AC 2008-995: THE UNIVERSITY OF TEXAS AT TYLER PSPICE ARCHIVE

David Beams, University of Texas at Tyler

DAVID M. BEAMS is an Associate Professor of Electrical Engineering at the University of Texas at Tyler. He received his BS and MS degrees from the University of Illinois at Urbana-Champaign in and the Ph.D. from the University of Wisconsin-Madison. He has had over 16 years of industrial experience in addition to his 10 years with UT-Tyler. He is a licensed professional engineer in Wisconsin and Texas and holds or shares four patents.

The University of Texas at Tyler PSpice Archive

Abstract

PSpice (Cadence, San Jose, CA), has become a *de facto* standard for courses in electric and electronic circuits. Its popularity stems from the ready availability of the evaluation (student) version and the inclusion of the evaluation version with a number of widely-used textbooks in these courses. Many textbooks also provide access to example circuit files either in CD form or through a companion web site. However, faculty at the University of Texas at Tyler have frequently found it valuable to develop their own analyses to better illustrate particular topics or to simulate circuits found in laboratory courses. These analyses include standard types of circuits (e.g., simple operational amplifier circuits, *LRC* circuits) and more-advanced ones (e.g., a current-feedback operational amplifier implemented at the transistor level). These circuit analyses had been accumulating over a period of some years; a recently-initiated effort aims to rationalize their organization and to document the resources available in the archive with the intention of making the archive publicly available. This paper describes the organization of the archive, the resources available, and its incorporation into the curriculum.

Introduction

PSpice is a *de facto* standard circuit-analysis package for courses in electric and electronic circuits. Its popularity is attested by the fact that searching the "books" category of Amazon.com for the keyword "PSpice" produced 520 "hits." (By comparison, a similar search for "Electronics Workbench" produced 312 "hits"). The evaluation version of PSpice is frequently bundled with textbooks. Examples of these include texts on general circuit analysis [1], general electronics [2,3], communication systems [4,5], power electronics [6], and photovoltaic systems [7]. There are also textbooks intended as primers or introductions to PSpice itself [8,9,10,11]. Publishers often support web sites that provide worked-out examples from the textbooks.

However, it is often easier to create one's own additional PSpice examples rather than to sift through the mountain of available resources. This was the genesis of the PSpice Archive of the University of Texas at Tyler. The collection began as an *ad hoc* effort to provide educational resources to students in courses in electronics and instrumentation systems and had no particular organization. As the number of examples grew, however, the need became apparent to give the archive a rational structure and logical organization, and it has since been organized into categories (e.g., diodes and applications, MOSFETS, and amplifier circuits).

Organization of the PSpice Archive

The archive is a work in progress and is continually being modified. At the end of the fall semester of 2007, however, the PSpice Archive was organized into the categories outlined in Table 1 below.

Table 1. Organization of the PSpice Archive (December, 2007)

Section	Description of available analyses	Total analyses
Diodes	*I-V* curves vs. temperature; half- and full-wave rectifiers; clamp circuit; voltage doubler; zener voltage regulator; triangle-to-sine converter; varactor diode.	9
BJTs	*I-V* characteristics; bias circuits; common-emitter and common-base amplifiers; emitter follower; cascode amplifier.	6
MOSFETs	*I-V* characteristics; MOS digital inverters; common-gate and common-source amplifiers; source follower	5
Amplifiers	Midband and high-frequency analyses of the BJT differential pair; instrumentation amplifier; op amp macromodels; bipolar op amp (constructed in EENG 4109 laboratory); CMOS op amp; current-feedback op amp; ac model of the emitter follower.	9
Oscillators	Astable multivibrator; Colpitts oscillator; Pierce oscillator; phase-shift oscillator; Wein bridge oscillator; crystal oscillator	7
Active filters	Two-pole low-pass filter (LPF); 5th-order Butterworth LPF; KHN biquad circuit (state-variable filter); Tow-Thomas biquad; capacitance multiplication and inductor synthesis with a generalized impedance converter (GIC); bandpass and band-reject filters using GICs.	10
Logic	Transient analysis of PMOS, NMOS, and CMOS inverters; pseudo-NMOS inverter; pass-transistor logic; edge-triggered D-type flip-flop	5
General circuits	Current mirrors; two-port network; Wheatstone bridge; quartz crystal ac model; impedance matching with pi-networks; three-terminal linear voltage regulator (similar to LM340); multiplying and switching phase-sensitive demodulation; ground-referenced current source; Fourier-series analyses	17
	Total:	68

All schematics for the circuits listed above were drawn with OrCAD Capture and simulated with the evaluation version of PSpice 9.2. It was vital that analyses be within the capabilities of the evaluation version of PSpice since few students have access to the professional version. This restriction imposes limits on the models available, but this also provides an opportunity for illustrating to students how to make and edit their own models.

