals

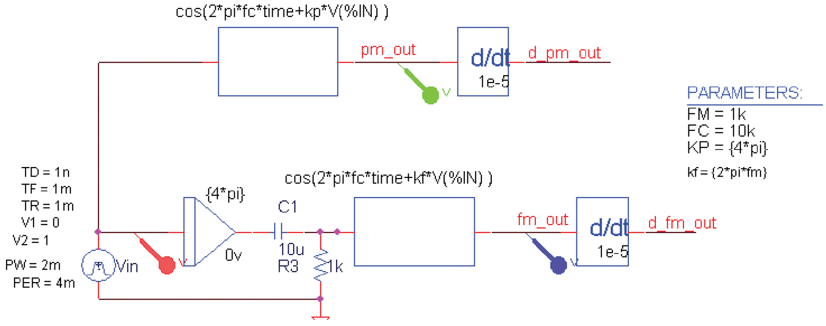


FIGURE 5.29: Developed by Anthony Kelly

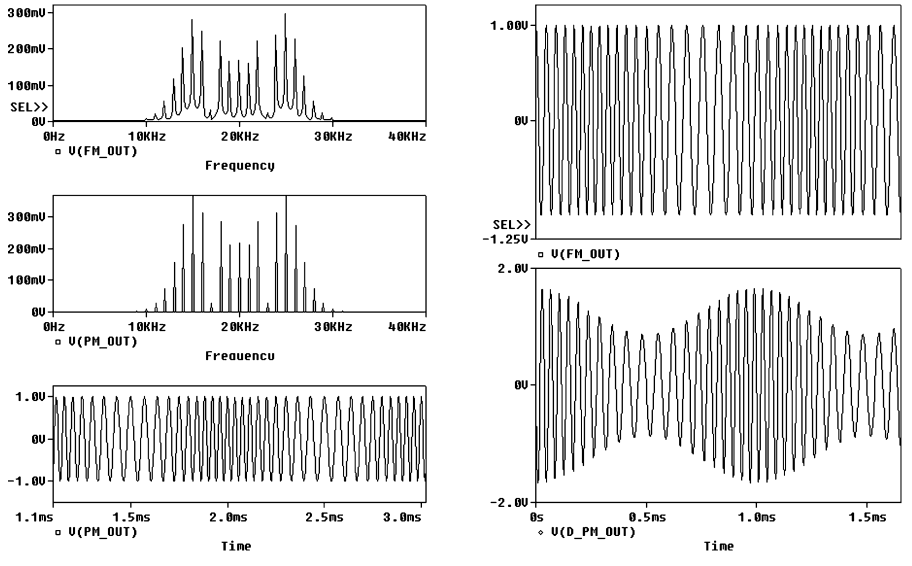


FIGURE 5.30: Signals for the schematic in Fig. 5.29

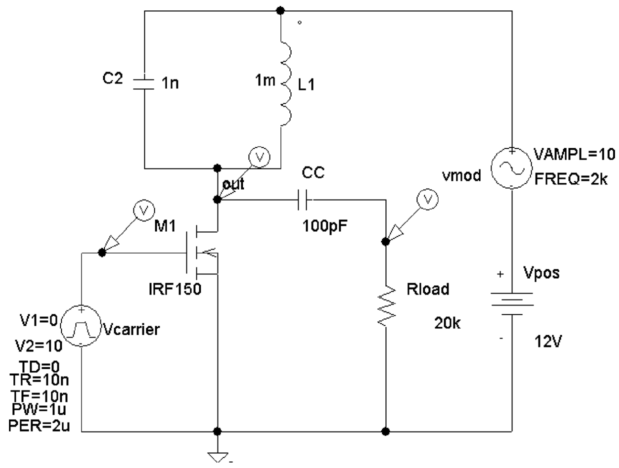


FIGURE 5.31: A class E power amplifier

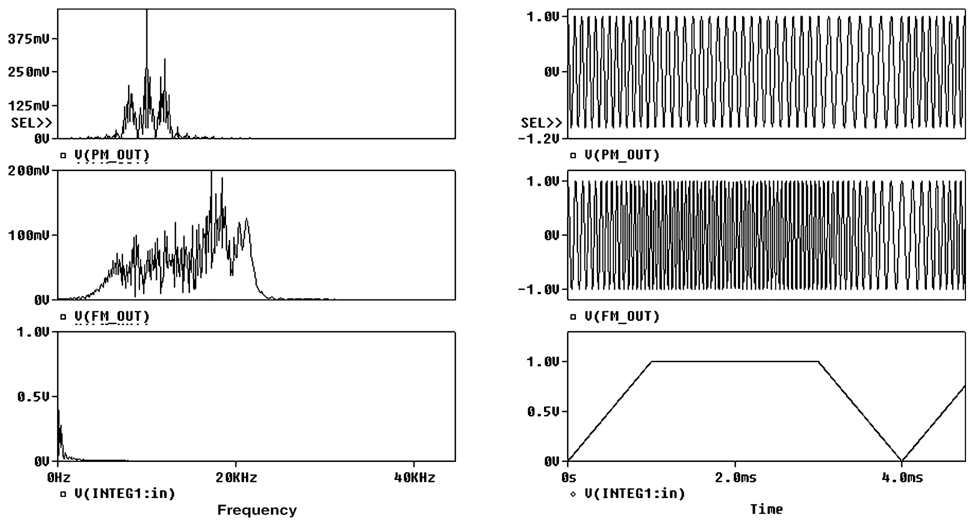


FIGURE 5.32: Class E waveforms

CHAPTER 6

Superhetrodyne Frequency Modulation Receivers

6.1 FM SUPERHETRODYNE RECEIVER

Frequency modulation (FM) signals use the VHF band from 88 MHz to 108 MHz. The principle of FM was created by Edwin Armstrong many years ago but, alas, he didn't get the recognition he deserved. At the FM transmitter, the upper frequency limit of the modulating signal is 15 kHz (CD music quality has a bandwidth of 20 Hz–20 kHz), with a maximum modulation index of 5. From the previous analysis, we have seen how an FM signal has a large number of sidebands and the bandwidth for a modulation index of 5 and a 15-kHz modulating signal bandwidth is calculated using Carson's modified rule as

$$BW = 2f_m(2 + m_f) = 2 \times 15 \times 10^3(2 + 5) = 210 \text{ kHz}. \tag{6.1}$$

Stations are assigned frequency allocations 0.2 MHz apart and will fit 100 stations into the FM bandwidth of 88 MHz to 108 MHz (20 MHz/0.2 MHz = 100 stations), which is about the same as the number of AM stations (107) in the medium wave band. The modulation index has to be reduced because stereo signal generation increases the baseband signal spectrum and it is necessary to compensate for this increase. We now investigate the process of superhetrodyning in an FM receiver. The first stage shown in Fig. 6.1 is an RF amplifier because FM signals are much weaker than AM signals.

Superhetrodyne action is achieved by coupling the tuning capacitor of the tuned circuits in both the local oscillator and the RF amplifier. The signals from both stages are fed to a common source JFET amplifier that acts as a nonlinear additive mixer (a multigate FET device is another solution). The output from the mixer produces a 10.7-MHz IF signal whose bandwidth is 220 kHz. Several IF amplifier stages are needed if good selectivity and large signal amplitude are to be achieved (large signals to produce an acceptable audio output). The final IF stage limits the amplitude of the wanted signal and undesirable AM noise to ensure correct phase lock loop (PLL) operation in the demodulator. The PLL VCO is set to a free-running frequency of 10.7 MHz and the error signal from the phase detector is a voltage v_d, which is

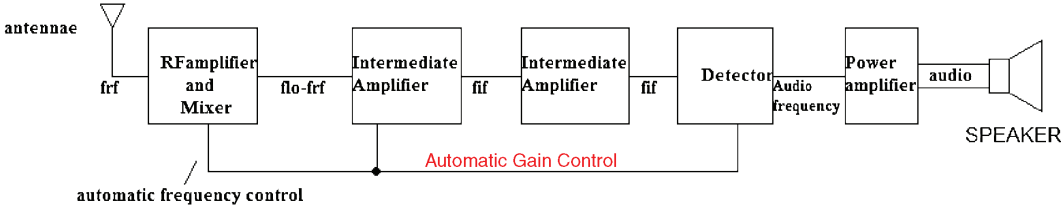


FIGURE 6.1: FM receiver

proportional to the phase difference between the incoming frequency and the VCO frequency. This error voltage is filtered and fed back to the VCO thus forcing it to track the incoming frequency over a limited range. This filtered signal is the modulating audio signal.

To avoid long simulation times, we set the input RF and IF frequencies to a much lower value than that in a real receiver. For example, the IF in this design is 455 kHz but in reality it is 10.7 MHz. An FM signal has a much larger bandwidth than an AM signal and so we must use double-tuned RF amplifiers for a flat response for the wider passband signal. The FM superhetrodyne receiver in Fig. 6.2 has an input FM signal defined in an uncommitted ABM block, where the input 600-kHz carrier has a modulation index = 1.5 and a 20 kHz modulating signal.

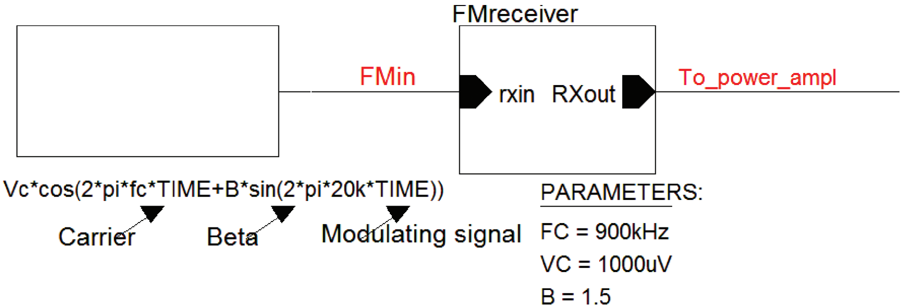


FIGURE 6.2: The FM superhetrodyne receiver

The main sections of an FM receiver are shown in Fig. 6.3.

6.1.1 Mixer Stage

If you are using the evaluation version then you will have to take out the power amplifier stage in order to simulate. The mixer stage in Fig. 6.4 combines the RF and local oscillator signals to produce the IF signal. The mixer was examine in Chapter 4 and the analysis is not repeated here.

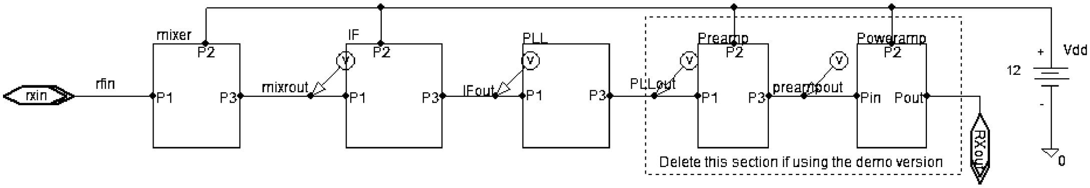


FIGURE 6.3: The individual stages of an FM receiver

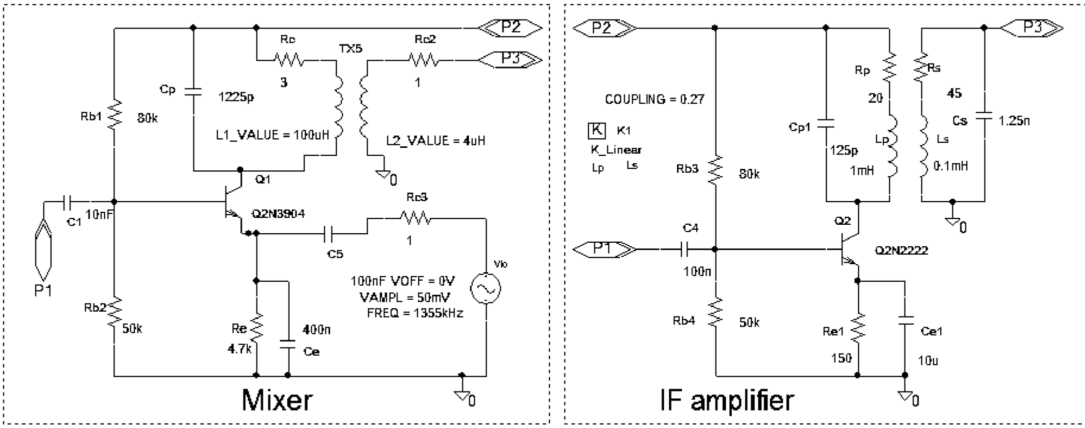


FIGURE 6.4: The mixer, stages first and second IF amplifier

The local oscillator is a **VSIN** part with the frequency **FREQ** = 1055k, **VAMPL** = 5V, and **VOFF** = 0. Let us now examine the individual parts of the FM system before looking at the overall receiver performance.

6.2 COUPLED-TUNED RF AMPLIFIERS

Double-tuned RF amplifiers are necessary in FM because the 220-kHz passband signal is much wider when compared to the 10-kHz AM signal and thus would not be accommodated in a normal single-tuned amplifier. Coupled–tuned RF amplifiers have wider bandwidths compared to single-tuned amplifiers. For example, a typical commercial FM signal has a signal bandwidth of 220 kHz, and thus requires a tuned amplifier with a flat response over this bandwidth. A double-tuned RF amplifier comprises two parallel, critically-coupled tuned circuits, and produces a flatter response centered on the intermediate frequency. In this experiment, we vary the coupling between tuned primary and secondary tuned circuits in order to investigate the effect on the circuit bandwidth. The **Param** part is used to define the coupling between tuned circuits. The schematic in Fig. 6.5 uses the current generator ***IAC*** part with amplitude set to 1 mA and a high-source resistance to avoid loading the tuned circuit.

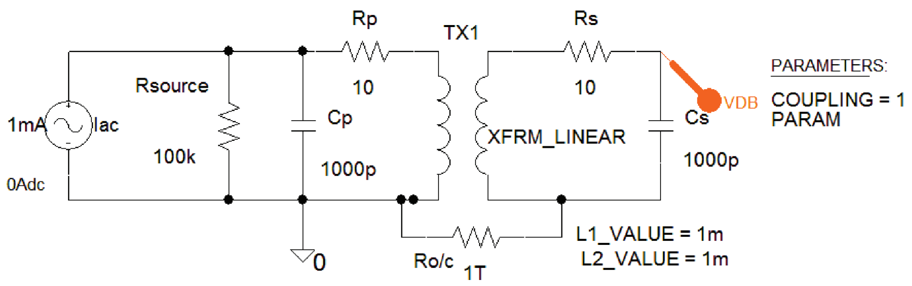


FIGURE 6.5: Varying the coefficient of coupling

DLClick the transformer part name **XFRM_LINEAR**, and associate {**coupling**} with the coupling to be varied. (Make sure to include the brackets {} in the value box.) The relationship between the primary and secondary inductor values, the mutual inductance M, and the coefficient of coupling, k, is

$$k = \frac{M}{\sqrt{L_1 L_2}} = \frac{1}{\sqrt{Q_s Q_p}}. \tag{6.2}$$

Select **Analysis Setup/Parametric,** and set the parameters: Select **Global Parameters** and **coupling** in the name box. **Start Value** = 0, **End Value** = 0.3, and **Points/Octave** = 0.05. Set the **Param** parameters **Rclick** and select **Edit Properties,** and enter **Name** = coupling, and **Value** = 1. From the **Analysis Setup** menu, select **AC Sweep**, and set the parameters: 1000 pts, 100k, 300k, and **linear**. The frequency response for a range of coupling values is shown in Fig. 6.6. Observe the effect of adjusting the coupling to the values: 0, 0.1, and 0.01. Measure the bandwidth for each value of coupling.

