The role of PSpice in the engineering teaching environment

Paul Tobin

Department of Electronic and Communications Engineering, Ireland
Dublin Institute of Technology, paul.tobin@dit.ie

Abstract - **We all have experienced engineering students who have difficulties with circuit fundamental principles and weak mathematical abilities. One solution is to introducing the teaching world of PSpice simulation at the start of a course rather than at mid point. The powerful, highly-visual simulation world of Orcad® PSpice® is a valuable tool for understanding circuit theory and electronic design, giving the student access to explore difficult aspects of a topic.**

Index Terms – PSpice simulation, mathematical rigor, education, transmission lines, Gaussian distribution, macros, 16-QAM.

INTRODUCTION

Electronic simulation software for examining systems behavior, can be categorized into two types: (i) Type one software packages are Matlab/Simulink®, Mustig®, and Scilab®, using functional blocks to examine a system, whereas (ii) Type two approaches system analysis from component level up to functional blocks. PSpice falls into the latter category and is a world standard that has proven itself in the area of circuit design and printed circuit production. However, students may also use it as a learning tool for understanding circuit principles with, or without, a rigorous mathematical treatment. Other software packages such as Tina®, Workbench®, perform similar roles but we will restrict ourselves to examples using PSpice only. Simulation software tools have been shown to be effective in communicating abstract ideas to students. Quoting from a fine paper [2], "The performance of students who used simulation, and those who used traditional laboratories, it was found that, the former group scored higher on a written exam".

In this present paper, I am not suggesting that simulation alone is the answer, but rather, can be used as a powerful tool in the learning process and quoting from another paper [4] "Software simulation does not provide an optimal learning experience by itself, but when combined with hands-on experience, theory and abstraction, however, simulation software can be a part of a learning environment that will likely produce excellent results". We will now examine a series of PSpice examples that can be used in the class and laboratory environments to reinforce circuit theory concepts. Example two looks at standing waves on transmission lines when they are incorrectly terminated. Example three introduces the MACRO, which is a very useful technique for

examining complex functions using equations rather than components. In this example, data, corrupted by noise, produces errors in a digital receiver. Example four looks at a digital signal processing (DSP) application. Here, the discrete Hilbert transform is implemented using a digital finite impulse response filter (FIR) for producing a constant ninety-degree phase shift in the phase/frequency response. Example five examines how component tolerances affect the frequency response of an active filter. The last example looks at a 16-quadrature amplitude modulation system (16-QAM). Here, construction uses hierarchical drawing techniques and the system tested using 'home-made test instruments'.

(1) CIRCUIT THEORY EXAMPLES

This first example looks at the concept of ac reactance normally introduced in a first year circuit theory class. The mathematics for capacitive reactance is quite simple, but here, PSpice helps students understand how capacitive reactance changes with frequency, and reinforces the idea that a capacitor is effectively an open circuit at DC, but a short circuit at very high frequencies [5]. Since current is proportional to the rate of change of charge with time, and $Q = CV$, so for a sinusoidal input signal, we can write

$$i_c = \frac{dQ}{dt} = C\frac{dV_c}{dt} = C\frac{d(V_m \sin \omega t)}{dt} = \omega C V_m \cos \omega t \quad (1)$$

Therefore, capacitive reactance is:

$$i_c = \frac{V_m}{X_C}\cos \omega t = I_m \cos \omega t \Rightarrow X_c = V_m / I_m = 1/ \omega C \ \Omega \ (2)$$

The schematic in Figure 1 shows a small resistance to represent the voltage source resistance, and a capacitance whose frequency response we wish to plot.
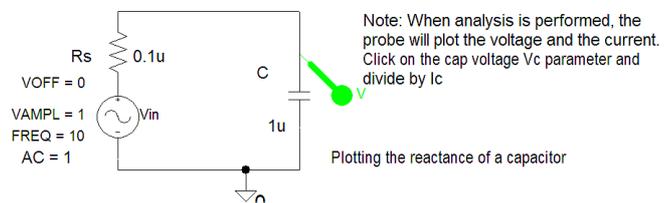

