

Intusoft Newsletter

Personal Computer Circuit Design Tools

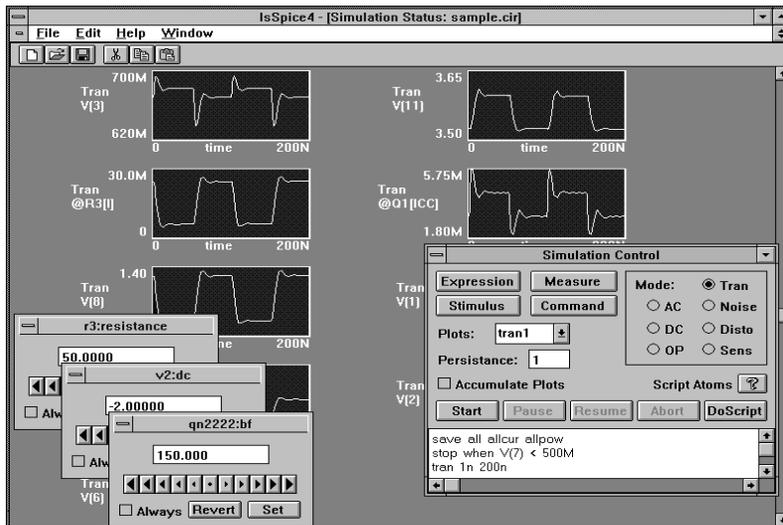
November 1993 Issue



Copyright © Intusoft, All Rights Reserved

(310) 833-0710 Fax (310) 833-9658

Interactive SPICE



Introducing IsSPICE4, a totally new version of SPICE. It is unlike anything you have run before. For years CAE tool developers have struggled to recreate the workbench on a computer. Now, with the advent of fast desktop PC's, this endeavor has been finally achieved by Intusoft. The new IsSPICE4 program is the first and only truly interactive version of SPICE.

IsSPICE4 is part of Intusoft's new ICAP Design Series. Two items give IsSPICE4 an

continued on page 33-2

Special Offer for the NEW ICAP/4 Windows, NT, or MAC

The ICAP/4Macintosh will be the first system available on Jan 18, 1994. ICAP/4 Windows and NT will be available April 4, 1994. As a special limited time offer, customers that purchase any current ICAP/4 system between Dec. 6, 1993 and April 4, 1994, will receive the upgrade to the new version for FREE.

In This Issue

- 2 New Interactive SPICE IsSPICE4 for Windows
- 5 SPICE Benchmarks
- 6 Modeling Sensors
- 12 The Modeling Corner: New Asia-Pacific Parts Variable Phase Source
- 13 Philips adds BiCMOS

New SPICE Goes Interactive

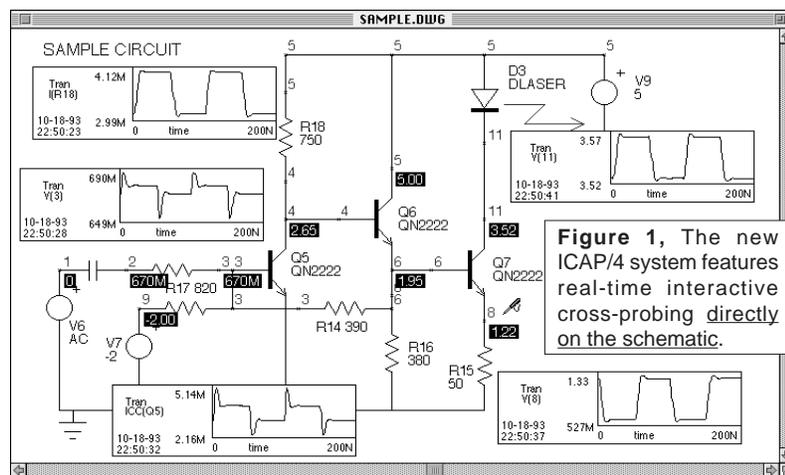
continued
from page
33-1

unprecedented simulation advantage. First, the designer can hand tweak any circuit parameter, while the simulation is running, and watch the resulting waveform changes. It's like having your hand on a potentiometer and watching an oscilloscope at the bench. Second, with the SPICENET schematic entry tool, you can peek and poke at any node or component and see the simulated waveforms during or after the simulation has run. This probing occurs directly on the schematic. No window switching to a post processor is needed. If you are using another schematic entry tool don't worry, IsSPICE4 is going to be integrated into your schematic soon. In the meantime, your schematic outputs a SPICE netlist that IsSPICE4 can use, still allowing the interactive simulation features to be totally utilized.