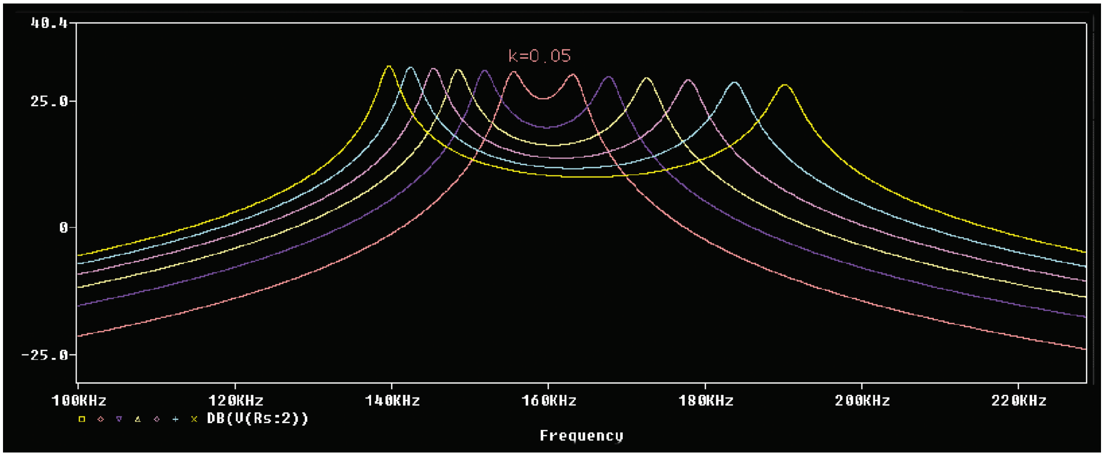


FIGURE 6.6: Frequency response for a range of k values

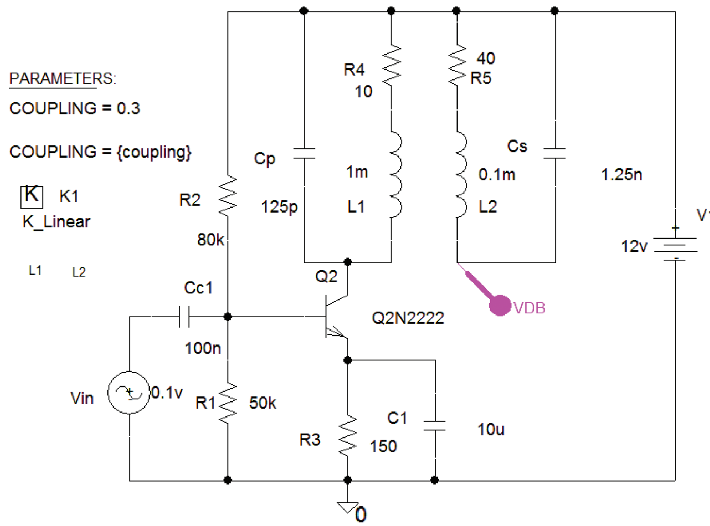


FIGURE 6.7: Double-tuned RF amplifier

The peaks are located at frequencies defined as

$$f = f_o/\sqrt{(1 \pm k)}. \tag{6.3}$$

6.2.1 Double-Tuned Intermediate RF Amplifier

Draw the double-tuned RF amplifier in Fig. 6.7. The **Param** part defines the coupling between the tuned primary circuit and the tuned secondary circuit.

The coupling between the inductors is related to the dot convention, and the direction in which the inductors are placed on the schematic. The mutual coupling between two or more inductors is set in the **K_linear** part which is in the **analog.olb**. **DLClick** the boxed red "**K**" part of **K_Linear** transformer and enter name L1, L2, and {**coupling**}. Be sure to include the brackets {}. **Rclick** the **K_linear** part, select **Edit Properties** and enter L1 and L2 in the **A** column and set **COUPLING** to {**coupling**}. From the **Analysis Setup** menu, tick **AC sweep** and **Parametric Sweep**. Select in the **Sweep variable box Global Parameter**, **Name** = **coupling,** with **Start Value** = **0**, **End Value** = 0.3, **Linear** and **Increment** = 0.05. Click on **AC Sweep** and set the parameters to 1000, 100k, 700k, and linear. Critical coupling produces the desired almost flat response as shown in Fig. 6.8.

To find the coupling coefficient associated with that of the responses, select one of the symbols to the left of the **VdB(R6:2)** and the **Section Information** will appear. This shows the coupling value associated with a desired plot, thus enabling us to set the coupling to this

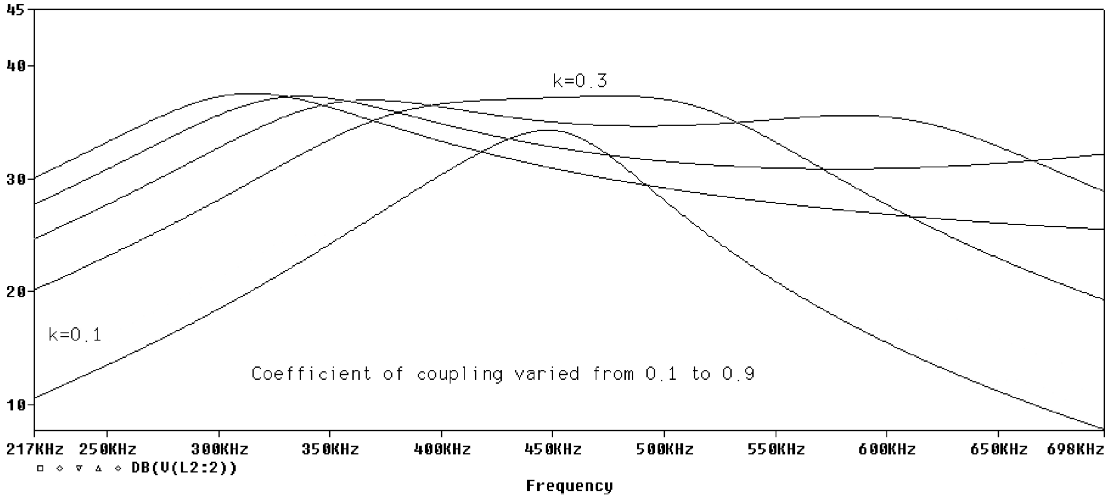


FIGURE 6.8: Double-tuned RF amplifier response

value. This "what if I set this value what happens" design approach is not the best but is useful at times.

6.3 AUTOMATIC GAIN CONTROL

The schematic in Fig. 6.9 investigates automatic gain control (AGC) derived from the demodulator to stabilize the receiver gain against variations in the received input carrier amplitude. Carrier input voltage amplitude variation changes because of transmission propagation variations and we simulate this effect by varying the battery voltage vagc for a range of values.

The AGC voltage, derived from the demodulator output, is fed back to the IF amplifier bias circuit to control the amplifier gain. The common-emitter amplifier voltage gain is proportional to the collector current and is related to the base current and emitter-base voltage as

$$|A_v| = \frac{I_c R_L}{26 \text{ mV}}. \tag{6.4}$$

The AGC voltage is swept for a range of voltages, by selecting **Parametric Sweep** from the **Analysis**. Select **Global Parameter** and in the **Parameter name** and enter vagc. Set **Start Value** = 0, **End Value** = 10, **Points/Octave** = 0.5. Fig. 6.10 shows the effect of the AGC voltage the RF amplifier gain.

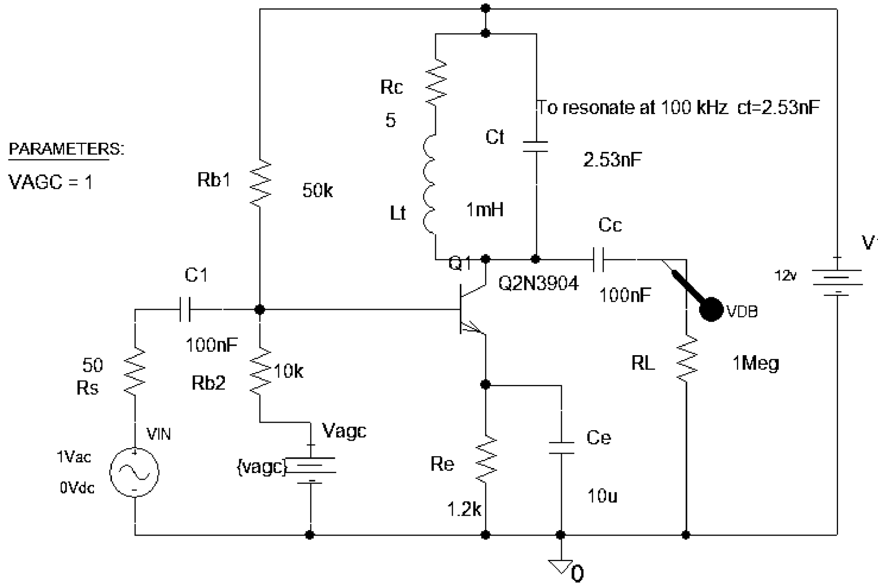


FIGURE 6.9: AGC voltage connected back to the input

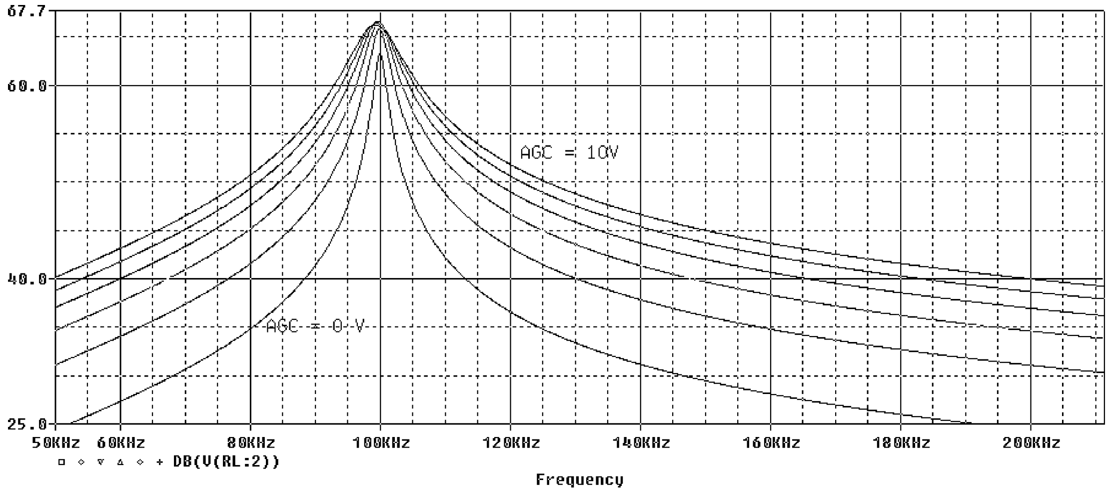


FIGURE 6.10: Frequency response applying AGC

6.4 THE PHASE-LOCK LOOP DETECTOR

Fig. 6.11 shows the major parts of a phase-lock loop, one of the most important devices in electronic systems.

The heart of the phase-lock loop (PLL) is the voltage-controlled oscillator (VCO) with a linear, limited frequency range, and a free-running frequency f_o. (The frequency it oscillates

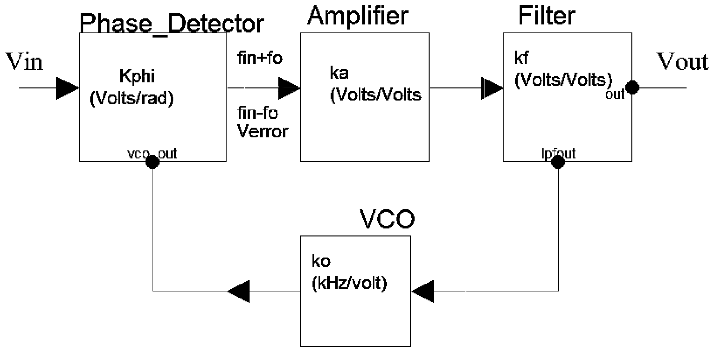


FIGURE 6.11: PLL block diagram

at without an external input signal.) The phase detector compares the input signal with the VCO signal and produces an error voltage that is low-pass filtered before being fed back to the VCO input. The filtered error signal forces the VCO to minimize the difference between the free-running VCO frequency and the input frequency. To model a PLL we derive an expression for the transfer function relating the output voltage to a change in input phase, i.e., V_o/θ_i. From the definition of radians:

$$\omega_i = \frac{d\theta}{dt} \quad \text{or} \quad \omega_i = s\theta. \tag{6.5}$$

Rearranging this equation in terms of frequency yields

$$f_i = \frac{1}{2\pi}\frac{d\theta_i}{dt} = \frac{1}{2\pi}s\theta_i \Rightarrow \theta_i = \frac{2\pi f_i}{s}. \tag{6.6}$$

The phase detector produces an error phase equal to the difference between the input and output phase as

$$\theta_i - \theta_o = \theta_\varepsilon \Rightarrow \theta_i = \theta_\varepsilon + \theta_o. \tag{6.7}$$

Determine expressions for phase quantities in (6.7) in terms of the output voltage. The VCO output phase is

$$\omega_o = 2\pi f_o = \frac{d\theta_o}{dt} = s\theta_o \Rightarrow \theta_o = \frac{2\pi f_o}{s}. \tag{6.8}$$

Substituting for f_o from the VCO modulator sensitivity $k_o = \frac{f_o}{V_o} \Rightarrow f_o = k_o V_o$, hence

$$\theta_o = \frac{2\pi k_o V_o}{s}. \tag{6.9}$$

An expression for the error phase is obtained by considering the phase detector parameter:

$$k_\theta = \frac{V_\varepsilon}{\theta_\varepsilon} = \frac{V_o/k_a k_f}{\theta_\varepsilon} \Rightarrow \theta_\varepsilon = \frac{V_o}{k_\theta k_a k_f}. \tag{6.10}$$

The input phase is

$$\theta_i = \left[\frac{V_o}{k_\theta k_a k_f} + \frac{2\pi k_o V_o}{s}\right] = V_o\left[\frac{1}{k_\theta k_a k_f} + \frac{2\pi k_o}{s}\right]. \tag{6.11}$$

Rearrange (6.11) to give the transfer function:

$$\frac{V_o}{\theta_i} = \frac{1}{1/k_\theta k_a k_f + 2\pi k_o/s} = \frac{s k_\theta k_a k_f}{s + 2\pi k_\theta k_a k_f k_o} = \frac{s k_\theta k_a k_f}{s + k_v}. \tag{6.12}$$