Example: *I-V* characteristics of MOSFETs

The following examples illustrate some of the analyses available within the PSpice archive. Figure 1 below from the MOSFETs section of the archive is for determining the *I-V* characteristics of MOSFETs. The two MOSFETs have electrical characteristics given by Sedra and Smith typical of a 5μm CMOS process[12]. This particular circuit includes eight separate dc sweep analysis profiles, four for each device (NMOS and PMOS):

- determination of I_D vs V_{DS};
- determination of transconductance g_m;
- determination of small-signal channel resistance r_d;
- determination of the body effect on I_D vs. V_{DS} characteristics.

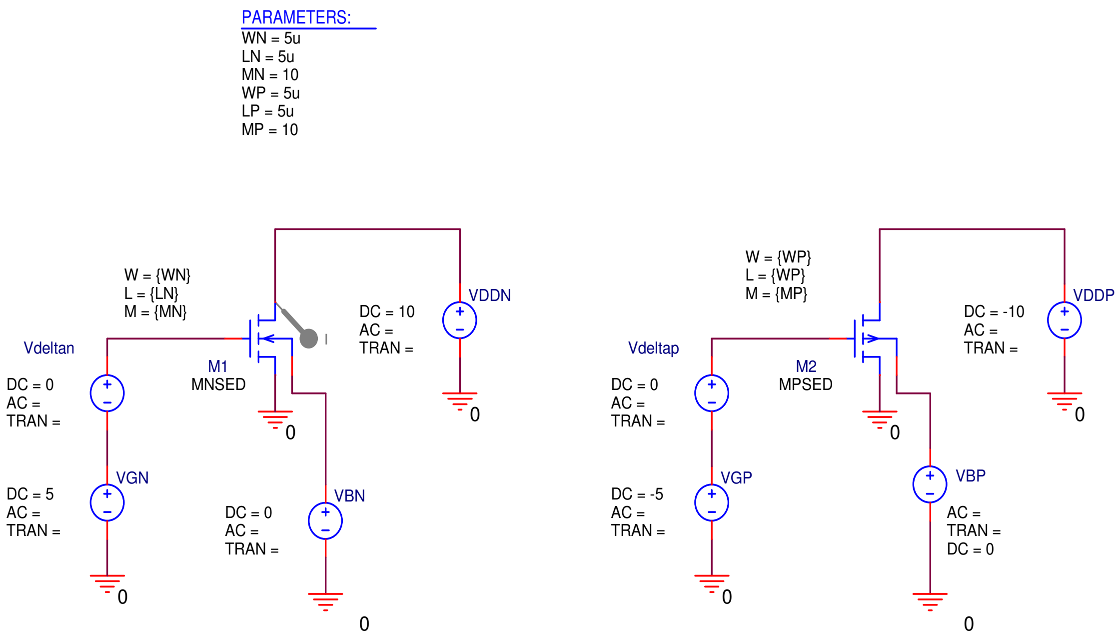

Fig. 1. PSpice archive circuit for determination of electrical characteristics of MOSFETs.

Figure 2 shows the results of simulation of the I_D vs V_{DS} characteristics of the NMOS device with gate-to-source voltages swept from 1.5V to 5.0V in increments of 0.5V.