FIGURE 1
SCHEMATIC FOR PLOTTING CAPACITIVE REACTANCE

Figure 2 shows how capacitive reactance varies with frequency showing it has a very high value at very low frequencies, but decreases with an increase in frequency to a very low value.
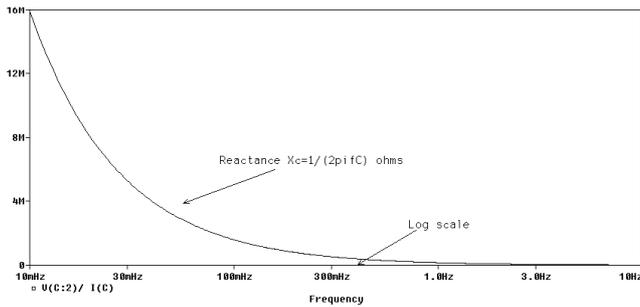
FIGURE 2
CAPACITIVE REACTANCE VERSUS FREQUENCY

Phasor relationships in CR, and LR circuits, together with complex algebra, seem to present major hurdles to some students. However, PSpice can show just how easy it is to observe the time delay between current and voltage in a capacitor and then translating this delay to an actual phase. To plot the phase relationships between the capacitor current and voltage, we carry out a transient analysis and measure the time difference as displayed in Figure 3 [5]. The phase difference between current and voltage signals is determined indirectly using cursors to measure the time difference, Δt. This measured time difference is then converting to phase in degrees, using the following expression:

$$\frac{T}{360} = \frac{\Delta t}{\theta} \Rightarrow \theta = \frac{360 x \Delta t}{T} = \frac{360 x 25 \text{ ms}}{100 \text{ ms}} = 90^0 \qquad (3)$$

T is the period of the signal equal to 100 ms for a 10 Hz input sinusoid, where there are 360° for a complete period.
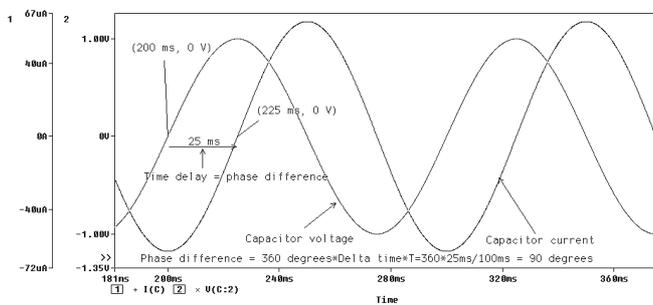

FIGURE 3
PHOROR RELATIONSHIPS

(2) TRANSMISSION LINES

Traditionally, the topic of transmission lines was introduced by deriving basic line equations using partial differential calculus- a luxury we can no longer afford. However, we may illustrate concepts, such as the voltage standing wave ratio (*VSWR*) defined as the ratio of the maximum voltage to the minimum voltage: $VSWR = V_{max}/V_{min}$ [6]. Figure 4 shows a 75 Ω (the characteristic impedance, Z_o) transmission line terminated with four values of load impedance, Z_L: (i) open-circuit, (ii) short-circuit, (iii) 37.5 Ω, and (iv) 75 Ω. This example demonstrates what happens when a line is terminated in a load other than the characteristic impedance. An expression for the voltage reflection coefficient, ρ, is:

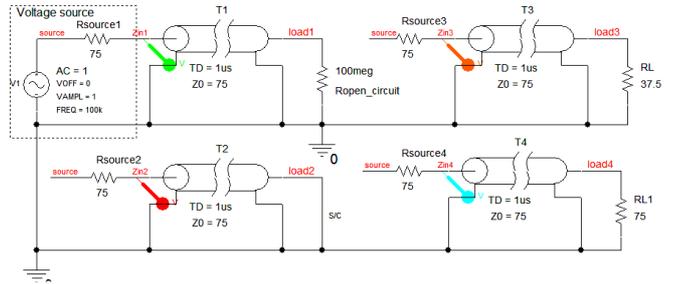$$\rho = \frac{VSWR - 1}{VSWR + 1} = \frac{|Z_L| - Z_0}{|Z_L| + Z_0} \qquad (4)$$

FIGURE 4
TRANSMISSION LINES TERMINATED WITH DIFFERENT LOADS

Figure 5 shows standing waves set up for each line terminated with different terminations. Note: there are no standing waves when the line is correctly terminated in a load whose value is the characteristic impedance, Z_o.
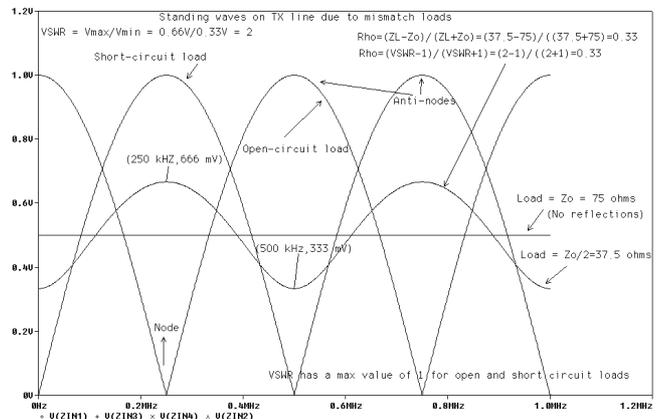

