The Role of PSpice in Analog and Mixed-Signal Simulation of Electrical Circuits

Y. Sari¹, A. Ferikoglu²

¹University of Sakarya, Turkey, sari@sakarya.edu.tr
²University of Sakarya, Turkey, af@sakarya.edu.tr

Abstract—Computer simulation is a common technique for predicting the real world behaviour of a circuit. This paper describes preparation and simulation of four analog and mixed-signal applications in Orcade-PSpice medium. The examples are selected considering their educational value. Simulation results have shown that Orcad PSpice is highly visual and effective for understanding the mathematical theory underlying the circuits.

Keywords—Simulation, Orcad PSpice, strain-gauge, dissipated power, multi-level signalling, delta modulator.

I. INTRODUCTION

Computer simulation is a common technique for predicting the real world behaviour of a circuit. Although simulation software only reflect the capability of the model used in the back-plane and they cannot substitute the real-time experimentation, they have proven educationally useful due to their easy to construct and visual properties. Orcad PSpice is a general purpose circuit simulator and one of various versions of SPICE family, capable of handling analog, logical and mixed-signal parts, circuits and systems [1,2]. It has been extensively used by Universities and semiconductor manufacturers as a CAD [3,4,5,6] tool because of its simplicity and effectiveness. Evaluation versions suited for educational purposes are available and can be found at the URL: www.pspice.com. Many parameters of circuits and devices can be profitably simulated using these free educational versions of PSpice.

PSpice simulations provide the advantage of the hierarchical circuit structures, where design takes place using sub-circuits. In the simulation procedure different approaches may be followed, namely, electrical component level, subsystem functional block level and higher system level comprising both of the previous ones [7].

This paper treats four educational examples, two about circuit theory and two about communication theory, selected due to their educational value.

II. WORKED EXAMPLES

Example 1:

The circuit in Figure 1a shows a Wheatstone bridge with two resistors, \( R_1 \) and \( R_2 \), representing strain-gauges. Obtain the output range if parameter \( d \) varies between (-0.01) and (+0.01).

Theoretical: Applying voltage divider rule yields the output voltage as,

\[
V_o = 10\left(\frac{R}{R+R_1} - \frac{R_2}{R_2+R}\right) V
\]

(1)

substituting \( R(1-d) \) and \( R(1+d) \), for \( R_1 \) and \( R_2 \), respectively , \( V_o \) becomes,

\[
V_o = 10\left(d^2/(4-d^2)\right) V
\]

(2)

According to (2), ± variation of \( d \) by 0.01 results in a change of 0 - 250 µV in \( V_o \).

PSpice solution: The voltage source is a DC one so DC analysis is needed together with a parametric sweep. However, since resistor variations are in opposite directions (one becomes more positive and the other more negative) two separate parametric sweep seem to be needed, which is not allowed with the student version of the Orcad PSpice. A simple manipulation puts the circuit in the form of Figure 1b, where the two variable resistors varies the same manner, with no change in the expression of output voltage. PSpice run with DC analysis (start value = end value = 10 V ) and parametric sweep for the variable resistor (from -10 to 10, taking arbitrarily \( R=1 \) KΩ) yields the output voltage variation of \( V_o \) as \( d \) varies between (-0.01) and (+0.01) according to Equation (2).

![Figure 1a: Wheatstone bridge circuit containing strain-gauges of Example 1.](image-url)
Example 2:
Calculate the long-time average power dissipated in the two circuits of Figure 2a, where all sinewave voltages have amplitude of 1 V and in circuit A both sources have a frequency of 1 Hz, and in circuit B one 1 Hz, and the other 2 Hz. Compare the results.