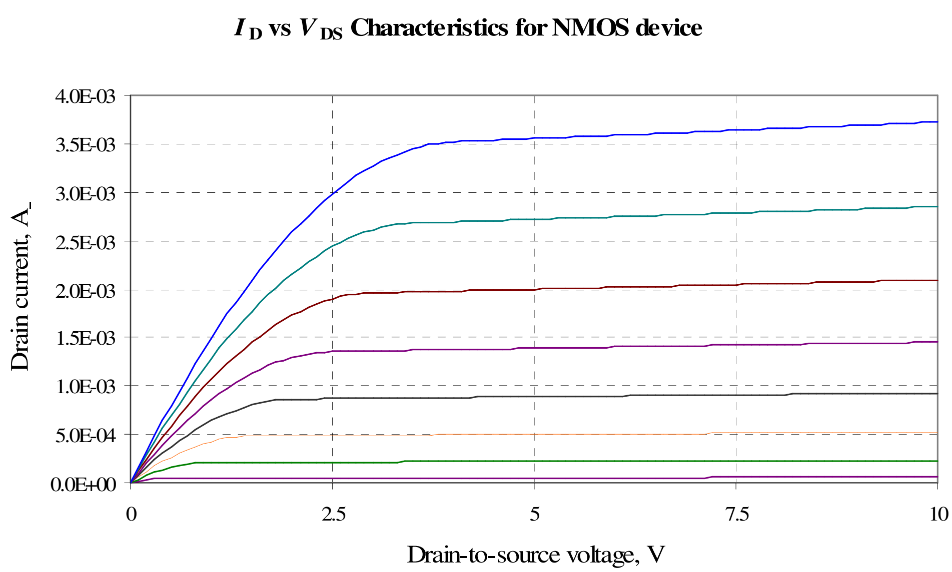

Fig. 2. Drain current vs. drain-to-source voltage characteristics of the NMOS transistor in Fig. 1. Results were copied from the Probe graphics post-processor into Excel for graphing.

Figure 3 shows results of the body effect in the NMOS transistor. The gate-to-source voltage (VGN in Fig. 1) was fixed at 5.0V and the drain current was recorded as the drain-to-source voltage (VDDN in Fig. 1) was swept from 0 to 10V for substrate-to-source voltages (VBN in Fig. 1) from 0V to –2.5V in increments of –0.5V.

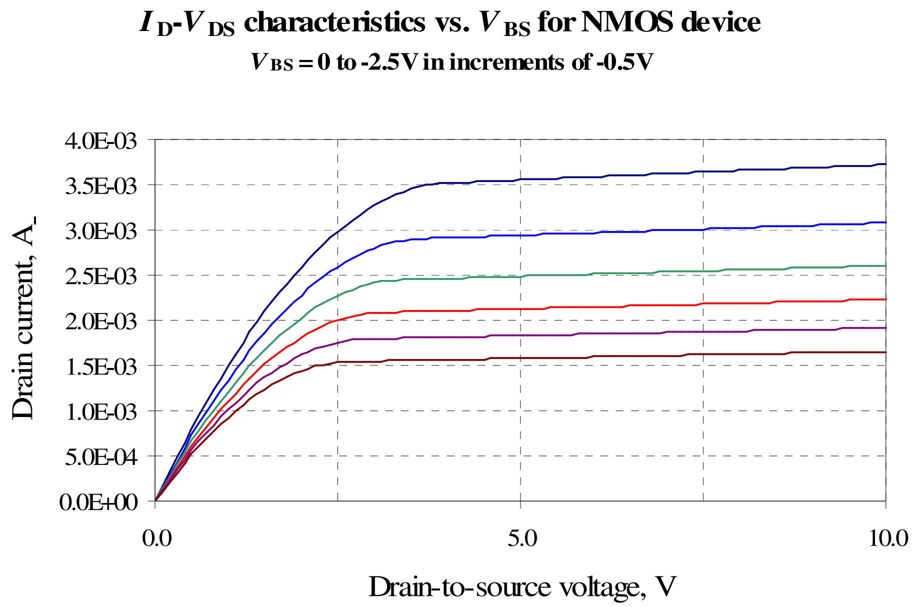

Fig. 3. Illustration of the body effect in the NMOS device of Fig. 1. The traces show drain current vs. drain-to-source voltage characteristics as body-to-substrate voltage varies in increments of –0.5V from 0V (top trace) to –2.5V (bottom trace).

Example: open- and closed-loop analysis of a Pierce oscillator

The Pierce oscillator shown in Fig. 4 is found in the Oscillators section of the archive. This circuit consists of a CMOS inverter operating as a linear amplifier and a feedback network consisting of capacitors C1 and C2 and inductor L1.