The transfer function has a pole at $s = -2\pi k_\theta k_a k_f k_o = -k_v$. Multiply the loop gain by 2π to express it in dB, i.e., $2\pi k_\theta k_a k_f k_o = k_v$. An input step-change in phase tracks out to zero in steady state, similar to a step change voltage applied to a high-pass *CR* filter. For frequency changes, the PLL produces a response that is similar to a low-pass filter. The relationship between phase and frequency is $\theta_i = 2\pi f_i/s$, so the transfer function becomes

$$\frac{V_o}{f_i} = \frac{2\pi k_\theta k_a k_f}{s + k_v}. \tag{6.13}$$

An input signal step change in frequency is $F_i(s) = \Delta f_i/s$, so the transfer function is

$$\frac{V_o}{\Delta f_i} = \frac{k_v/k_o}{s(s + k_v)}. \tag{6.14}$$

Express in terms of the output voltage as

$$V_o(s) = \frac{\Delta f_i k_v/k_o}{s(s + k_v)}. \tag{6.15}$$

Applying the inverse Laplace transform to (6.15) yields the output voltage in time. The output voltage may be compared to the step response for a low-pass CR filter. Use partial fraction expansion and Laplace tables (Appendix A) to write the output voltage as

$$v_o(t) = \Delta f_i/k_o(1 - e^{-k_v t})\ \mathrm{V}. \tag{6.16}$$

6.4.1 PLL Compensation

To speed up the PLL response to signal changes, add a low-pass filter to the loop to produce a compensated PLL (ref: 6, Appendix A). Substitute the filter transfer function $k_f = \omega_C/(s + \omega_C)$ into (6.15) to yield the transfer function:

$$\frac{V_o}{\Delta f_i} = \frac{2\pi k_\theta k_a \omega_C/(s + \omega_C)}{s(s + 2\pi k_\theta k_a \omega_C/(s + \omega_C)k_o)} = \frac{k_v/k_o \omega_C}{s(s^2 + s\omega_C + k_v\omega_C)}. \tag{6.17}$$

From the Laplace tables, write the output voltage as

$$v_o(t) = \frac{\Delta f_i}{k_o}\left[\frac{1 - e^{-\delta\omega_n t}}{\sqrt{1-\delta^2}}\sin\left(\omega_n t\sqrt{1-\delta^2}+\theta\right)\right]. \tag{6.18}$$

By comparing equations, we have the following relationships for lag compensation as

$$\omega_n^2 = k_v \omega_c . \tag{6.19}$$

The damping factor is defined as

$$\delta = \frac{\omega_c}{2\omega_n} = \frac{\omega_c}{2\sqrt{k_v\omega_c}} = \frac{1}{2}\sqrt{\frac{\omega_c}{k_v}}. \tag{6.20}$$

The modulator and phase detector sensitivities are $k_o = -0.75$ kHz V^{-1}, $k_\theta = 0.318$ V rad^{-1}, $k_a = 1$ V V^{-1}. The 565 free-running frequency timing components, $R_o = 10$ kΩ, $C_o = 220$ pF sets the frequency to

$$f_o = \frac{1}{3.7 \times R_0 C_0} = \frac{1}{3.7 \times 10^4 \times 220 \times 10^{-12}} = 136 \text{ kHz}. \tag{6.21}$$

The loop gain is dependent on the supply voltage V, and is calculated as

$$k_v = k_\theta k_0 = 2\pi \frac{0.318 \text{ V}}{\text{rad}}\frac{750 \text{ Hz}}{\text{V}} = 1495 \text{ s}^{-1} = \frac{33.6 f_o}{V}. \tag{6.22}$$

The lock range (hold-in range) is

$$f_L = \pm\frac{8 f_o}{V} = \frac{8 \times 136 \times 10^3}{5} = \pm\, 181 \text{ kHz}. \tag{6.23}$$

The filter resistor $R_f = 3.6$ kΩ is in the 565 PLL, so for a filter capacitance $C_f = 330$ pF, the natural bandwidth is

$$f_c = \frac{1}{2\pi}\sqrt{\frac{2\pi f_L}{R_f C_f}} = \frac{1}{2\pi}\sqrt{\frac{2\pi 181 \times 10^3}{3.6 \times 10^3 \times 330 \times 10^{-12}}} = 156 \text{ kHz}. \tag{6.24}$$

The loop gain affects the damping factor of the second-order phase model. Higher damping results in a quicker convergence to phase lock in the linear operating range, but decreases the lock range:

$$\delta = \frac{1}{2}\sqrt{\frac{\omega_c}{k_v}} = \frac{1}{2}\sqrt{\frac{1}{k_v C_f R_f}} = \frac{1}{2}\sqrt{\frac{1}{3600 \times 330 \times 10^{-12} \times 0.75 \times 10^3 \times 0.318 \times 2\pi}}$$
$$= 0.8622. \tag{6.25}$$

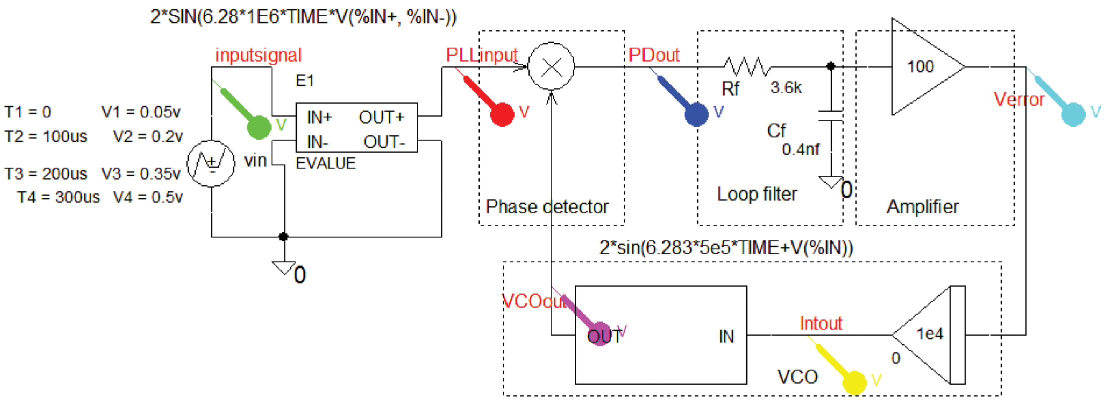


FIGURE 6.12: ABM PLL model

6.4.2 The Lock and Capture Range

To obtain the lock and capture range, determine suitable components so the VCO free-runs at 100 kHz. Connect a scope and sketch the VCO signal. With the signal generator set at 500 mV, increase the frequency from zero until you notice that lock is achieved (monitor the input signal as well). Note the frequency where lock is achieved. Increase the signal frequency further until the PLL goes out of lock. Note this frequency. From a very high frequency, repeat the above procedure but in the opposite direction, i.e., sweep from a high frequency and note the frequency where the PLL captures the signal.

6.4.3 ABM Phase-Locked Loop

A PLL contains three major parts: a voltage-controlled oscillator (VCO), a phase detector (PD), and a low-pass filter. The PD compares the output of the VCO to the input signal and produces an error signal that is filtered and then applied to the VCO to control its frequency, thus forming a feedback control loop. The **EVALUE** part in Fig. 6.12 creates an FM-type signal whose input is swept by a **VPWL** part with parameters as shown. The **TIME** parameter is the PSpice A/D internal sweep variable used in transient analysis. The VCO comprises two **ABM** parts: **ABM1** and an **INTEG** part to generate a sinusoidal function. An additional function controls the phase as a function of the input voltage **V(%IN)**. However, we will improve on this model in the next section and eliminate the integrator.

Set the **Analysis** tab to **Analysis type: Time Domain (Transient)**, **Run to time** = 300 μs, and **Maximum Step size** = 0.01μs, **Output File Options/Print values in the output file** = 200 μs. Press **F11** to display the swept input signal as shown in the top diagram of Fig. 6.13.

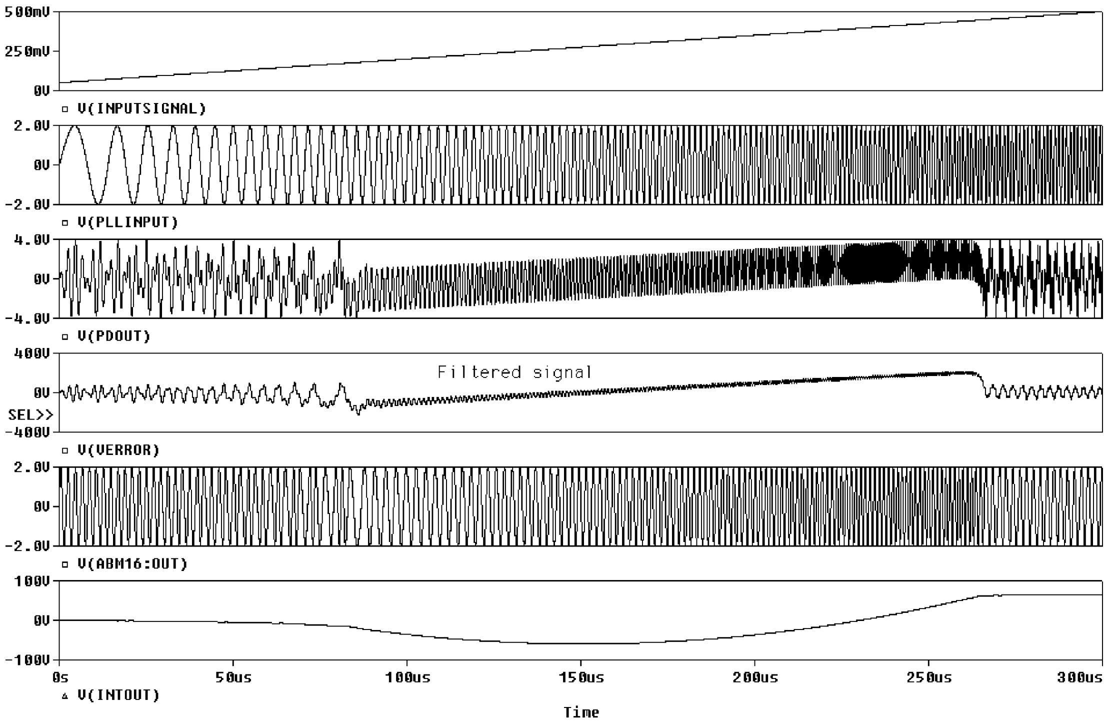


FIGURE 6.13: PLL time signals

The error signal in the middle diagram in Fig. 6.14 forces the VCO frequency to track the input signal. Press the **FFT** icon to obtain the frequency spectra as shown.

6.5 FREQUENCY DEMODULATION

Set the PLL free-running VCO frequency to the input FM carrier frequency. The error signal from the phase detector, when filtered, is the original modulating signal. The *FM* demodulation schematic uses the *ABM* model created previously. Create a symbol for the **ABM** PLL and include it in the schematic in Fig. 6.15. An alternative is to use the modular construction block.

Set the FM **VSFFM** generator part parameters as shown. The contents of the PLL block are shown in Fig. 6.16.

Set the **Analysis** tab to **Analysis type: Time Domain (Transient)**, **Run to time** = 10 ms, and **Maximum Step size** = 0.01 µs. A long **Run to time** ensures good **FFT** frequency resolution. The recovered 2-kHz modulating signal in Fig. 6.17 is distorted with the distortion harmonics shown in the spectrum.

From the **Probe** screen, select the **FFT** icon to observe the demodulator signal spectra shown in Fig. 6.18.