Theoretical: By straightforward power analysis, for the circuit A,
\[(V_{1A} + V_{2A})^2/R_A = [\sin(2\pi t) + \sin(2\pi t)]^2\]  \hspace{1cm} (4)
\[P_{RA}(\text{avg}) = 2W\]  \hspace{1cm} (5)

and, for the circuit B,
\[(V_{1B} + V_{2B})^2/R_B = [\sin(2\pi t) + \sin(4\pi t) + 2\sin(2\pi t)\sin(4\pi t)]\]  \hspace{1cm} (6)
\[P_{RB}(\text{avg}) = 1W\]  \hspace{1cm} (7)

This example numerically show that \(\text{avg}[\sin(at)\sin(bt)]=0\), for \(a\neq b\) and, \(= 1\), for \(a = b\).

PSpice solution: Time (transient) analysis for a long time compared to the longest period is needed. After a run of ten seconds application of the analog operator \(AVG(\ )\), which expects name of the variable to be averaged, yields the results of Figure 2b and Figure 2c. Since the operator takes the time shift average the instantaneous power curves approach to the values of 1 W for circuit A and 2 W for circuit B.

Example 3:
The third example is about multilevel signalling.

Theoretical: Higher bit rates may be transmitted over a certain communication channel using multilevel (M-ary) signalling. By the famous theorem of Shannon-Hartley, the maximum transmission rate, or the channel capacity, \(C\), is directly proportional to number of levels, \(M\), as,
The Role of PSpice in Analog and Mixed-Signal Simulation of Electrical Circuits

\[ C \sim \log_2 M \]  \hspace{1cm} (9)

where the number of levels \( M \) is related to bit number as,

\[ M = 2^n \]  \hspace{1cm} (10)

**PSpice solution:** A PSpice block-level design for illustrating multilevel signalling for \( M=4 \) is in Figure 3a. A two-input ABM part, together with a four-bit digital stimulator are used in the circuit schematic. In this conceptional scheme, the functional expression of the ABM part is written as,

\[ \text{if}(v(d_0)>0.1, \text{if}(v(d_1)>0.1, 4, 3), \text{if}(v(d_1)>0.1, 2, 1)) \]  \hspace{1cm} (11)

where \( V(a) \) and \( V(b) \) are input nodal names.

Easily obtained and highly illustrative input and output waveforms are provided in Figure 3b together with their FFTs in Figure 3c.

**Example 3:**

**Theoretical:** The circuit of Figure 4a contains a simple (one bit) version of the commonly used delta modulator. The comparator compares the input and output signals and produces an error signal, which triggers the D-flip-flop whose output integrated and feedback to the input, thus enforcing the output signal to track the input signal.

**PSpice solution:** The modulator in Figure 4a is constructed and run in PSpice. Components parameters appear beside them. The capacitor should be initialized with zero voltage by double clicking on the component and entering zero for the IC (initial condition). Similarly, flip-flop initial value should be set to zero by selecting successively simulation settings, options and gate-level simulation. The results of a 500ms time analysis, which well illustrates the theory, is given in Figure 4b.

---

**Figure 3a:** Functional block schematic of binary to \( M \)-ary (\( M=4 \)) converter of Example 3.

**Figure 3b:** Binary input signals to the binary-to-4-level converter \((d_0: 0111001; d_1:0010011)\) and output of the converter \((V(\text{out}): 00-10-11-10-00-01-11)\) of Example 3.

**Figure 3c:** FFTs of the inputs and output of the binary-to-4-level converter of Example 3.

**Figure 4a:** One-bit delta modulator schematic of Example 4.
Figure 4b: Input [V(d_1)], output [V(d_3)] and the feedback [V(d_2)] signal waveforms of the delta modulator of Example 4.

III. CONCLUSION

The educational role of the powerful simulation software Orcad PSpice in the analysis of analog and mixed-signal electrical circuits is investigated over four worked examples which effectively utilize the potential of the simulator. The simulation results being highly visual is observed to be congruent and illustrative to the theory underlying the treated examples.

REFERENCES

[1] PSpice User’s manual, OrCAD Corp. (Cadence Design Systems, Inc.)