Two simulation profiles have been created for this circuit. One profile performs ac analysis from 100kHz to 100MHz with the feedback loop open to determine whether the loop gain meets the Barkhausen criterion for oscillation at any frequency. The other performs transient analysis with the feedback loop closed to determine whether the circuit manifests steady-state oscillatory behavior. It is of particular pedagogical interest to compare the findings of the ac analysis of loop gain with the transient response. The ac analysis profile includes a parametric sweep in which the value of parameter TRAN in Fig. 4 is set to a value of 0. This causes the computed value of resistor R7 to become very small (10mΩ), short-circuiting the output of the CMOS inverter to ground through capacitor C6 (whose reactance is negligible at the analysis frequencies). At the same time, computed parameter AC takes on the value of 1 causing resistor R6 to become very large (10MΩ). This opens the feedback loop. Current source I2 provides the ac stimulus; the response is the current flowing through C6. By giving I2 a value of 1A, the magnitude and phase of the current in C6 are numerically equal to the magnitude and phase of the loop gain.

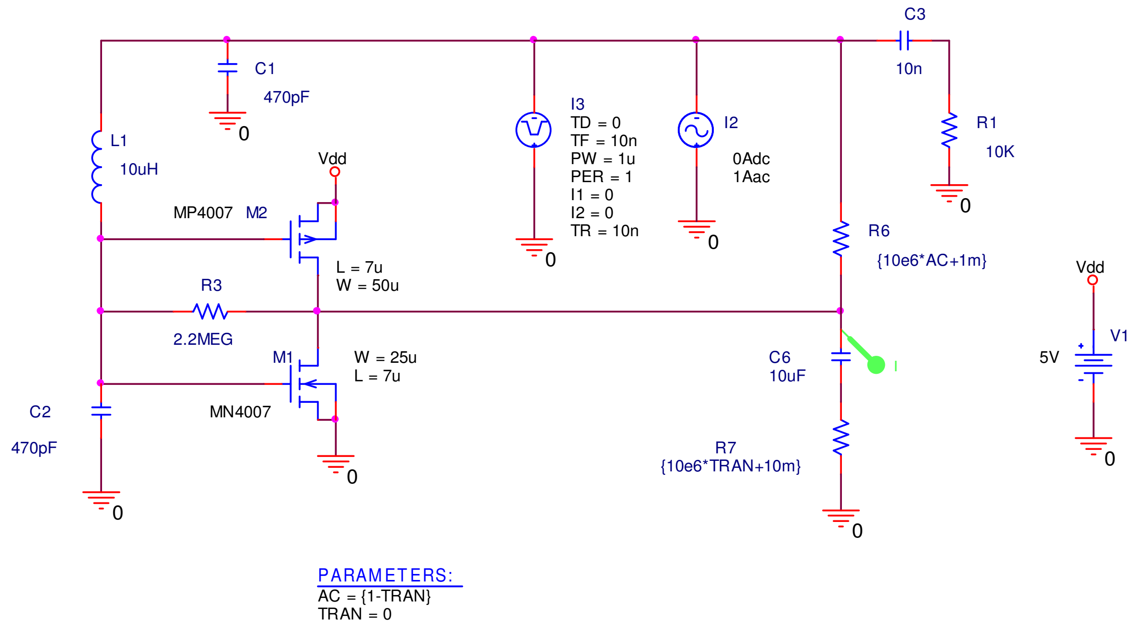

Fig. 4. Pierce oscillator circuit based upon a CMOS inverter. The MOSFET models were derived from laboratory measurements of CD4007 devices.

Figure 5 shows the loop gain of the Pierce oscillator. The phase of the loop gain passes through 0° at 3.277MHz; at this frequency, the magnitude of the loop gain is 8.85dB.