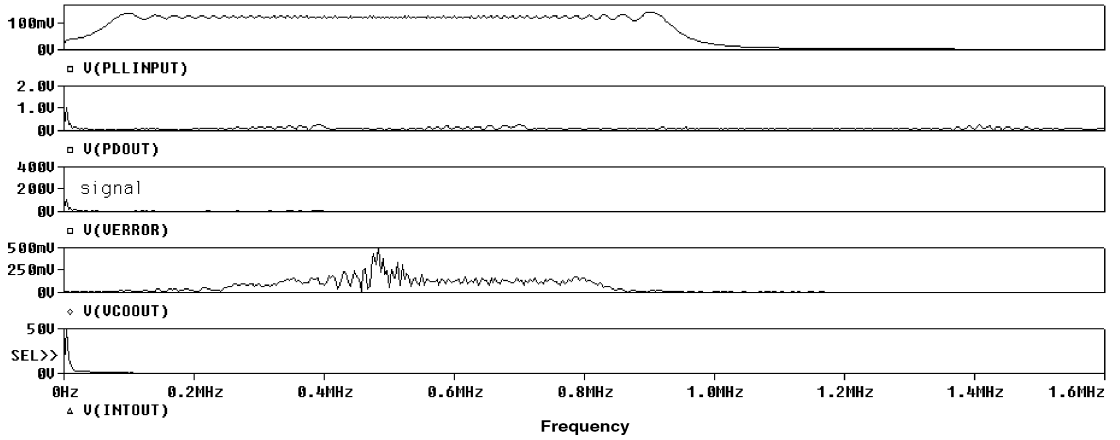


FIGURE 6.14: Spectrum of PLL signals

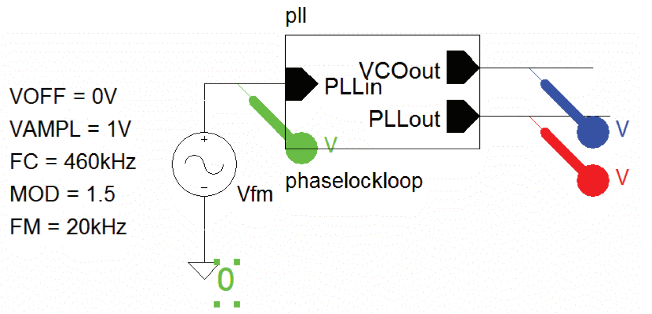


FIGURE 6.15: The PLL using modular construction

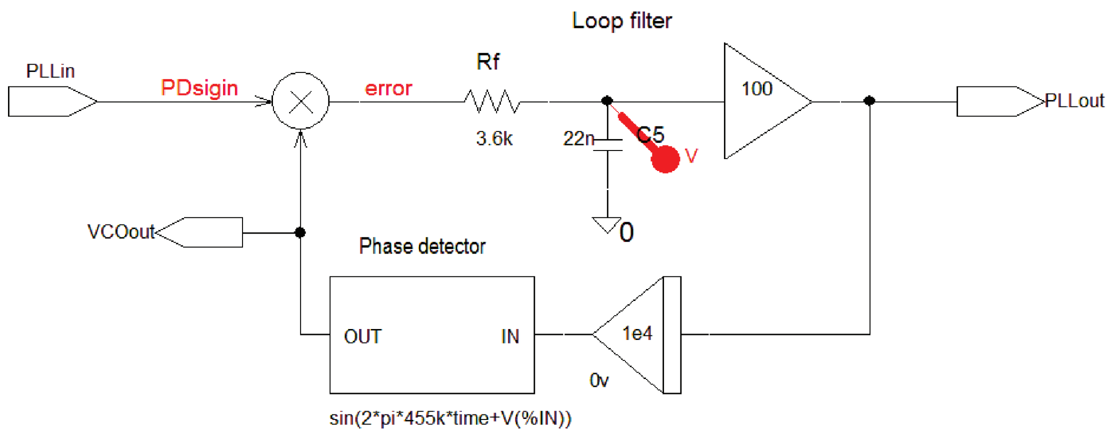


FIGURE 6.16: The contents of the PLL

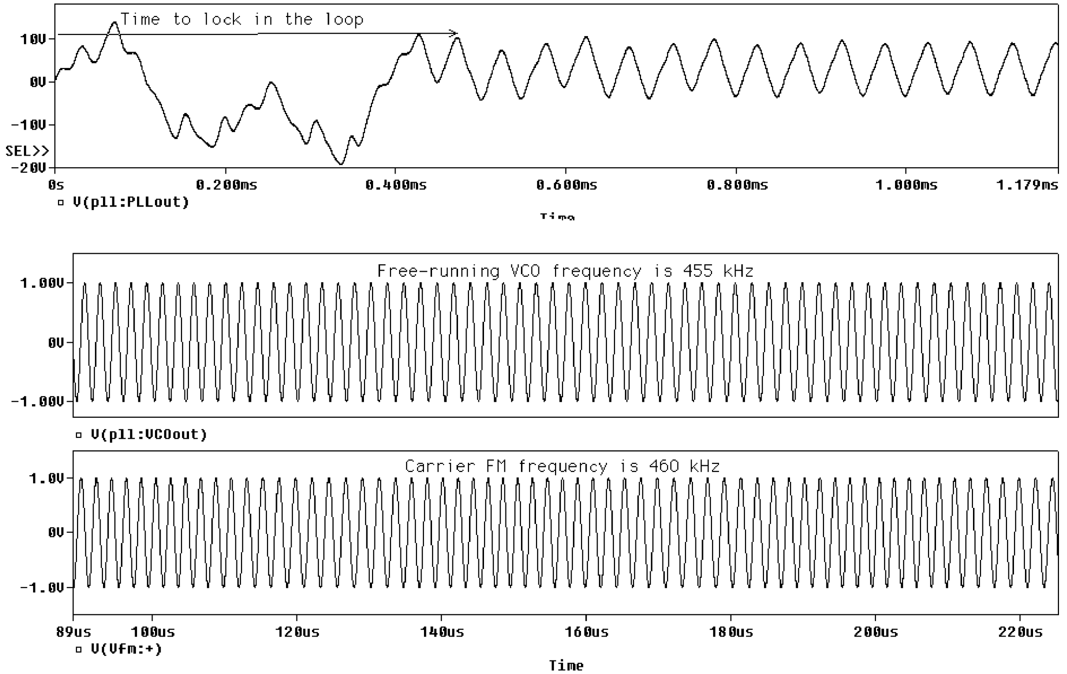


FIGURE 6.17: The demodulated signal

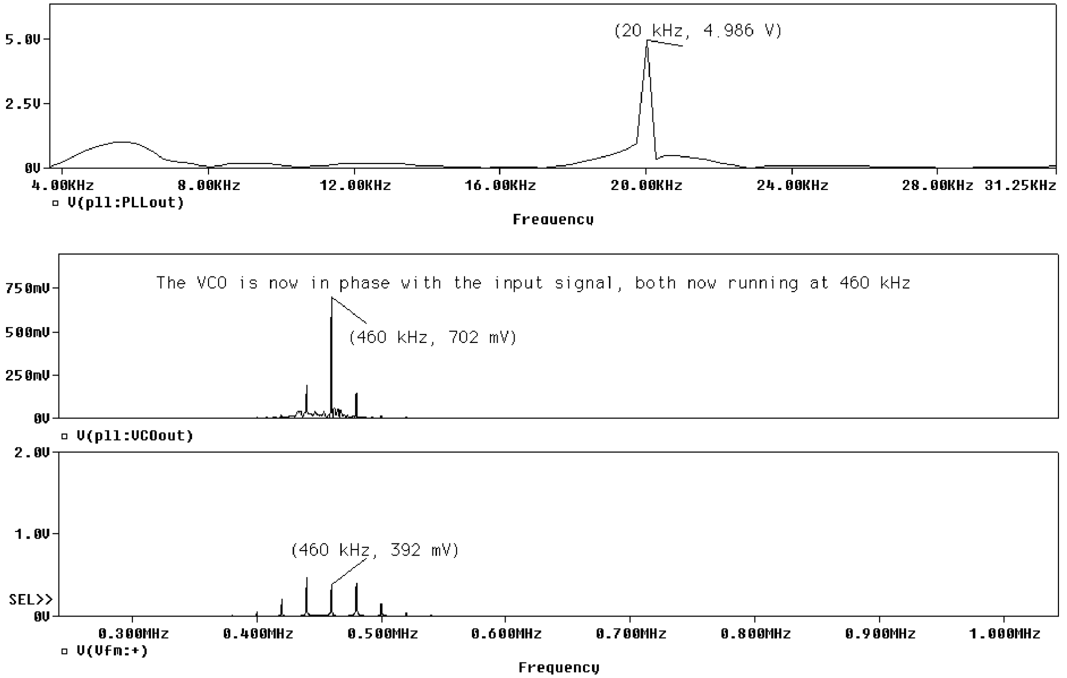


FIGURE 6.18: Waveforms in the frequency domain

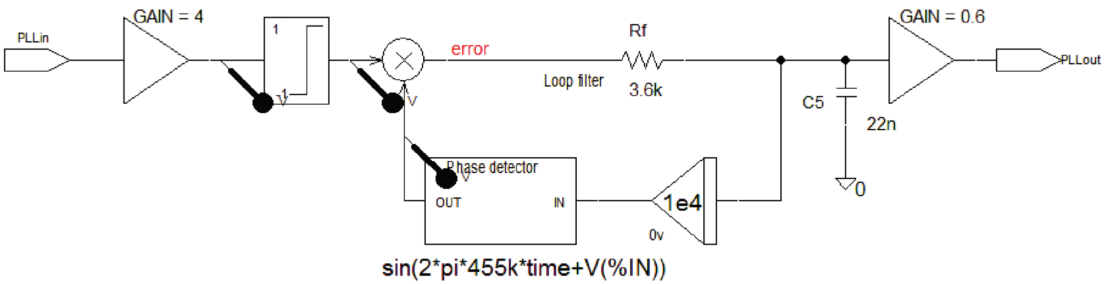


FIGURE 6.19: The FM detector

The ABM PLL detector circuit in Fig. 6.19 is the same as that investigated previously but is preceded by a **Limit** part to eliminate any amplitude variations in the RF signal. The **Gain** part behaves like a volume control and sets the signal level before the preamplifier.

Transient analysis parameters: **Run to time** = 2 ms, and **Maximum step size** = 0.01 μs. Press **F11** to simulate.

6.5.1 Receiver Waveforms

Now that we have familiarized ourselves with the inside parts of the FM receiver (the mixer was examined in Chapters 4), we are in a position to simulate the complete receiver. Set the **Analysis** tab to **Analysis type: Time Domain (Transient), Run to time** = 2 ms, and **Maximum Step size** = 10 μs and press **F11** to simulate. The receiver waveforms are shown in Fig. 6.20.

From the **Probe/File/Run** menu, select the log command **Fig.6-020.cmd**. This separates the signals into two windows: RF signals and audio signals. Note the AM signal in the IF output. Place a marker on the output of the limiter in the detector block. Superhetrodyne action is clearly demonstrated in Fig. 6.20 where the 600-kHz RF input signal is shifted down to the IF of 455 kHz. Of course, in a real FM receiver, the IF is 10.7 MHz but this smaller value speeds up simulation times. The audio signals from the power amplifier are displayed in Fig. 6.21.

Inject a second input FM signal (place the two **VFFM** generators in series), but at a different frequency, and test the receiver adjacent signal rejection properties. Good **FFT** resolution requires a large **Run to time** but increases the simulation time.

6.6 EXERCISES

(1) Fig. 6.22 shows a phase detector implemented with an XOR, and the VCO is a 555 IC whose clock frequency is

$$f_o = \frac{1.44}{(R_1 + 2R_2)C_1}. \tag{6.26}$$

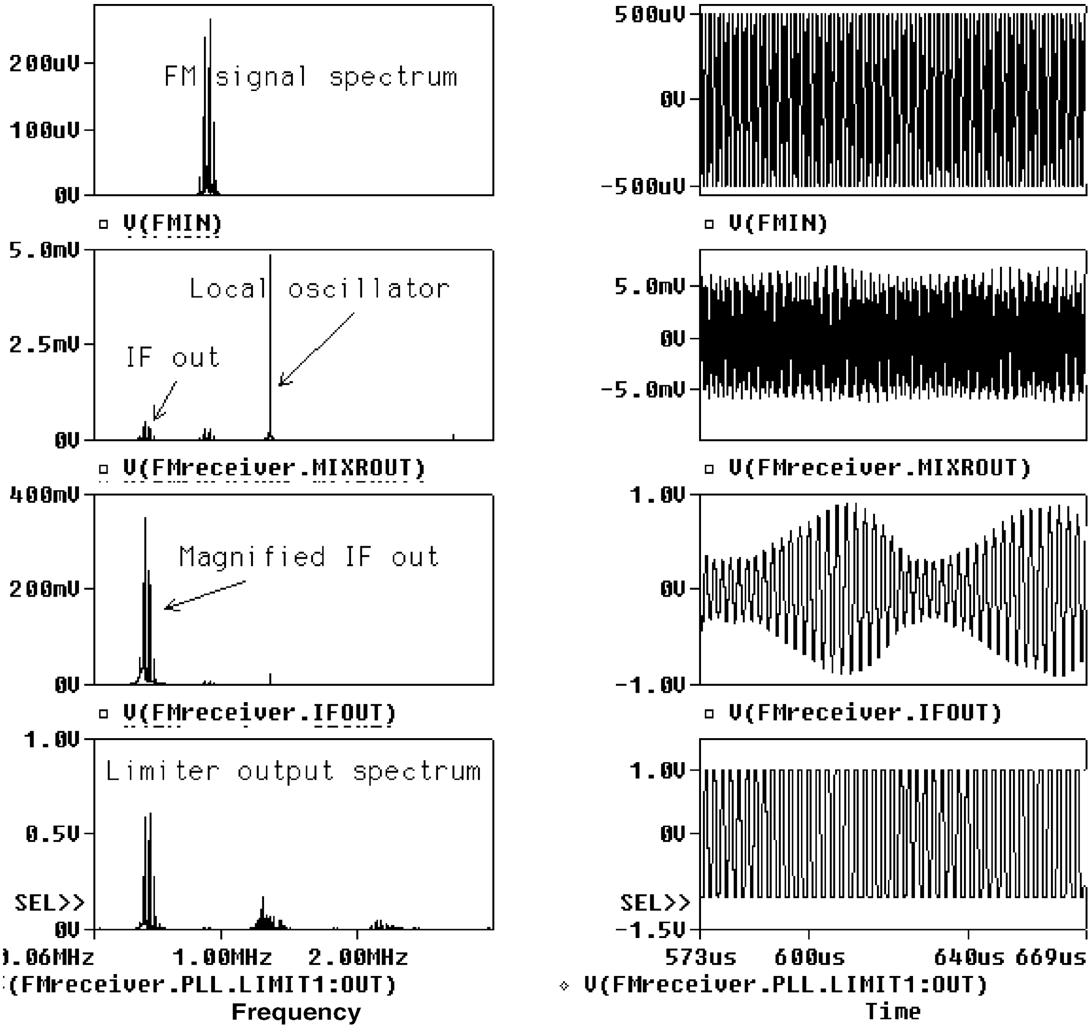


FIGURE 6.20: FM receiver waveforms

A 50% duty cycle is achieved when $R2 \gg R1$, with the mark–space ratio set as $[(R1 + R2)/(R1 + 2R2)]*100$.

The transient PLL waveforms are shown in Fig. 6.23.

(2) Apply a step voltage to the ABM PLL shown in Fig. 6.24 to investigate the closed-loop frequency response. Set the transient analysis parameters.

Simulate with **F11**, and select the **FFT** icon to produce the frequency response shown in Fig. 6.25. The frequency range from 0 to 1/**Run to time** is incorrect. Change the x-axis to log, and the first time value to 10 Hz, if the **Run to time** = 100 ms.