FIGURE 5
VOLTAGE STANDING WAVES FOR DIFFERENT LOADS

Other transmission line concepts, such as quarter-wave matching, and time domain reflectrometry, can be explored by PSpice in order to reinforce results as measured in a transmission line laboratory.

(3) PSPICE MACROS

A much under-used facility in PSpice is the MACRO. A macro can carry out a particular task, by passing variables to an equation created, and used, in Probe [7]. This example illustrates macro usage by plotting Gaussian probability distribution, for a binary data signal corrupting by noise. This probability distribution is described by the equation:

$$P_e(v_d, v_n) = \frac{1}{v_n \sqrt{2\pi}} e^{-(v_n - v_d)^2 / 2v_n} \qquad (5)$$

The mean value of the bipolar data signal without noise is $v_d = \pm 2V$, and v_n is the RMS value of the noise voltage. To plot the Gaussian distribution, centered on the mean of the data voltages of -2V and +2 V, we create two macros that run from the **PROBE\Trace\Macros** menu. The schematic

in Figure 6 shows an ASCII noise file, comprising time-voltage pairs created in Matlab, being added to the binary data simulated using a simple square wave pulse generator. Figure 7 show a +4 V signal representing logic one, and –4 V for logic zero. The period of the signal is 2 ms. The threshold (the decision level), for deciding whether 1 or 0 is present, is set to zero volts (i.e. the average of the two signals).
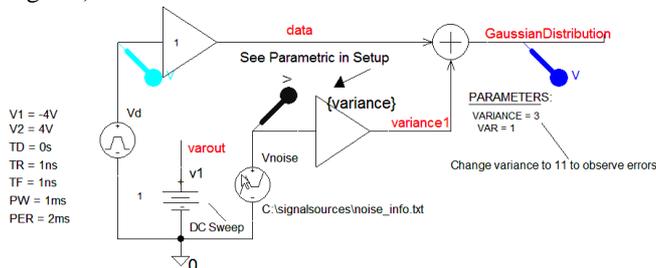

FIGURE 6
DEMONSTRATING BINARY SIGNAL PROBILITY DISTRIBUTION

The occasional positive noise peak signal causes the signal to dip below the zero threshold decision, and the negative noise signal to rise above the zero threshold level. This causes the receiver to misinterpret the input signal, and thus, over the 5 ms period shown, several errors will occur. To quantify these errors we need to look at the probability of these errors existing over a certain time period. Any text editor, such as **Notepad®,** can be used to create, and enter, the following expressions:

Gaussian1(vd,vn)=(1/(sqrt(rms(vn)*rms(vn)*2*pi)))*exp(-((vd-2)*(vd-2)/(2*rms(vn)*rms(vn))))

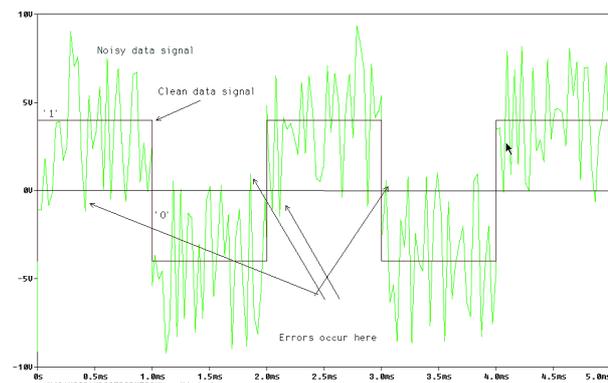**Gaussian2(vd,vn)=(1/(sqrt(rms(vn)*rms(vn)*2*pi)))*exp(((vd+2)*(vd+2)/(2*rms(vn)*rms(vn))))**

FIGURE 7
BINARY SIGNAL WITH NOISE

These two equations thus locate the two distributions around – 2 Volts and + 2 Volts, as shown in Figure 8.