IsSPICE4 gives you an added speed advantage. Because it runs continuously, you don't have to restart the program, re-read the input netlist, or perform a DC operating point. These features, along with the best 32-bit compilers and latest Berkeley SPICE code make IsSPICE4 the fastest SPICE simulator on the market. Benchmarks are shown in Table 1.

Real-Time Interactive Cross-Probing and Viewing

IsSPICE4 allows all waveforms to be viewed in real time. To view a node voltage simply click on the node with the new probing tool as shown in Figure 1. Clicking on a component causes the current in the component to be displayed. A new "save all allcur allpow" command allows all the voltages, currents, and device power dissipations to be available. Waveforms may be added to the IsSPICE4 display, shown in Figure 3, at any time.



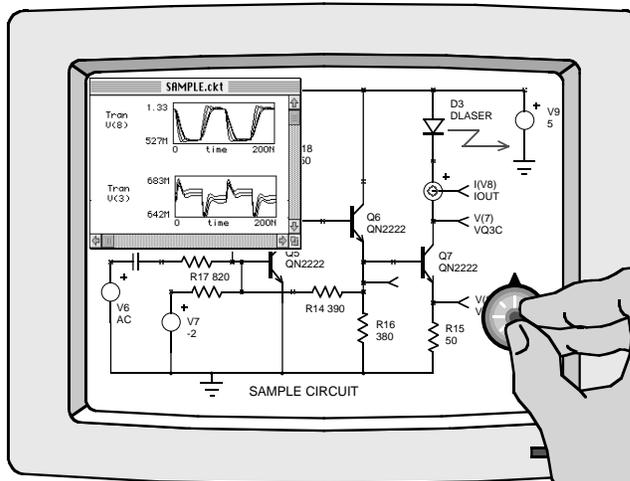
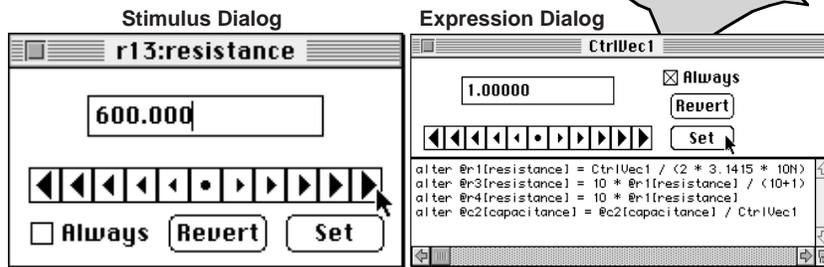


Figure 2, With the new Stimulus dialog you can interactively hand tweak any parameter value and watch the resulting circuit performance variations.

Circuit values can be changed individually or swept in groups, all while the simulation is running.



Interactive Component Variation - Singly or In Groups

Virtually any circuit parameter can be varied directly from the schematic or the simulator. When the Interactive Stimulus tool is selected you can click on any device. The Stimulus dialog will be displayed with the component's value as shown in Figure 2 bottom left. A new value can be entered and a simulation automatically re-run. If the "Always" button is checked, the analysis will be immediately run as soon as the value is changed. If the mouse button is held down the parameter will be changed and a new analysis run as soon as the current analysis is finished. In this way, it is possible to control a circuit variable and watch the waveforms change. Any cross-probed waveforms on the schematic will be instantly updated in real time.

The Expression dialog, shown in Figure 2 bottom right, works in a manner similar to the Stimulus dialog. However, several circuit variables may be swept in tandem. The circuit parameters may be independent or functions of other variables. Using this dialog, curve families can be instantly built up in INTUSCOPE.

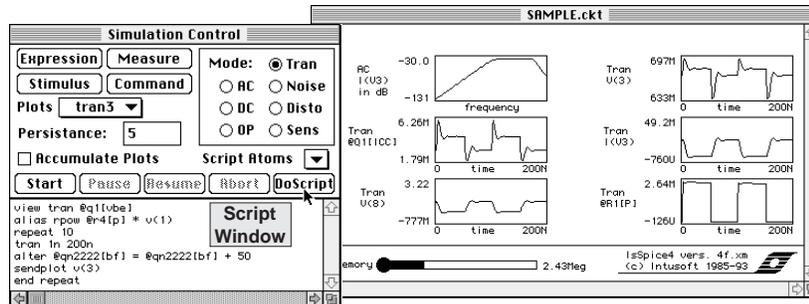


Figure 3, The IsSPICE4 simulation control dialog (left) allows you to interactively direct the flow of the simulation. The real time display (right) provides real-time access to all circuit waveforms.