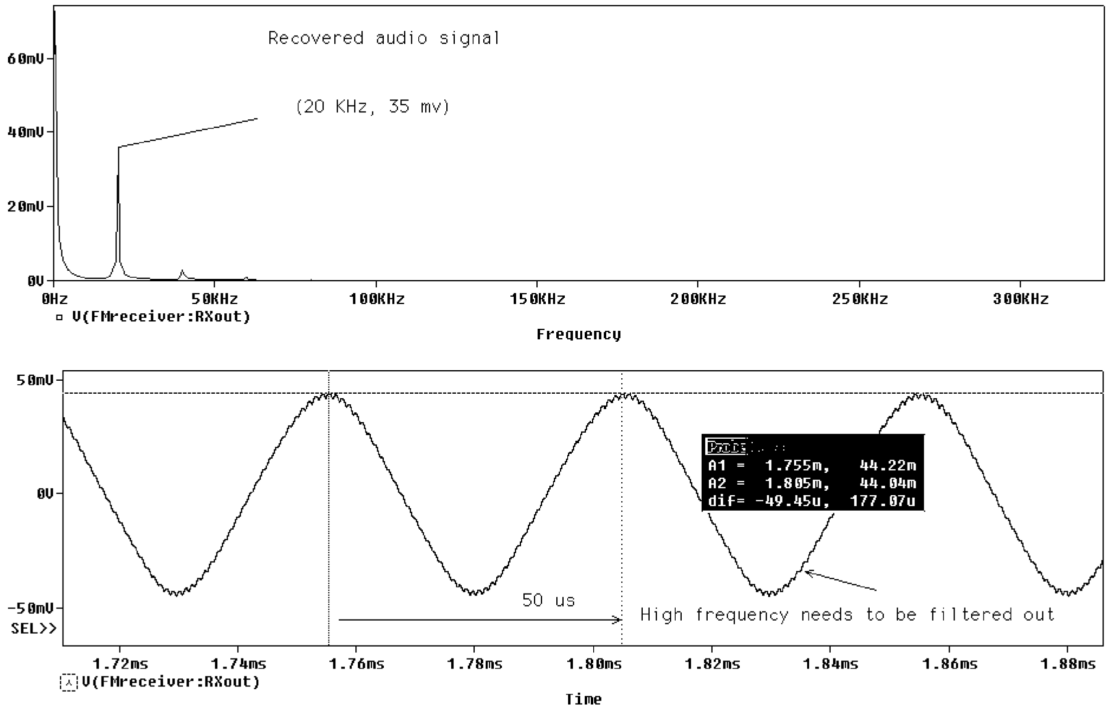


FIGURE 6.21: Audio waveforms

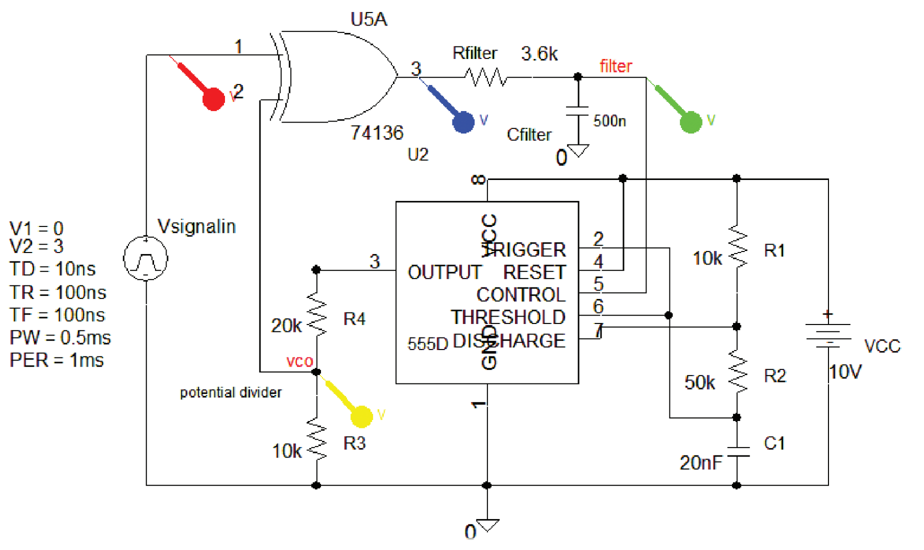


FIGURE 6.22: XOR-based PLL

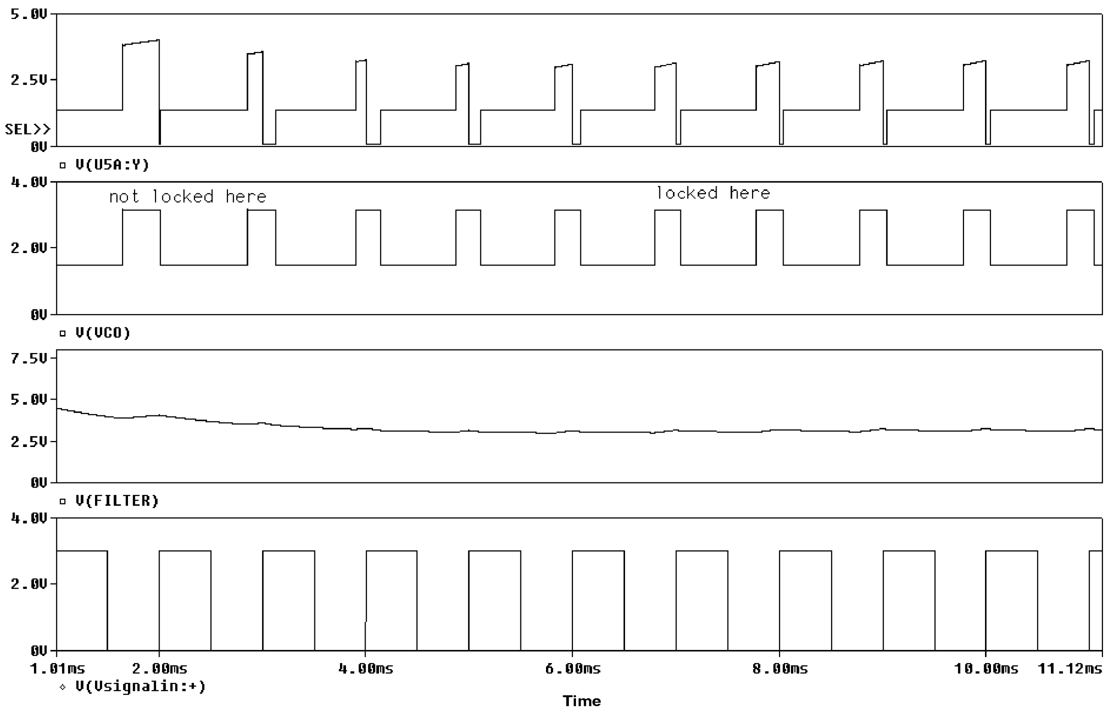


FIGURE 6.23: PLL waveforms

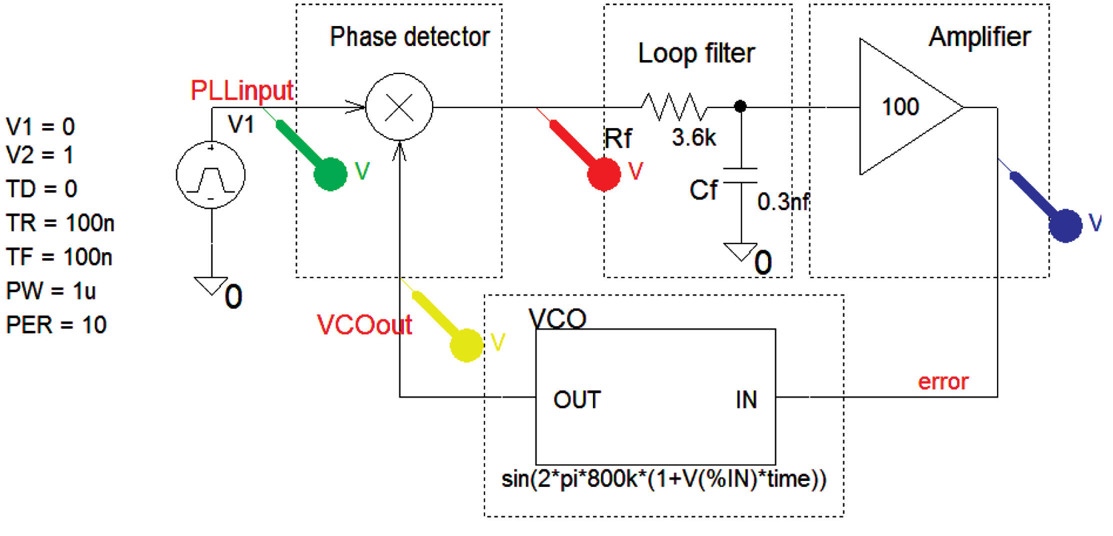


FIGURE 6.24: PLL step response

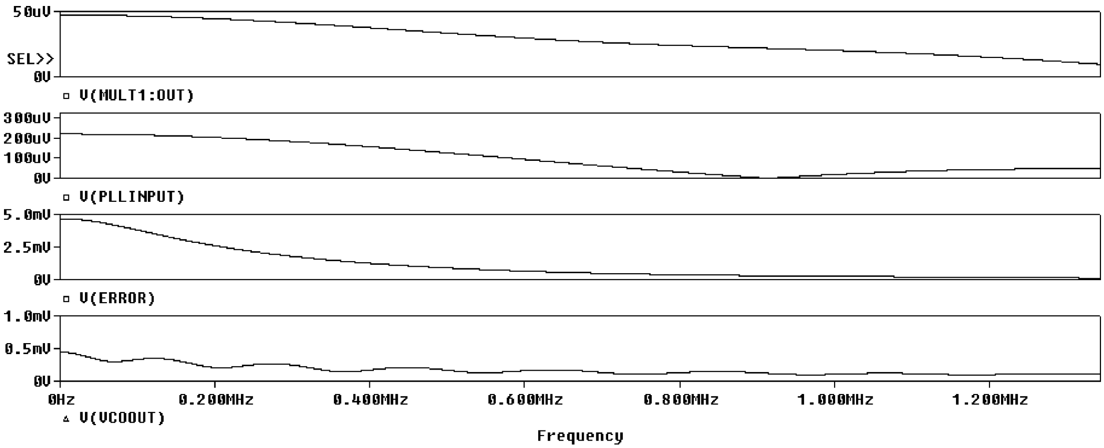


FIGURE 6.25: PLL spectrum

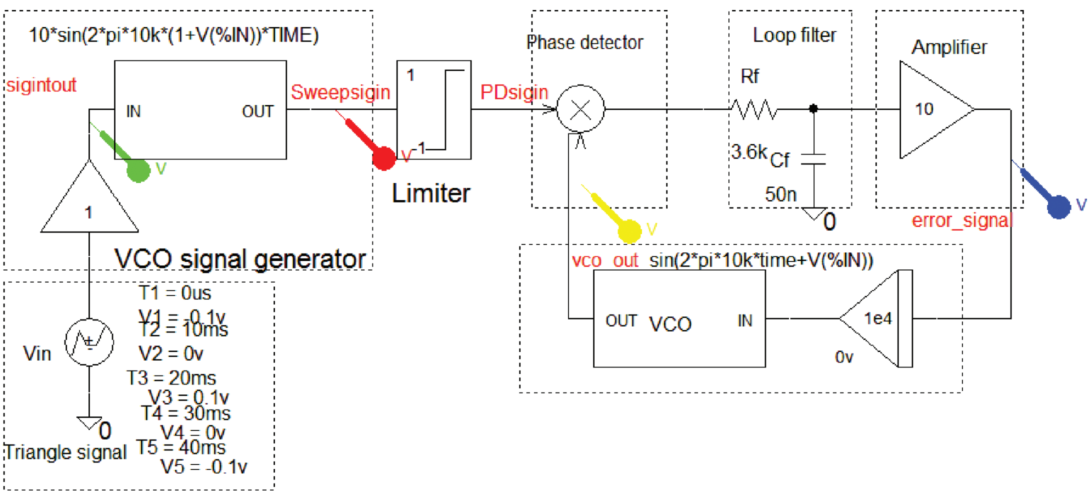


FIGURE 6.26: An alternative method of applying an FM input signal

(3) An alternative to the **EVALUE** part for sweeping the input signal is shown in Fig. 6.26. The free-running frequency of the VCO is set to the input frequency. Investigate the digital PLL shown in Fig. 6.27.

(4) Investigate the digital . . .

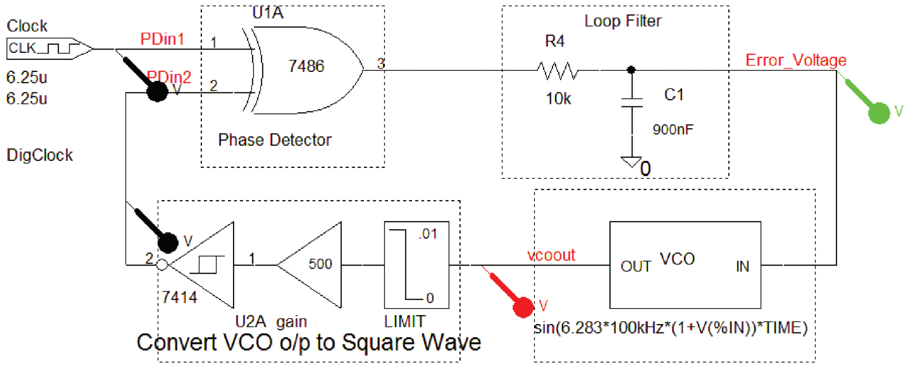


FIGURE 6.27: Digital PLL

CHAPTER 7

Noise

7.1 SOURCES OF NOISE

Natural and man-made noise is present in all electronic systems and produces data errors in digital receivers and annoying hiss in analog sound systems. It is possible, however, to minimize these errors by ensuring that the system has a good signal-to-noise ratio and also by applying clever coding algorithms to the data stream before transmission. Examples of **man-made** noise sources are: Car ignition systems (spark plugs, electrical motors), internal noise in electrical, and electronic circuits (due to random motion of electrons in conductors). Electronic noise from integrated circuit devices and components is categorized as thermal noise, shot noise, and frequency-dependent noise. Thermal noise (also called Johnson noise) is temperature dependent caused by random interaction between free electrons and vibrating ions in the crystal lattice.

Besides man-made noise, there are **natural noise sources** such as lightning storms, cosmic, and solar radiation. A technique for reducing receiver noise is to band-limit the signals, since, according to (7.1), noise power is proportional to bandwidth. However, the bandwidth size must be consistent with good signal quality. White noise (called this since it has a spectrum from 0 Hz to 10^{13} which is similar to light) has a constant power spectral density measured per Hertz of bandwidth (kT W Hz^{-1}). Resistors are a major source of noise in electronic systems, and Fig. 7.1 shows an *RMS* noise voltage v_n, in series with a resistance R connected to a noiseless resistance, R_L.

The **RMS** noise voltage generated in a resistance R, is

$$v_n = \sqrt{4kTBR} \tag{7.1}$$

where k is Boltzmann's constant (Ludwig Boltzmann 1844–1906) equal to 1.38×10^{-23} j K^{-1}. T is the resistor temperature in degree Kelvin (K) = (273° + room temp °C), and B is the bandwidth in Hz. Maximum power is transferred for $R = R_L$ so the voltage across R_L is half the input voltage and thermal noise power is

$$P_n = \frac{[v_n/2]^2}{R} = \frac{(\sqrt{4kTBR})^2}{4R} = kTB. \tag{7.2}$$

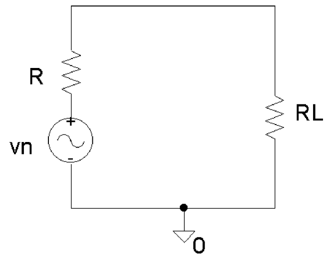


FIGURE 7.1: Measuring noise

Noise is a random signal and $\boldsymbol{N}_0$ is the amount of noise in each Hz and has units of W Hz^{-1} in the range 10^{-7} to 10^{-21} W Hz^{-1} (**White noise**). Thus, if $\boldsymbol{N} = \boldsymbol{kTB}$, then $N_0 = N/B$, or $\boldsymbol{N} = \boldsymbol{N}_0\boldsymbol{B}$.

7.2 NOISE FACTOR AND NOISE FIGURE

All systems introduce noise and the amount added is quantified by the noise factor. However, first we must define the signal power to noise power (S/N ratio or SNR) as

$$SNR = 10\log_{10}\frac{S}{N}\ \text{dB}. \tag{7.3}$$

The noise factor quantifies the effect of system noise on the overall noise performance. The noise ratio is the ratio of the input S/N ratio to the output S/N ratio, i.e.,

$$F = NR = \frac{S}{N}\text{in}/\frac{S}{N}\text{out}. \tag{7.4}$$

All electronic systems add to the overall noise, so the output SNR is always less than the input SNR so that F is greater than one. This is the factor by which a system degrades the SNR presented at the system input. The noise ratio in dB is the noise figure (NF):

$$NF = 10\log_{10} NR = 10\log_{10}(F)\ \text{dB}. \tag{7.5}$$

7.3 DEFRIIS' FORMULA

The DeFriis formula calculates the noise ratio for a multistage system, and shows how the first stage is the predominant factor on the overall noise factor. The **first stage** must have a **lower noise ratio**, and a **higher gain than other stages**, since its influence on the overall noise figure is much greater.

$$NR = N_{R1} + \frac{N_{R2} - 1}{A_1} + \frac{N_{R3} - 1}{A_1 A_2} + \cdots. \tag{7.6}$$

All power gains and noise ratios are in ratios, and not in dB.

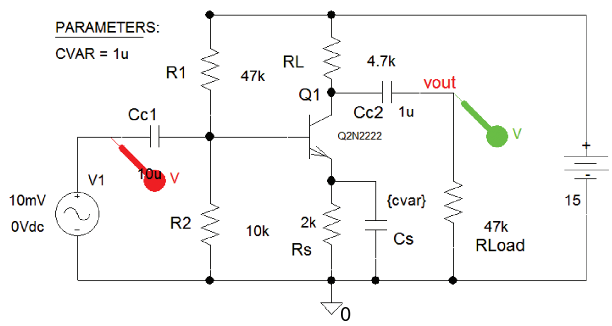


FIGURE 7.2: Common emitter amplifier

7.3.1 Common Emitter Amplifier

Fig. 7.2 is a common emitter amplifier to investigate device noise.