(4) THE DISCRETE HILBERT TRANSFORM

Traditionally, Digital Signal Processing (DSP) principles are normally introduced to the student using Matlab/Simulink [1], [3], but here an alternative method models digital filter delays using a transmission line, or a Laplace part. The discrete Hilbert transform (HT) can be applied to introduce a

constant 90-degree phase shift over a band of frequencies. This has many applications in telecommunications transmission systems (e.g. production of a single sideband suppressed carrier modulation scheme i.e. SSBSC).
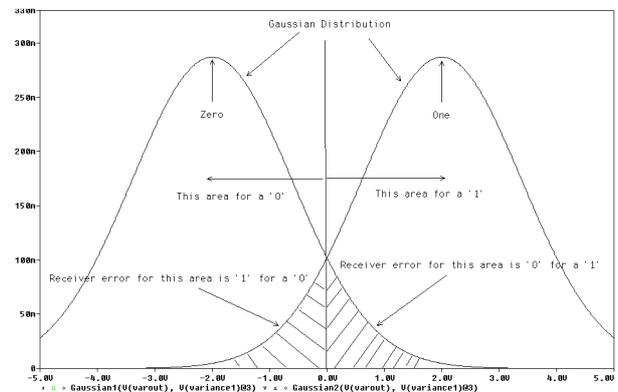

FIGURE 8
GAUSSIAN DISTRIBUTION

Figure 9 shows how a Hilbert transform is implemented using a high-order FIR filter, in quadrature with a delay $D = (N-1)T/2$, where T is the sampling period, and N is the filter order [1], [8]. Seventeen FIR filter coefficients are calculated, but half of them are zero. The FIR filter uses the Laplace analogue behavioral model (ABM) part, with the transfer function entered into the ABM numerator as a series of weighted exponential terms having a form such as 0.0909*exp(-s*T), and a unity denominator.
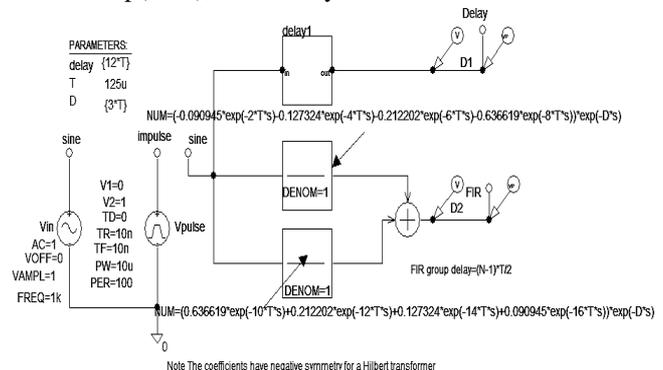

FIGURE 9
THE DISCRETE HILBERT TRANSFORM USING FIR FILTERING

Figure 10 plots the input impulse signal and corresponding impulse response (the impulse amplitudes are the filter coefficients).
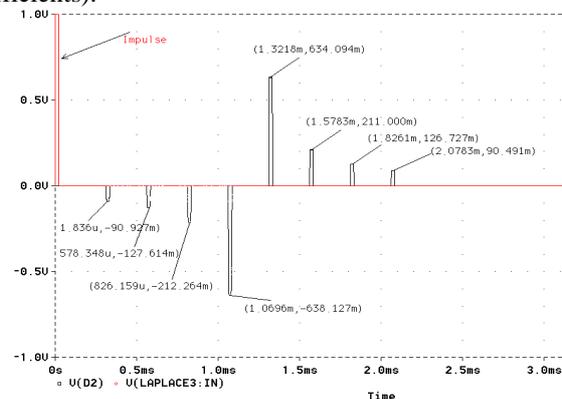

FIGURE 10
THE IMPULSE RESPONSE

Figure 11 shows the amplitude response with ripple in the passband region. The constant 90 degree phase shift over the desired frequency band is plotted using a pair of phase markers.
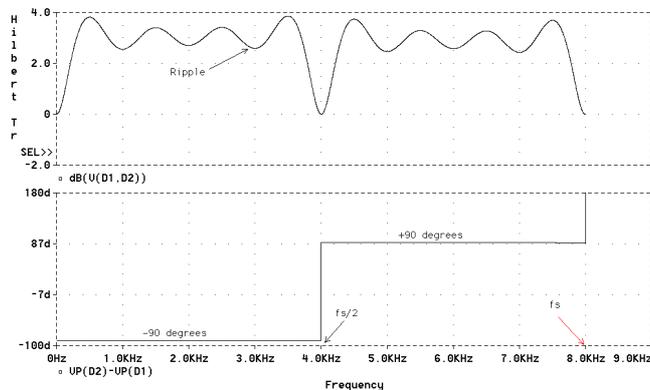