Simulation Control Dialog - Figure 3

The Start, Pause, Resume, and Abort buttons are used to interactively control the IsSPICE4 simulation. Any of the powerful functions offered by IsSPICE4, such as Simulation Breakpoints or other analyses, may be typed into the script window and interactively executed. If you want a different transient analysis you don't have to restart the simulator. You just type in what you want and run it. The Script Atoms pop-up contains all of the available Interactive Command Language functions that can be used to build simulation scripts and test procedures. The Plots pop-up window is used to instantly recall waveforms from past simulations for comparison.

INTUSCOPE and New Model Libraries

INTUSCOPE has been upgraded as well. It now uses extended memory to view large waveforms and has access to all the waveforms from past or current simulations. Several scaling improvements have been made including: Linking of Y and X scales for all waveforms on a graph, tiling of waveforms, and a new algorithm for better offset calculations. The new ICAP systems include part libraries with over 5000 models.

RISC vs. CISC, Caching, and Compilers

The tremendous speed increases brought about by the Pentium, Alpha and MIPS computers have enabled interactive SPICE simulation to become a reality. However, there still remains many obstacles to getting the most out of each machine. As we developed the Windows NT applications, it became evident that application speed was not increasing as fast as clock speed.

Our first encounter was with a 486/66 which ran large circuits no faster than a 486/33. As it turns out, SPICE places extraordinary demands on a computers' memory system when large circuits are simulated. The assumption that successive memory



SPICE Benchmark Runtimes

Table 1, Partial list of the benchmark circuits used to test the speed and convergence of various SPICE programs. Index is compiled from a suite of 28 circuits. Long Index is based on LONG time only. (Times are in seconds)

Software Hardware Op. Sys.	2G.6 Vax8700 VMS	IsSpice4 Alpha/150 NT	IsSpice4 MIPS/100 NT	IsSpice4 Pentium/60 NT	IsSpice3 Pentium/60 DOS	IsSpice3 486/33 DOS	PSpice®/16 486/33 DOS	Number of Devices
Standard Benchmark Circuits								
Diff. Pair	1.64	0.42	0.58	0.47	0.77	2.65	4.12	12
RCA3040	4.66	0.68	1.10	0.85	1.10	4.02	6.76	26
UA741	11.74	0.95	1.82	1.47	1.75	5.83	13.02	41
MECL Gate	3.61	0.50	0.87	0.75	0.87	2.97	5.05	35
DC Convergence Test Circuits								
Flip-Flop	0.20	0.05	0.07	0.05	0.05	0.33	0.55	17
74STTL	3.30	0.48	0.80	0.67	0.82	2.92	4.67	18
74LTTL	3.46	0.47	0.75	1.37	0.77	2.87	5.33	16
Transient Test Circuits								
Simple RC	0.49	0.20	0.15	0.13	0.17	0.93	0.93	4
ECL Gate	1.51	0.30	0.37	0.35	0.43	1.70	2.31	14
Schmitt Trig	1.95	0.32	0.43	0.38	0.48	1.92	2.80	15
Intusoft General Test Circuits								
SAMPLE	3.83	0.60	0.95	0.77	0.98	5.00	5.60	15
MOS Chain	4.93	1.48	3.23	2.47	2.57	8.47	17.68	12
Elliptic Filter	12.21	1.07	1.62	1.38	1.58	7.03	22.14	118
DIV/4	47.03	5.70	12.08	9.38	9.77	27.80	78.22	256
LONG	2311.40	329.52	760.70	498.32	506.90	1272.80	2854.72	2967
Index	.76	5.81	3.23	3.88	3.45	1.00	0.46	
Long Index	.55	3.86	1.67	2.55	2.51	1.00	0.45	

PSpice is a registered trademark of Microsim Corp.

accesses are to nearby address locations no longer holds. Interestingly, the same thing happens for programs written in object oriented languages. When memory access is over a large spread of address space, the cache system will eventually fail and data will have to be fetched from slower dynamic RAM.