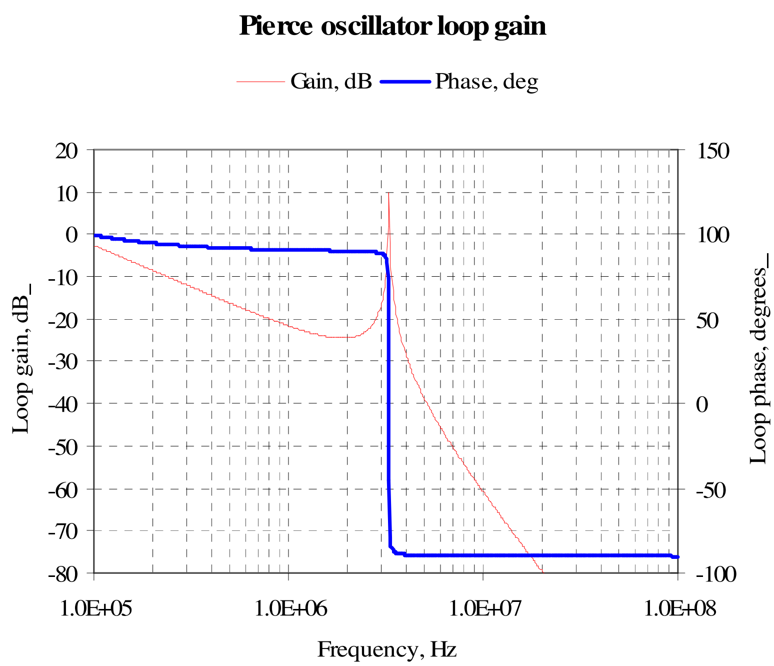

Fig. 5. Loop gain plot produced by ac sweep analysis of the Pierce oscillator of Fig. 4. The plot indicates that oscillation should take place at a frequency of 3.277MHz when the loop is closed.

Selecting the transient analysis profile sets parameter TRAN to 1 and causes parameter AC to take on the value of 0. These make the value of R7 very large (10MΩ) and the value of R6 very small (1mΩ); the ac short-circuit to ground through C6 and R7 is removed and the feedback loop is closed. A transient current source (I3 in Fig. 4) is used to provide initial energy to start the oscillation. Figure 6 shows the results of transient simulation of the circuit of Fig. 4. Measurement of the period of the sinusoidal waveform in Fig. 6 gives an output frequency of 3.235MHz, which agrees well with the oscillation frequency predicted by ac analysis of the loop gain. The analyses of Figs. 5 and 6 may be repeated with various values of components (e.g., load resistor R1 in Fig. 4) or FET parameters (e.g, channel width W) to illustrate the application of the Barkhausen criterion to determine under what conditions this circuit will become a self-sustaining oscillator.

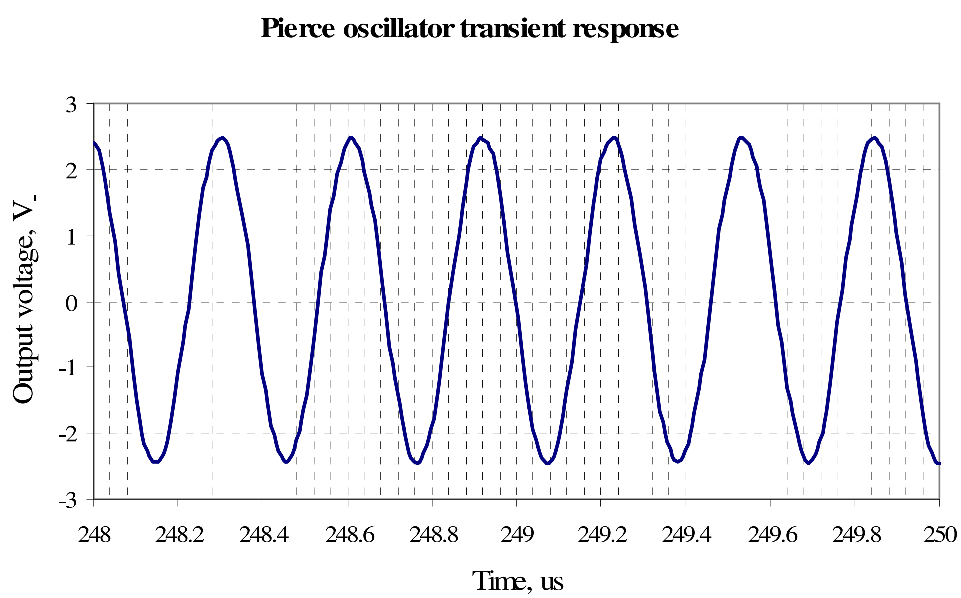

Fig. 6. Steady-state transient response of the Pierce oscillator of Fig. 4.