FIGURE 11
FREQUENCY AND PHASE RESPONSE

(5) MONTE CARLO ANALYSIS AND HISTOGRAMS

In the real electronics world, resistors, capacitors, operational amplifiers etc., possess non-ideal values. Thus, a 1 kΩ resistor, with twenty percent tolerance, could have a range between 800 to 1200 Ω. Sensitivity analysis predicts circuit behavior when using components with finite tolerances. This analysis is tedious but, thankfully, PSpice comes to the rescue. Figure 12 shows a sixth–order active low-pass active filter, where we place 20 % tolerance on all resistors. **Monte-Carlo** and **worst-case** analysis techniques are available in Probe, and, after ac simulation, we see in Figure 13 how the high tolerance components produces an unacceptable spread in the passband (or cut-off) frequency. The results can also be displayed in **Histogram** form. Note: To fix this problem whose cut-off frequency deviates from a desired value, we should ensure components have a one percent tolerance, or lower.
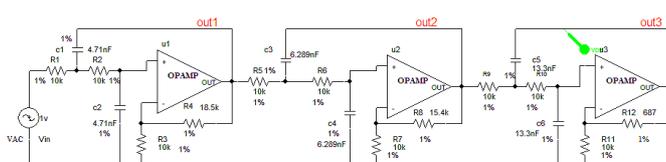

FIGURE 12
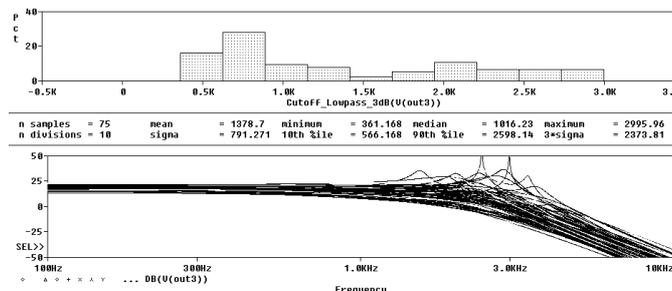SIXTH-ORDER ACTIVE FILTER WITH RESISTOR TOLERANCE

FIGURE 13
HISTOGRAM AND FREQUENCY RESPONSE

(6) DIGITAL COMMUNICATIONS APPLICATIONS

Figure 14 shows an application of hierarchical drawing construction to construct a sixteen-Quadrature Amplitude Modulation (16-QAM) system. This is tested using 'home-made instruments to apply noise to the final output signal lines, and a scatter meter to plot the constellation. Here, the input data, applied with a Filestim part, is split by a shift register into quad bits. The quad bits are then applied, using 2-to-4 level ABM parts, to generate In-phase and Quadrature signals using two orthogonal carriers. To plot a complete **VECTOR** diagram (a constellation plot), as in Figure 15, we need to connect the noise and scatter meters to each quadrature output and then change the horizontal axis after simulation [7]. The 16-QAM spectrum, shown in the right pane, is centered on the carrier frequency.

(8) CONCLUSIONS

A worrying trend in engineering education is a reduction in the time allotted to teaching fundamental principles, to accommodate ever-increasing amounts of software, programming, and protocols associated with information technology. This may produces students with an overall poorer understanding of good engineering practice and design techniques. The paper showed how PSpice can help students understand fundamental concepts, and do so with a minimum of mathematical rigor. Another benefit is in the area of system design. Here, the student can examine and simulate, individual sub systems and then consider the problems of matching when the sub-systems are connected together. It is often quite difficult in a real system, to measure input and output impedances and to see how a second stage can load down a first stage and change, the overall characteristics. This is easily done in PSpice and thus gives the student valuable experience not easily obtained in practice.

REFERENCES

[1] Bateman, Andrew and Paterson-Stephens, Iain. The DSP handbook, Prentice Hall PTR; CD-Rom edition (October 16, 2002)

[2] Feisel, Lyle D, Rosa, Albert J., Role of the Laboratory in Undergraduate Engineering Education, The Journal of Engineering Education, Jan 2005

[3] Mitra, K Sanjit, Digital Signal Processing. A computer-based Approach WCB/McGraw-Hill, 1st edition (1998)

[4] Reamon, Derek T. Sheppard, Sheri D. The role of simulation software in an ideal learning environment., Proceedings of DETC'97, 1997 ASME

[5] Tobin, Paul, PSpice for Circuit Theory and Electronic Devices. Morgan Claypool publishers, April 2007.