Select the output wire segment beside Cc_2, and add the **Net alias** called **vout**. From the **Edit simulation settings/Analysis type/ AC Sweep** and set **Logarithmic, Start Frequency** = 1, **End Frequency** = 100k, **Points/Decade** = 1001. Tick **Noise Enabled**, and **Output Voltage v(vout)**, **I/V Source** = Vin, and **Interval** = 1001.

The input voltage source **Vin** is entered in the **I/V Source** box to enable PSpice to associate the equivalent input noise with this source.

7.4 THE OUTPUT FILE

The output file shown in Fig. 7.3 is obtained by selecting **PSpice/View Output** from the main toolbar menu. The output file will have a slightly different appearance from the output file when accessed from Probe.

This file contains noise voltage details at each **Interval** entered in the **AC Sweep and Noise Analysis** submenu. The **Points/Decade** interval = 1001, produces details every decade,

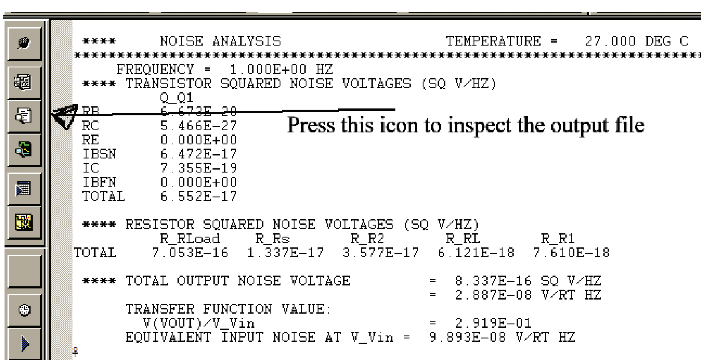


FIGURE 7.3: The output file

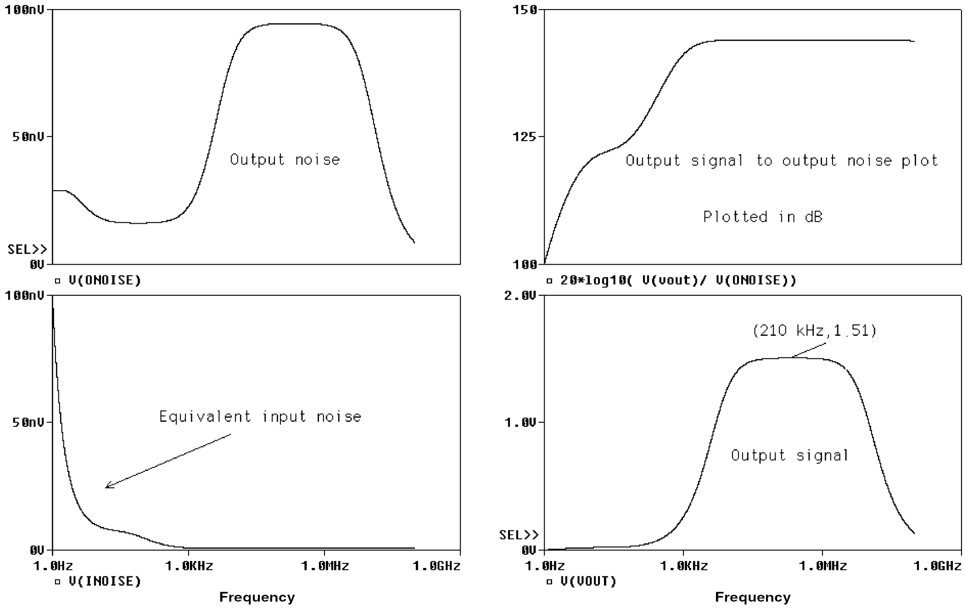


FIGURE 7.4: Output and equivalent input noise

i.e., 1e0, 1e2, 1e3, 1e4, 1e5. The emitter resistor is decoupled by a capacitor **cvar** that is made variable by replacing the default 1 nF with {**cvar**}. Place a **PARAM** part and enter **NAME1** = **cvar** and **VALUE1** = 1 µ. The **PARAM** part and the parametric sweep will show how the decoupling capacitor value affects the lower cut-off frequency. However, for the moment ignore this part. Fig. 7.4 shows how the noise voltage varies with frequency.

The signal-to-noise is plotted by typing **20*log10(v(vout)/v(onoise))** in the **Trace Expression** box.

7.5 PROBE EXPRESSION COMMANDS

PSpice has a very comprehensive list of Probe operators that may be used when in the Probe output screen. From the **Trace** menu, select an operation from the list on the right-hand side.

Logical and relational operators are used within the **IF( )** function. Digital parts are modeled using logical operators in Boolean expressions.

7.6 THE "(IF, THEN, ELSE)" COMMAND

The **IF(argument, then, else)** statement is used within ABM parts such as **ABM1**, **GVALUE**, **EVALUE,** etc, where the statement is entered in the expression box. The arguments are voltages, currents through voltage sources, arithmetic symbols, logical symbols, and relational symbols. For example, **IF(V(signal1)*V(signal2)< =5,2,1)**. Here the two-wire segment names

ABS(x) = |x|	the absolute value
AVGX(,) =	average of x over the range
SQRT(x)	$= x^{1/2}$
EXP(x)	$= e^{x}$
LOG(x) = ln(x)	the natural log to base *e*
LOG10(x) = log(x)	the log base 10
PWR(x,y)	$= x^y$ in transient analysis only
PWRS(x,y) = $+x^y$ (if $x > 0$) = $-\|x\|y$ (if $x < 0$)	
SIN(x) = sin(x)	with x is in radians
ASIN(x) = $\sin^{-1}$ (x)	where the result is in radians
SINH(x) = sinh(x)	where x is in radians
COS(x) = cos(x)	where x is in radians
ACOS(x) = $\cos^{-1}$ (x)	where the result is in radians
COSH(x) = cosh(x)	where x is in radians
TAN(x) = tan(x)	where x is in radians
ATAN(x) = ARCTAN(x) or $\tan^{-1}$ (x)	where the result is in radians
ATAN2(y,x) = $\tan^{-1}$ (y/x)	result in radians
TANH(x) = tanh(x)	where x is in radians
M(x) = magnitude of x	the same as ABS(x)
P(x) = phase of x* in degrees;	returns 0.0 for real numbers
R(x) r(x) = real part of x*	**applicable to AC analysis only**
IMG(x) = imaginary part of x	**applicable to AC analysis only**
DT(x) time derivative of x,	**applicable to transient analysis**

"signal1" and "signal2" are multiplied together. If the result is less than, or equal to 5 V, then the output value is 2, otherwise it is 1. Another example: **IF(I(v1) == 50m&v (input)>12,{1/R1},{R2})**. The two parts are separated by a Boolean **AND** function. If the current through the voltage source, v1, is 50 mA **AND** the voltage on the wire seg-

arithmetic + addition, or string	
Concatenation	
–	Subtraction
*	Multiplication
/	Division
**	Exponentiation
logical * ~	unary NOT
|	Boolean OR
^	Boolean XOR
&	Boolean AND
relational* ==	equality test
! =	nonequality, test
>	greater than, test
>=	greater than or equal to, test
<	less than test
<=	less than or equal to test
IF(argument, then, else)	

ment name "input" is greater than 12 V, then the value is 1/R1, otherwise it is R2, where R1 and R2 are defined in a **PARAM** part. The **IF** statement format in PSpice is: **IF(T,X,Y)**, which reads: IF **T** THEN **X** ELSE **Y**. **IF** statements are nested by adding additional **IF** statements in the **ELSE** part of the argument. For example, **IF(T,X,IF(A,B,C))**, which reads: IF T THEN A ELSE (IF B THEN R ELSE C). You may continue to nest IF statements inside IF statements. Fig. 7.5 shows a schematic to test the "**(if, then, else)**" command entered in the two-input **ABM2/1** part. The **IF** statement performs the logical **OR** function between two signals and sets the output to 1 if true but if not true, it sets the output to 3. The **OR** symbol is obtained by selecting **shift**+ \ **key** (located to the left of the Z key on some keyboards). Press the **V** icon to display the voltages on the schematic after simulation.

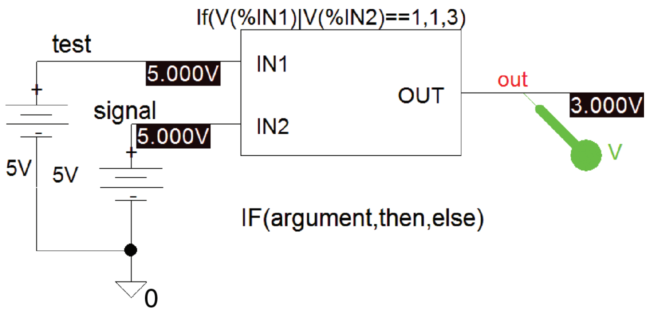


FIGURE 7.5: ABM2/1

7.7 IMPORTING NOISE

We may import noise using an ASCII file created in Matlab. Open up Matlab and enter, at the command line, the following code. Alternatively, create an *m* file (**wnoisegen.m** in the miscellaneous directory), and enter the details.

```
fs = 31250;                          % change to a higher value for finer detail noise
ms = 1000;                           % duration in milliseconds
a = rand(ceil(fs*ms/1000), 1)
   * 2.0 – 1.0;                      % a = rand(10000, 1); % generates a random noise
% a = a-mean(a);                     % removes the DC from the signal
t = (1:1:length(a))/fs;              % creates a time vector to match the random vector
t = t';                              % Produces a column vector
wnoise = [t a];                      % Creates time–amplitude pair and store in info
save C:\signalsources\
   wnoise.txt wnoise info -ASCII;    % saves info as an ASCII file plot(t, a)
```

The ASCII file **noise_info.txt** is available from the MorganClaypool site if you have no access to Matlab, or are not familiar with it.

7.7.1 Adding Noise to the Input Signal

The schematic in Fig. 7.6 has the output wire segment labeled **out**.

The **VPWL_F_RE_FOREVER** generator reads in the ASCII noise file **noise_info.txt** located in the **C:\Pspice\Circuits\signalsources\noise\noise_info.dat**. The noise amplitude is set by a **GAIN** part, but we can also use the VSF attribute to control the noise amplitude. Set the **Analysis** tab to **Analysis type: Time Domain (Transient)**, **Run to time** = 15 ms. Press **F11** to produce to display signals in Fig. 7.7. Noisy data and noise signals are plotted into two displays using key sequences: **alt PP, ctrl X, and ctrl V**.

7.8 EXERCISES

(1) Use a **Param** part to couple the receiver front-end input tuning capacitor and the local oscillator capacitor.

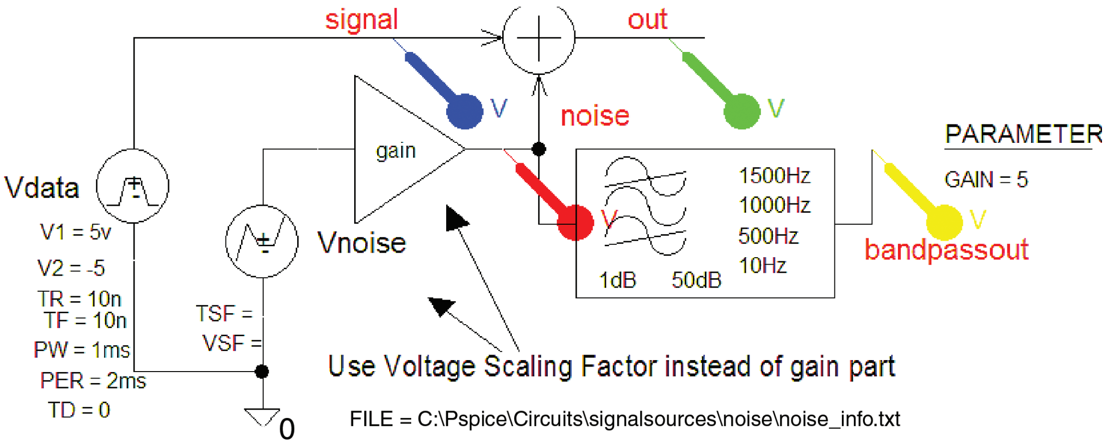


FIGURE 7.6: Importing noise into PSpice

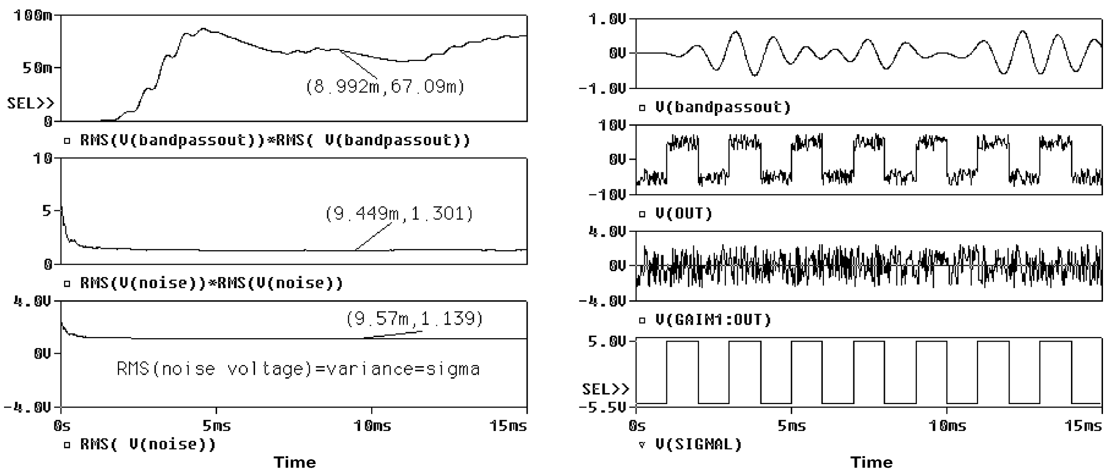


FIGURE 7.7: The data signal plus noise

(2) Download from the Net, PSpice models for the power transistors TIP 31 and TIP32. It is necessary to create symbols for each transistor. When the symbols are created and stored in a library, investigate the power amplifier used previously, and apply feedback from the output back to the input.

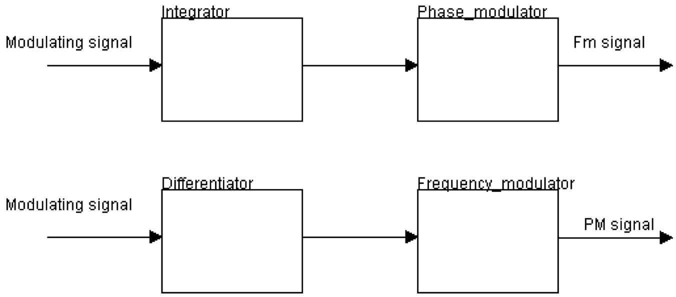


FIGURE 7.8: Production of PM from FM

(3) Create an actual circuit for linearly summing the two audio signals in the stereo generation schematic.

(4) Investigate the production of phase modulation using the block diagram shown in Fig. 7.8 as a guide. Use the **DIFF** and **VSFFM** parts.
How would you produce wideband FM signal from narrowband FM (hint: see stereo FM production)?