Example: simple operational amplifier

One of the experiments in EENG 4109 (Electronic Circuit Analysis II Laboratory) is the construction and test of a simple operational amplifier using LM3086 matched-transistor arrays. A PSpice model of this amplifier whose schematic is given in Fig. 7 was developed for comparison with measured performance. The model parameters for the LM3086 and 2N4401 devices were obtained from commercial on-line sources and the PSpice model editor was used to create the LM3086 and Q2N4401 models from the generic QBreakN NPN transistor model. The circuit of Fig. 7 shows the operational amplifier configured with external resistors R7 and R8 as a noninverting amplifier with a nominal closed-loop gain of +11. The signal voltages for the amplifier are either a 20kHz square wave or a sinusoidal ac source. The simulation profiles created for this circuit are ac response and transient response. Parametric sweep is enabled in both profiles to perform the analyses with values for compensation capacitor C1 of 1pF, 10pF, and 100pF.

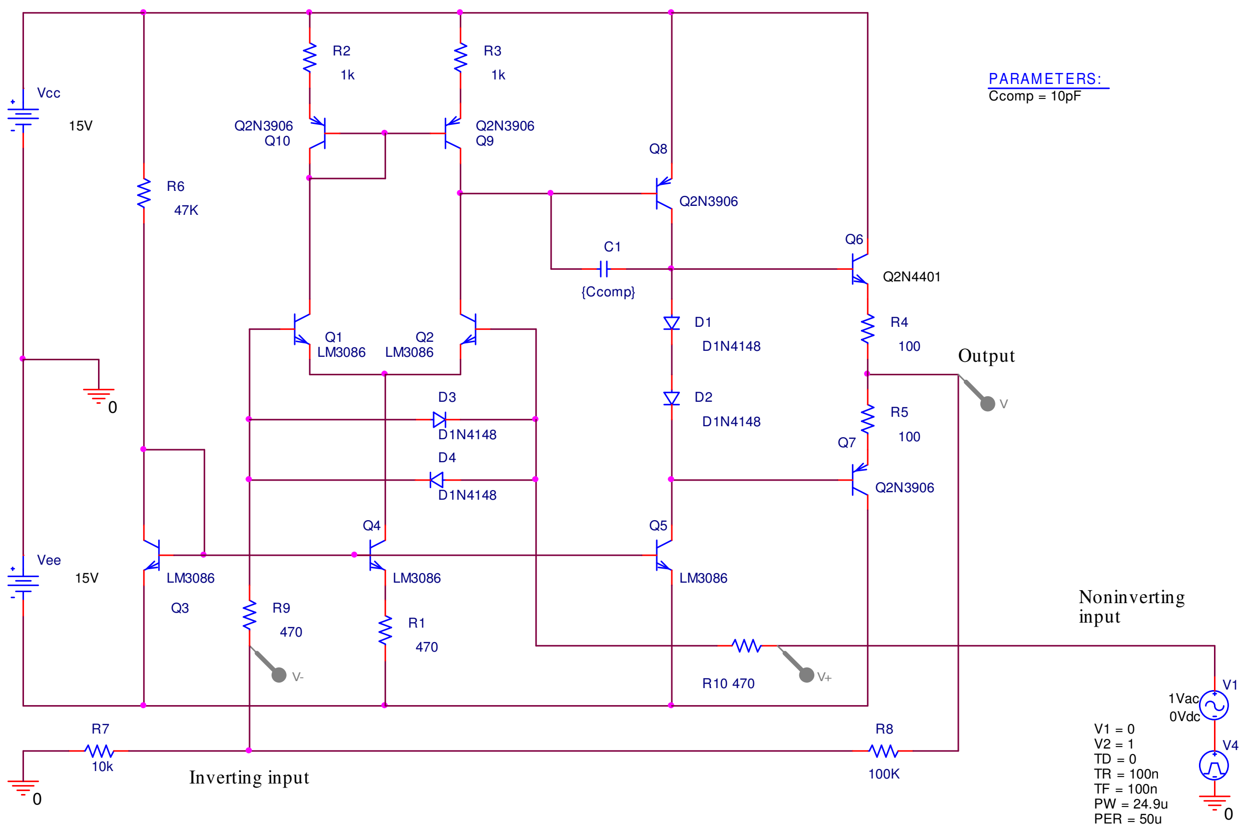

Fig. 7. Schematic diagram of a simple operational amplifier constructed in EENG 4109. Also included is a resistive feedback network (R7 and R8) and driving-point sources for ac and transient analyses.