[6] Tobin, Paul, PSpice for Filters and Transmission lines. Morgan Claypool publishers, April 2007.

[7] Tobin Paul PSpice for Digital Communications Morgan Claypool publishers April 2007.

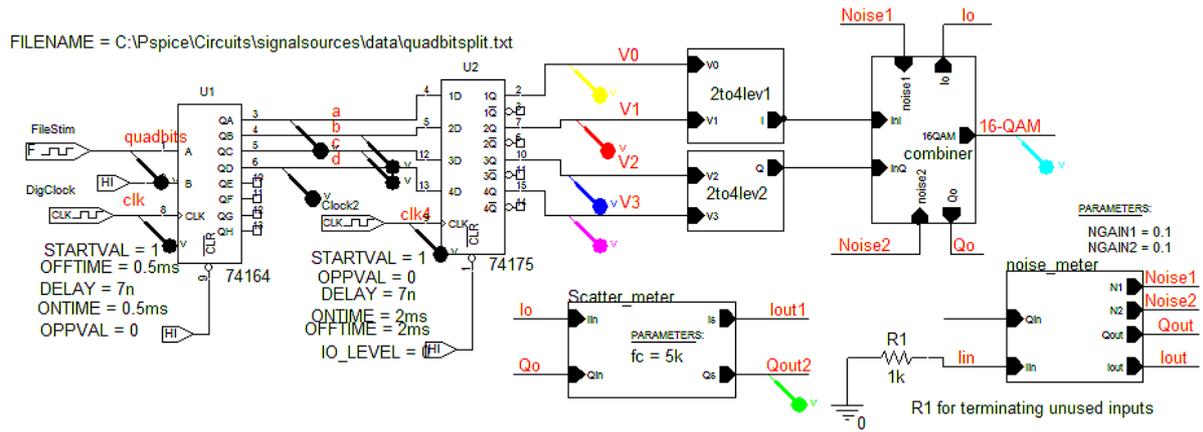[8] Tobin Paul PSpice for Digital Signal Processing. Morgan Claypool publishers, April 2007
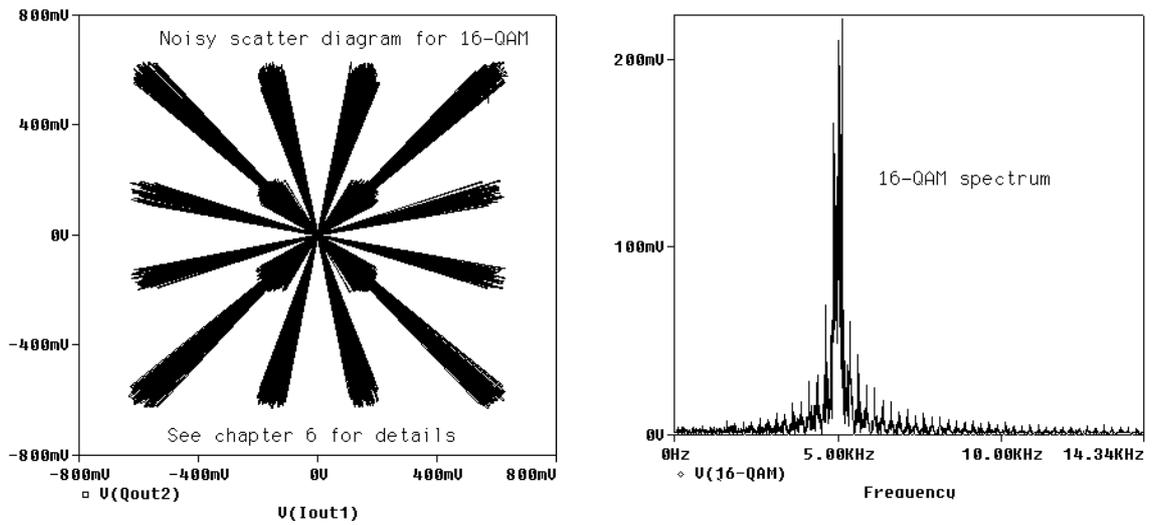
FIGURE 14
16-QUADRATURE AMPLITUDE

MODULATION

FIGURE 15
SCATTER DIAGRAM AND SPECTRUM FOR 16-QAM