(5) Compare the selectivity of the two tuned RF amplifiers shown in Fig. 7.9.

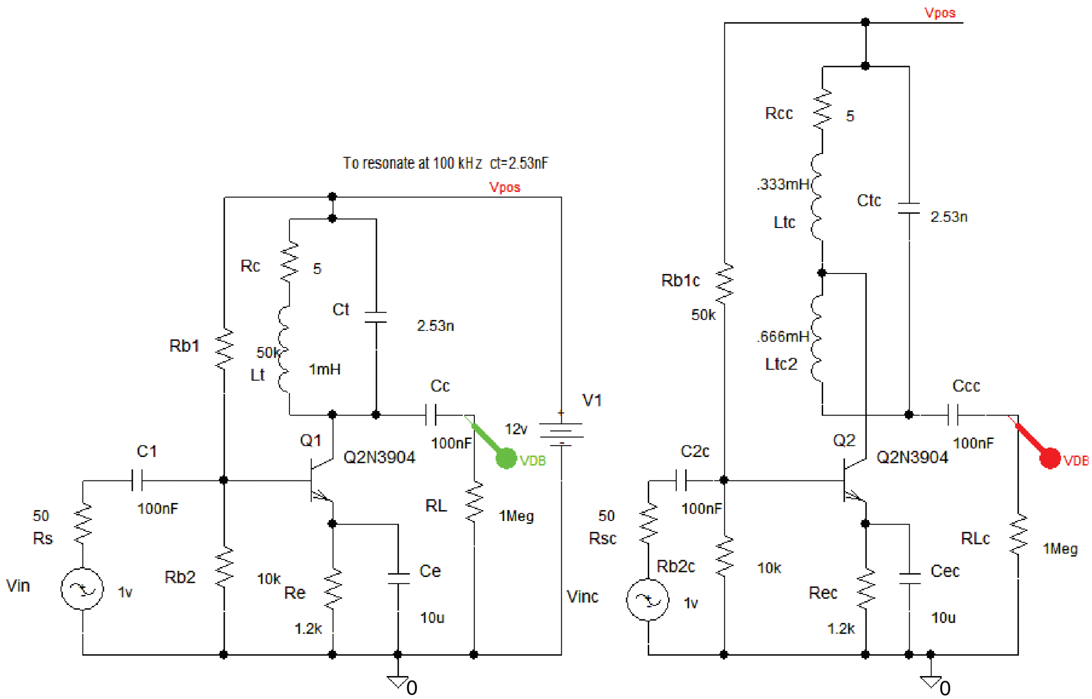


FIGURE 7.9: Tapped resonant circuit for better matching and selectivity

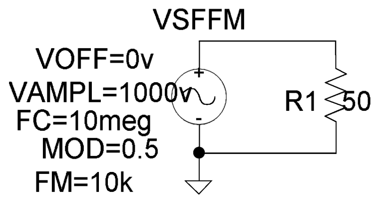


FIGURE 7.10: FM signal generation

(6) Fig. 7.10 shows an *FM* signal is $v(t) = 1000\cos(2\pi 10^7 t + 0.5\cos 2\pi 10^4 t)$ connected across a 50-Ω resistance. Determine

- the modulation index and peak frequency deviation Δf_c,
- the modulator sensitivity for an input voltage of 0.5 V to produce Δf_c,
- the FM spectral components,
- the total power and the sideband power (information power), and
- the 99% signal bandwidth.

The total power is plotted in **Probe** by entering **RMS(V2(R1))*RMS(V2(R1))/(50)** in the **TRACE EXPRESSION** box. The information power is calculated by subtracting the carrier power from the total power as **RMS(V2(R1))*RMS(V2(R1))/(50)-940*940/100**. However, if the carrier (the $J(o)$ component) is zero (for example, when the modulation index = 2.4), we would need to add the power components from each of the Bessel components. Set the transient parameters: **Maximum step size** 1 μs to and **Run to time** to 2 ms [ref: 5 Appendix A]. This simulation produces a narrow band FM signal with only two side frequencies (see Table 3 in Appendix A).

(7) Investigate the digital PLL in Fig. 7.11.

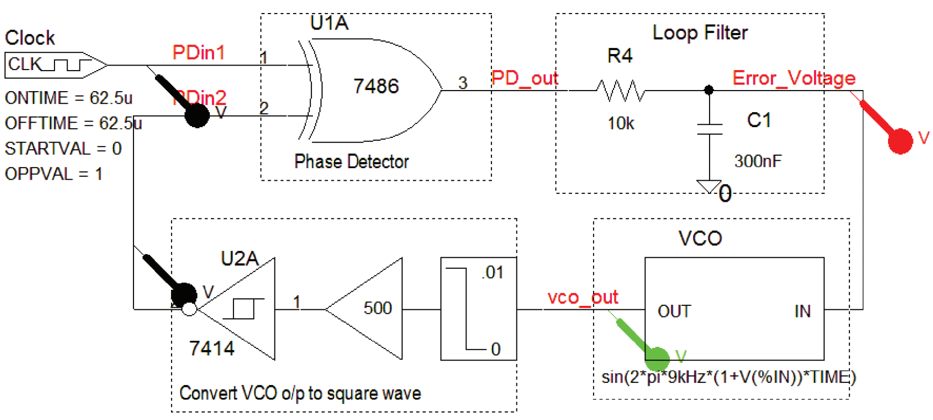


FIGURE 7.11: Digital PLL

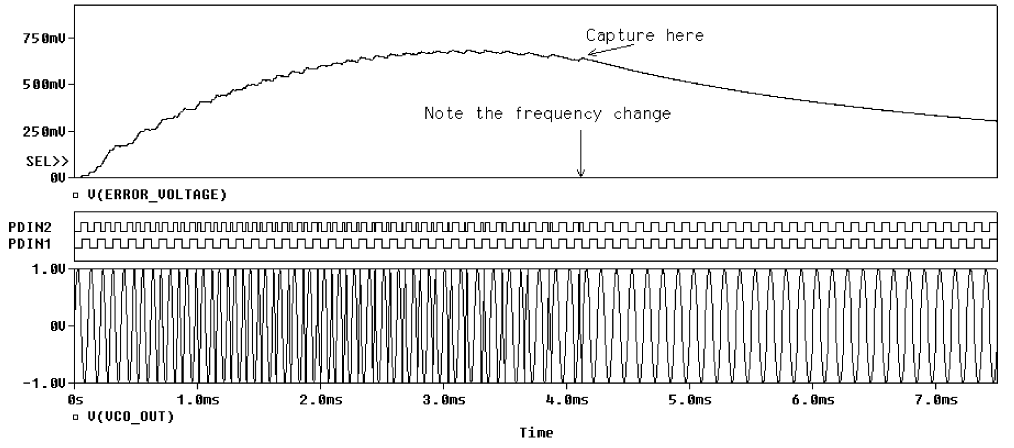


FIGURE 7.12: PLL signals

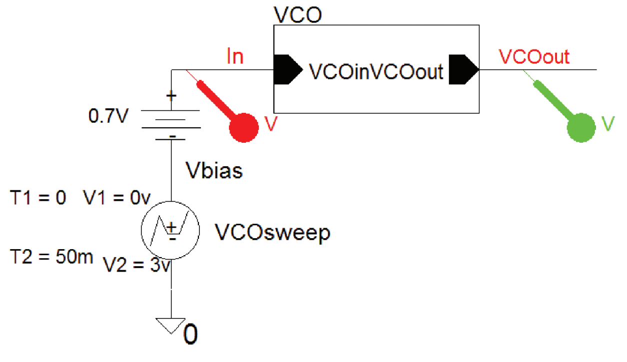


FIGURE 7.13: VCO with ramp input signal

The output signal is displayed in Fig. 7.12.

(8) Investigate the VCO in Fig. 7.13 using the PLL VCO from exercise (10).

(9) The ideal operational amplifier used in the VCO shown in Fig. 7.14 is used here to comply with the evaluation criteria. Investigate VCO performance and see if you can determine the voltage sensitivity k_o.

(10) Investigate the ABM VCO in Fig. 7.16.
The waveforms are shown in Fig. 7.17. Connect each generator in turn. The modsweepin is a ramp signal that starts at 50 µs, ramps for 100 µs, and then stays at a constant value. Modsine can be used for FM production.

(11) Investigate the PLL frequency synthesizer in Fig. 7.18.
The frequency divider block content is shown in Fig. 7.19.
The output results are shown in Fig. 7.20.