A command file for the Probe graphics post-processor displays the open-loop voltage gain of the operational amplifier. Fig. 8 shows the magnitude and phase of the open-loop voltage gain of the circuit of Fig. 7 with a compensation capacitor C1 of 10pF. The simulation gives an approximate dc open-loop gain of 87dB (22,000) with a gain-bandwidth product of 26MHz. The dc open-loop gain measured by experiment is typically close to this value, although lower gain-bandwidth products are usually noted (typically 18MHz). Figure 8 also shows that the operational amplifier should be stable at unity gain with a gain margin of approximately 9dB.

Computation of open-loop gain vs. frequency allows the determination of closed-loop phase and gain margins, and these may in turn be related to transient response. Figure 9 shows the predicted transient response of the simple operational amplifier with a compensation capacitor of 10pF configured as a nonverting amplifier with a nominal gain of 11. The transient response shows little overshoot or ringing, commensurate with an estimated gain margin in this circuit of 29dB.

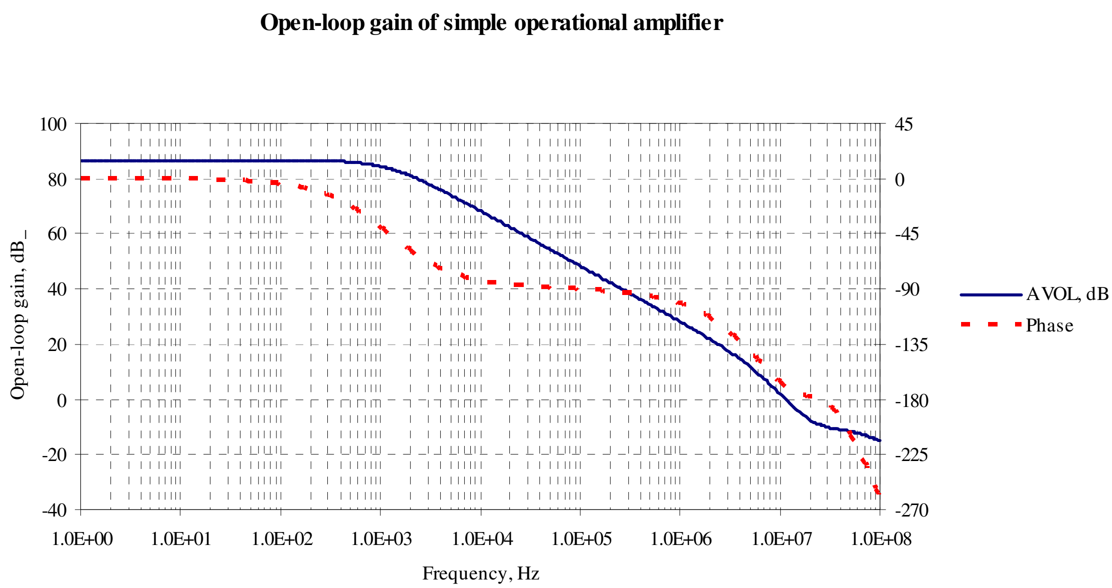

Fig. 8. Computed open-loop voltage gain vs. frequency for the simple operational amplifier of Fig. 7 with a compensation capacitor of 10pF. This plot shows that the simple operational amplifier with unity feedback would have a gain margin of 9dB and would be unity-gain stable. This agrees with laboratory experience.