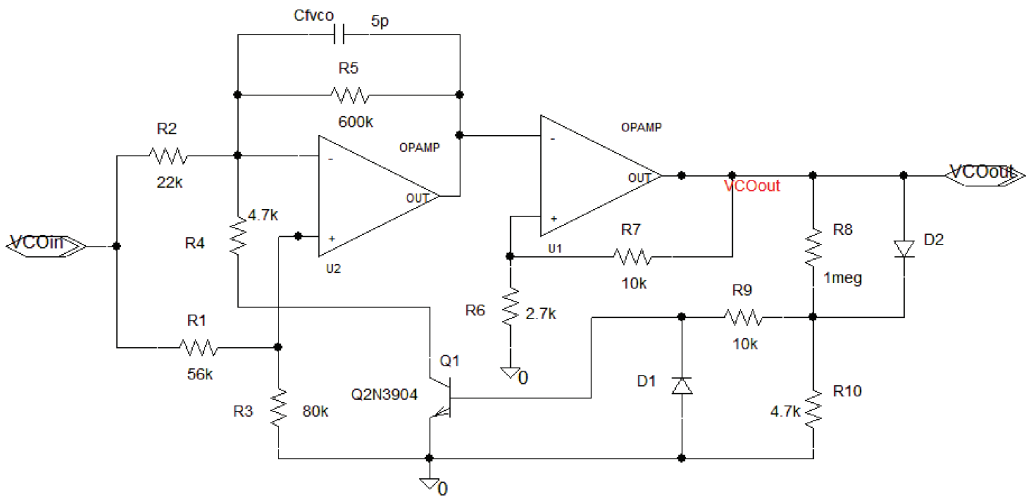


FIGURE 7.14: Voltage-controlled oscillator

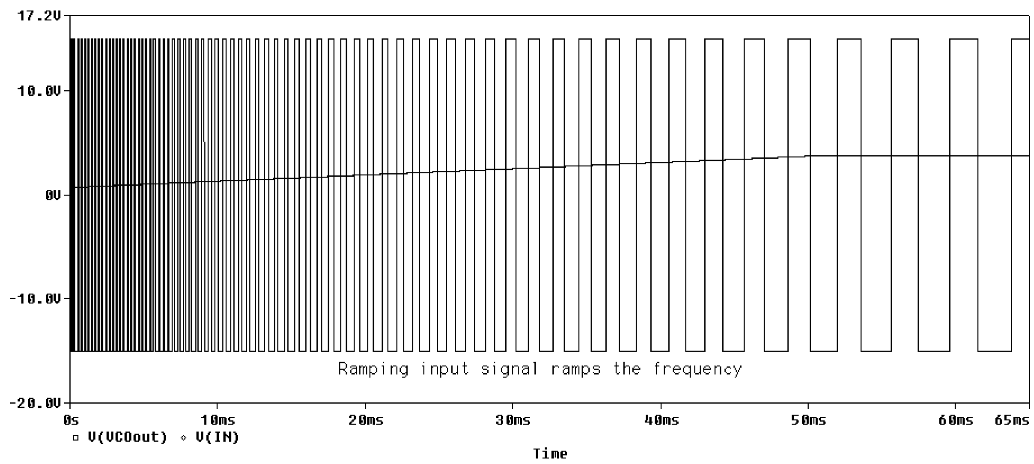


FIGURE 7.15: Ramp voltage changing the VCO frequency

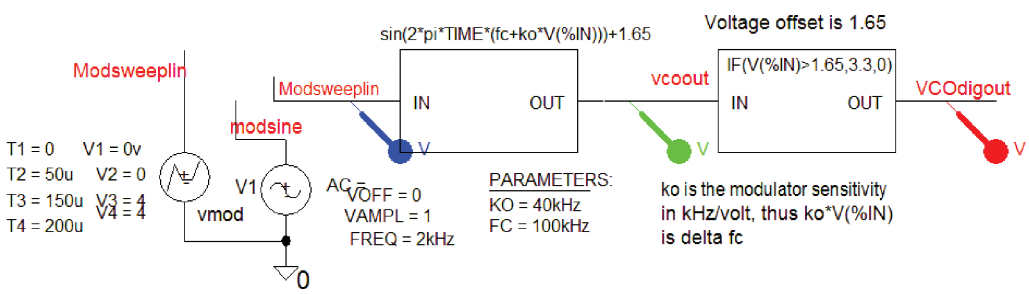


FIGURE 7.16: VCO

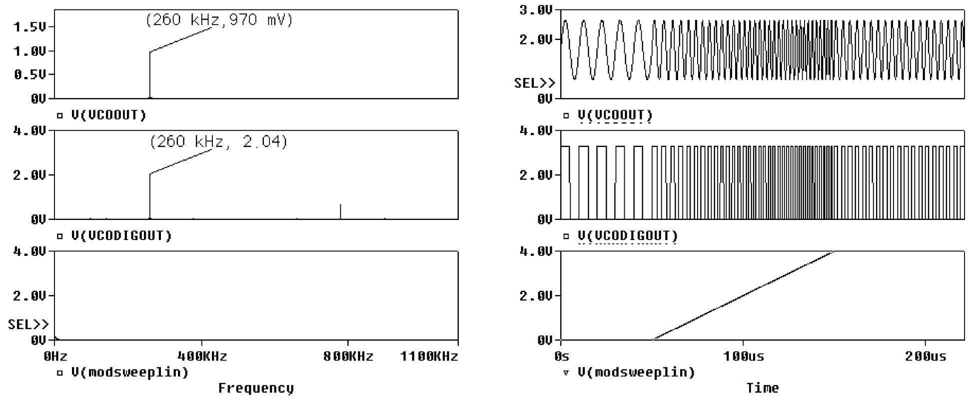


FIGURE 7.17: VCO waveforms

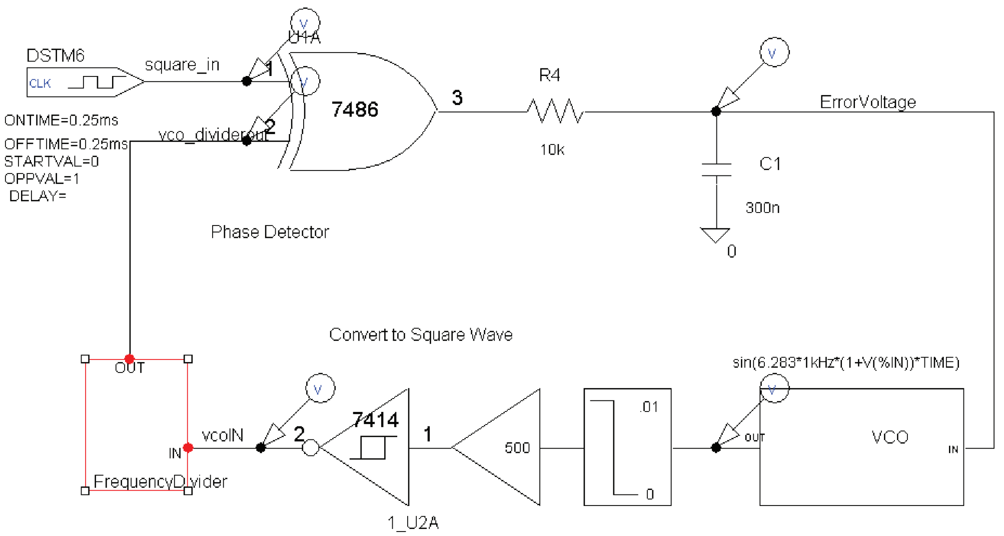


FIGURE 7.18: Frequency synthesizer

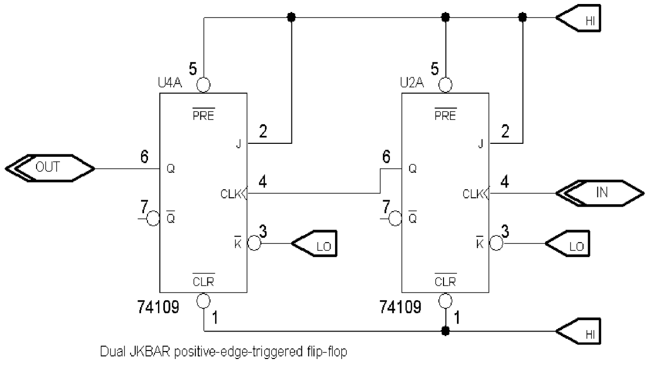


FIGURE 7.19: Frequency division

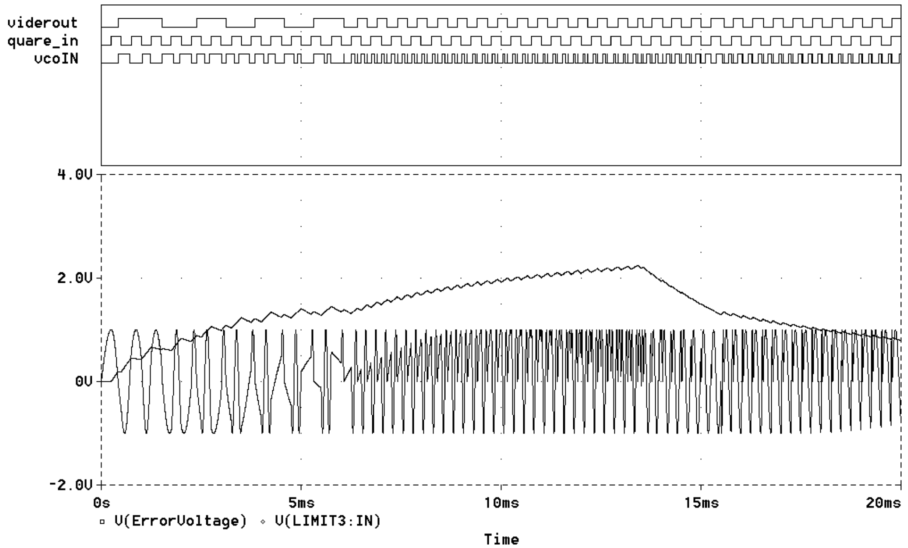


FIGURE 7.20: Output waveforms

APPENDIX A

TABLE 7.1: Laplace and z-transform Table

FUNCTION	F(T)	LAPLACE TRANSFORM	F(N)	z-TRANSFORM ($t = nT = n$)
Unit step	$u(t)$	$1/s$	$u(n)$	$z/z-1$
Unit impulse	$\delta(t)$	1	$\delta(n)$	1
Unit ramp	T	$1/s^2$	N	$nz/(z-1)^2$
Polynomial	t^n	$n!/s^{n+1}$	t^n	$T^2 z(z+1)/(z-1)^2$ for $n=2$
Decaying exponential	e^{-at}	$1/(s+a)$	$e^{-an}u(n)$	$z/z-e^{-an}$
Growing exponential	$1/a(1-e^{-at})$	$1/(s+a)(s)$	$1/a(1-e^{-an})$	$z(1-e^{-an})/a(z-1)(z-e^{-an})$
Sine	$\sin(\omega t)$	$\omega/(s^2+\omega^2)$	$\sin(n\theta)u(n)$	$(z\sin n\theta)/(z^2-2z\sin n\theta+1)$
Cosine	$\cos(\omega t)$	$s/(s^2+\omega^2)$	$\cos(n\theta)u(n)$	$\dfrac{z(z-\cos n\theta)}{z^2-2z\cos n\theta+1}$
Damped sine	$e^{-at}\sin(\omega t)$	$\omega/((s+a)^2+\omega^2)$	$e^{-an}\sin(n\theta)$	$\dfrac{ze^{-an}\sin(n\theta)}{z^2-2ze^{-an}\cos n\theta+e^{-2an}}$
Damped cosine	$e^{-at}\cos(\omega t)$	$(s+a)/((s+a)^2+\omega^2)$	$e^{-an}\cos(n\theta)$	$\dfrac{z^2-ze^{-an}\cos(n\theta)}{z^2-2ze^{-an}\cos n\theta+e^{-2an}}$
Delay	$f(t-k)$	e^{-sk}	$f(n-k)$	z^{-k}

TABLE 7.2: Fourier Transform Table

FUNCTION	F(T)	FOURIER TRANSFORM
Constant	1	$\delta(f)$
Impulse	$\delta(t - t_0)$	$e^{(-j2\pi f_0)}$
Squarewave pulse	T	$T\dfrac{\sin(\pi f T)}{\pi f T}$
Cosine	$\cos(\omega_0 t + \theta)$	$0.5e^{j\theta}\delta(f - f_0) + 0.5e^{-j\theta}\delta(f + f_0)$
Triangular signal	T	$T\left[\dfrac{\sin(\pi f T)}{\pi f T}\right]^2$
Delay Td	$x(t - T_D)$	$X(f)e^{(-j2\pi f T_D)}$
Frequency translation	$x(t)e^{(j2\pi f_0 t)}$	$V(f - f_0)$

TABLE 7.3: Bessel Functions

BETA	J(0)	J(1)	J(2)	J(3)	J(4)	J(5)	J(6)	J(7)	J(8)	J(9)
0	1	0	0	0	0	0	0	0	0	0
0.25	0.9844	0.124	0	0	0	0	0	0	0	0
0.5	0.9385	0.2423	0.0306	0	0	0	0	0	0	0
0.75	0.8642	0.3492	0.0671	0	0	0	0	0	0	0
1	0.7652	0.4401	0.1149	0.0196	0	0	0	0	0	0
1.25	0.6459	0.5106	0.1711	0.0369	0	0	0	0	0	0
1.5	0.5118	0.5579	0.2321	0.061	0.0118	0	0	0	0	0
1.75	0.369	0.5802	0.294	0.0919	0.0209	0	0	0	0	0
2	0.2239	0.5767	0.3528	0.1289	0.034	0	0	0	0	0
2.25	0.0827	0.5484	0.4047	0.1711	0.0515	0.0121	0	0	0	0
2.4	0.0025	0.5202	0.431	0.1981	0.0643	0.0162	0	0	0	0
2.5	−0.0484	0.4971	0.4461	0.2166	0.0738	0.0195	0	0	0	0
2.75	−0.1641	0.426	0.4739	0.2634	0.1007	0.0297	0	0	0	0
3	−0.2601	0.3391	0.4861	0.3091	0.132	0.043	0.0114	0	0	0
3.5	−0.3801	0.1374	0.4586	0.3868	0.2044	0.0804	0.0254	0	0	0
4	−0.3971	−0.066	0.3641	0.4302	0.2811	0.1321	0.0491	0.0152	0	0
4.5	−0.3205	−0.2311	0.2178	0.4247	0.3484	0.1947	0.0843	0.03	0.0091	0
4.75	−0.2551	−0.2892	0.1334	0.4015	0.3738	0.228	0.1063	0.0405	0.0131	0
5	−0.1776	−0.3276	0.0466	0.3648	0.3912	0.2611	0.131	0.0534	0.0184	0
5.5	−0.0068	−0.3414	−0.1173	0.2561	0.3967	0.3209	0.1868	0.0866	0.0337	0.0113

TABLE 7.4: Useful Trigonometrical Identities

FUNCTION	EXPANSION
sin A sin B	0.5 cos(A − B) − 0.5 cos(A + B)
cos A cos B	0.5 cos(A − B) + 0.5 cos(A + B)
sin A cos B	0.5 sin(A − B) + 0.5 sin(A + B)
$e^{\pm j\theta}$	$\cos\theta \pm j sin\theta$
$\cos\theta$	$(e^{j\theta} + e^{-j\theta})/2$
$\sin\theta$	$(e^{j\theta} - e^{-j\theta})/2j$
$1 = \cos^2\theta + \sin^2\theta$	
$\cos 2\theta = \cos^2\theta - \sin^2\theta$	

REFERENCES

[1] Tobin Paul PSpice for Circuit Theory and Electronic Devices Morgan Claypool publishers Jan 2007.

[2] Tobin Paul PSpice for Digital signal processing Morgan Claypool publishers (March 2007)

[3] Tobin Paul PSpice for Digital Communications Morgan Claypool publishers March 2007

[4] Bateman Andrew Design for the Real World with CDROM (Paperback) Addison-Wesley

[5] Young Paul. H. Electronic Communications Techniques USA: Maxwell-MacMillan 2003

[6] Boylesland Robert and Nashelsky Louis, Electronic Devices and Circuit Theory, Prentice Hall International, 1982

[7] Tobin Paul (March 2007) PSpice for Filters and Transmission Lines. Morgan Claypool publishers.

Index

abm1 part, 5
abm2/1 part, 120
abs(x), 119
acos(x), 119
ad633, 86
additive mixer, 51
adjacent signals, 58
agc, 23, 100
amplifier noise, 117
amplitude modulation, 3
amplitude modulator, 10
arithmetic, 120
ascii file, 121
asin(x), 119
atan(x), 119
atan2(y,x), 119
automatic gain control, 67
avgx, 119

bandwidth, 26, 115
Barkhausen criterion criteria, 33
beta network, 35
Boltzmann constant, 115
Boolean and, 120
Boolean or, 120
boolean xor, 120
bubble part, 44, 121

centre-tapped transformer, 26
class C amplifier, 89
coherent detection, 26
Colpitts oscillator, 33
common emitter amplifier, 117
compensated PLL, 103
const part, 8
convergence problems, 11
cos(x), 119
cosh(x), 119
crossover distortion, 67

damping factor, 104
DeFriis', 116
diagonal clipping distortion, 21
diff part, 87
division, 120
double-sideband suppressed carrier, 10
DSBFC, 7
DSBSC, 11
dt(x) time derivative of x, 119
dynamic impedance, 61, 63

Edit Properties, 5
emult part, 73
envelope, 3
exp, 119
exponentiation, 120

FFT, 4, 24
fifty percent duty cycle, 110
floating error, 36
four-quadrant multiplier, 10, 28
frequency deviation, 82

frequency division multiplexing, 2
frequency doubling, 88
frequency-doubler, 12

gain margin, 44
gain part, 121
global parameters, 98
greater than, 120
greater than or equal to, 120

harmonic distortion, 44
Hartley oscillator, 38
heterodyne, 56

i/v source, 117
iac source, 41, 97
if, then, else, 120
image frequency, 53
img(x) = imaginary part of x, 119
intermediate frequency, 52, 57
internal noise, 115
issfm generator, 77
ITU, 95

jfet ac equivalent circuit, 64

k_linear part, 26

less than, 120
lock and capture range, 105
lock range, 104
log, 119
log command, 69
log commands, 24
log10, 119
logical, 120
loop gain, 36

m(x), 119
magnifying tool, 36
man-made noise, 115
maximum power transfer, 115
Maximum step size, 26
modified Carson's rule, 78
modulation index, 95
modulator sensitivity, 75
mult part, 8
multiplication, 120
multiplicative mixer, 51
multiplier, 10

natural noise sources, 115
π network, 91
noise factor, 116
noise power, 115
non-equality test, 120
non-linear mixer, 52
npn, 65
Nyquist plot, 44

open plot, 24
oscillator, 33

p(x), 119
Param part, 16
param part, 99
parametric, 98
peak frequency deviation, 75
phase detector, 105
phase difference, 13
phase margin, 44
phase-locked loop, 101
phase-shift method, 30
phase-shift oscillator, 43
pilot carrier, 83

pilot tone, 86
pivot, 5
power transistors, 122
probe log, 24
probe operators, 118
push-pull, 67
pwr(x,y), 119
pwrs(x,y), 119

quadratic relationship, 51
quartz crystal, 41

r(x) = real part of x*, 119
rand, 121
relational, 120
Run to time, 26

skip initial transient, 37
scrambling technique, 69
section information, 99
selectivity, 58
side frequencies, 3
sin(x), 119
sinh(x), 119
spectral resolution, 4
speech files, 84
sqrt, 119
– subtraction, 120
SSBSC, 30
stability, 45
superhetrodyne action, 95
superhetrodyne receiver, 51, 57, 58
superhetrodyning, 53
sustained oscillations, 33

tan(x), 119
tanh(x), 119
thermal (Johnson noise), 115
time constant, 21
trace add, 63
trf receiver, 57
TSF, 8

varactor diode, 88
vco, 105
vpulse generator, 81
vpwl generator, 5
VPWL_F_RE_FOREVER part, 8, 121
VSF, 8
vsffm generator, 77
vsin generator, 3, 12

wav2ascii program, 86
wav2ascii.exe, 70
Wien network, 46

Author Biography

Paul Tobin graduated from Kevin Street College of Technology (now the Dublin Institute of Technology) with honours in electronic engineering and went to work for the Irish National Telecommunications company. Here, he was involved in redesigning the analogue junction network replacing cables with PCM systems over optical fibres. He gave a paper on the design of this new digital junction network to the Institute of Engineers of Ireland in 1982 and was awarded a Smith testimonial for one of the best papers that year. Having taught part-time courses in telecommunications systems in Kevin Street, he was invited to apply for a full-time lecture post. He accepted and started lecturing full time in 1983. Over the last twenty years he has given courses in telecommunications, digital signal processing and circuit theory.

He graduated with honours in 1998 having completed a taught MSc in various DSP topics and a project using the Wavelet Transform and neural networks to classify EEG (brain waves) associated with different mental tasks. He has been a ‘guest professor’ in the Institut Universitaire de Technologie (IUT), Bethune, France for the past four years giving courses in PSpice simulation topics. He wrote an unpublished book on PSpice but was persuaded by Joel Claypool (of Morgan and Claypool Publishers) at an engineering conference in Puerto Rico (July 2006), to break it into five PSpice books. One of the books introduces a novel way of teaching DSP using PSpice. There are over 500 worked examples in the five books covering a range of topics with sufficient theory and simulation results from basic circuit theory right up to advanced communication principles. Most of these worked example circuit have been thoroughly ‘student tested’ by Irish and International students and should mean little or no errors but alas. . . He is married to Marie and has four sons. His hobbies include playing modern jazz on double bass and piano but grew up playing G-banjo and guitar. His other hobby is flying and obtained a private pilots license (PPL) in the early 80’s.