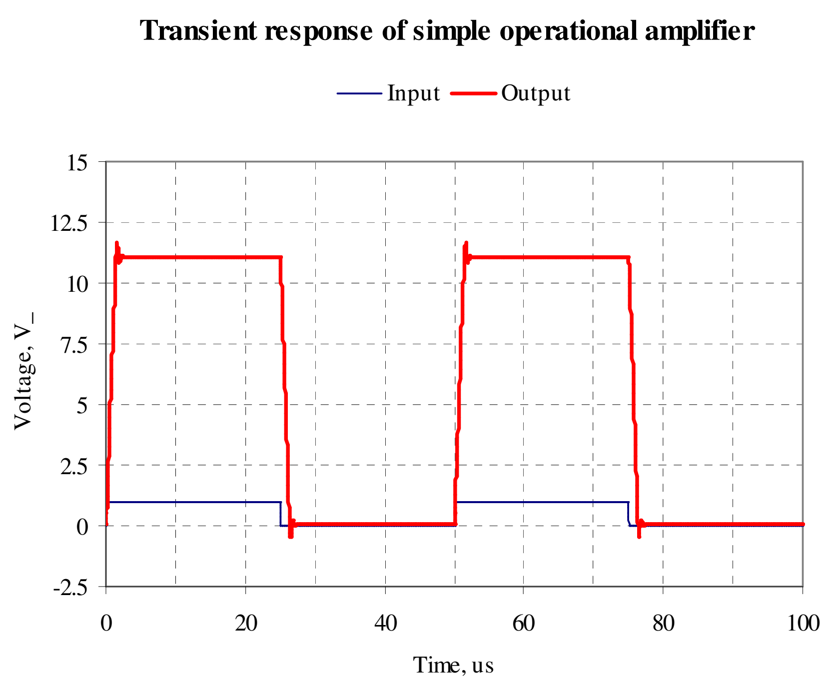

Fig. 9. Transient response of the simple operational amplifier configured as a noninverting amplifier with voltage gain of +11. The open-loop gain vs. frequency of the operational amplifier is shown in Fig. 8.

Curricular use

The analyses represented in the PSpice Archive have been accumulated over a period of some years, but there was no effort to systematize the circuits until 2007. The PSpice archive has since been integrated into both EENG 3306 (Electronics I) and EENG 4309 (Electronics II) and their companion laboratory courses. Students find the archive to be a valuable asset; for example, 10 students in Electronics I in the fall semester of 2007 gave the PSpice archive as an instructional resource a mean rating of 2.6 on a scale from 0 (useless) to 3 (essential). Assessment data from EENG 4309 will be obtained at the end of the current semester.

It is not the intention to present the PSpice Archive as a closed canon; it instead should be regarded as an ongoing work-in-progress. To this end, additional circuits and additional analyses of existing circuits are solicited, and submissions included in the PSpice Archive will be credited to their originators.

Conclusion

The PSpice Archive has been developed to illustrate both a wide range of circuits and analysis techniques. It has shown itself useful as a pedagogical tool in demonstrating both the principles underlying the circuits and techniques for using PSpice. The archive is a work in progress and is constantly under revision and expansion, and example circuits from other contributors for inclusion in the archive are welcomed. Index files (in the form of Word documents) are being added to the archive sections to explain what circuits and analyses are available. The current edition of the PSpice Archive is available for download as .zip files from the following URL: http://ee.uttyler.edu/David_Beams/Projects/pspice archives/PSpice Archives.htm

Bibliography

1. Nilsson, J., and Riedel, S. *Electric Circuits, 8th Edition.* Prentice-Hall, 2007.
2. Hambley, A. *Electronics, 2nd Edition.* Prentice-Hall, 2000.
3. Sedra, A., and Smith, K. *Microelectronic Circuits, 5th Edition.* New York: Oxford University Press, 2003.
4. Tobin, P. *PSpice for Digital Communications Engineering.* Morgan and Claypool, 2007.
5. Tobin, P. *PSpice for Analog Communications Engineering*. Morgan and Claypool, 2007.
6. Rashid, M., and Rashid, H. *SPICE for Power Electronics and Electric Power, 2nd Edition.* Boca Raton, FL: CRC Press, 2006.
7. Castaner, L., and Silvestre, S. *Modelling Photovoltaic Systems Using PSpice.* John Wiley, 2003.
8. Rashid, M. *Introduction to PSpice Using OrCAD for Circuits and Electronics, 3rd Edition.* Prentice-Hall, 2003.
9. Nilsson, J., and Riedel, S. *Introduction to Pspice Manual: Electric Circuits : Using Orcad Release 9.1.* Prentice-Hall, 1999.
10. Tront, J.G., and Tront, J. *PSpice for Basic Circuit Analysis*. McGraw-Hill, 2004.
11. Attia, J.O. *PSPICE and MATLAB for Electronics: An Integrated Approach*. Boca Raton, FL: CRC Press, 2002.
12. Sedra, A., and Smith, K., *op. cit.*, p. 355.