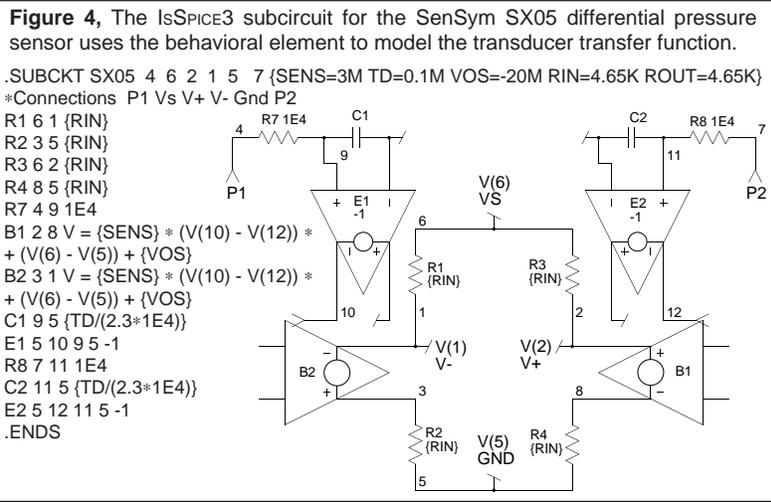
In the case of the DEC Alpha running at 150 MHz, data access can be up to 20 times slower than its instruction execution rate. Moreover, we found that compilers for the newer machines were not executing standard library functions as fast as the compilers available for the older architectures. Math library functions are especially important for SPICE; however, almost all library functions ran a factor of 2 slower than the raw machine speed would indicate.

The conundrum for the "power" user - what computer is best - is now exacerbated by the memory architecture, operating system, and compiler design. Our advice is be aware. Benchmark programs may not be a fair indication of the speed at which SPICE will run.

Modeling Sensors

The ability of IsSPICE to simulate mixed domain systems containing electrical, mechanical, physical, and mathematical elements is well documented. With behavioral models, Laplace functions, and a variety of electrical analogs as modeling tools, IsSPICE can lead to improved understanding of the interfaces between various domains. Simulations should be kept as simple as possible in order to limit run time and analysis complexity. Normally, it is recommended that devices that convert from a physical domain to an electrical domain be replaced by a signal generator. For instance, simulating the transfer function of a thermocouple (temperature in-voltage out) is not necessary unless the output of the system is a temperature that is destined to be feed back as an input to the thermocouple. A piece-wise linear voltage source would simply be used to emulate the output voltage of the thermocouple and eliminate the transfer function. In many applications, however, sensors are required to convert from one domain to the electrical domain and the transfer function is of great interest. In this application note we will explore how both pressure and Hall effect sensors can be modeled and show some example applications of simple mixed domain systems.

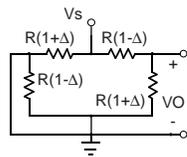
IC fabrication technology and traditional sensing mechanisms have been combined to create reliable, high performance sensors. For instance, advances in semiconductor materials and processing, have produced materials with high Hall coefficients that generate Hall voltages orders of magnitude higher than those produced with earlier materials. For applications



where pressure must be measured, mechanical switches are the most prevalent. However, for applications where reliability is a major concern, piezoresistive sensors offer a distinct advantage.

Pressure Transducers

A pressure transducer is simply a nonlinear resistive bridge as shown left. They are fabricated as ion implanted resistors in a simple bridge configuration using common silicon processing techniques. When pressure is applied to the device, the resistors change by an amount Δ producing an output voltage described by the following equation;



$$V_o = S * P * V_s \pm V_{os}$$

where S is the sensitivity of the transducer, typically in mV/V/psi , P is the input pressure in psi , V_s is the supply voltage, and V_{os} is the zero pressure offset voltage at 25°C . The generic subcircuit shown in Figure 4 describes the simple linear behavior of a pressure transducer. The IsSPICE3 behavioral sources, B1 and B2, construct the output voltage equation. The sensitivity, SENS, and the zero pressure offset voltage, VOS, parameters are passed into the subcircuit and evaluated automatically when a simulation is run. The pressure, P1 - P2, is an input to the subcircuit as is the supply voltage, V(6) - V(5). The resistance, RIN, models the resistive bridge seen by the supply voltage and load. The sensitivity units are assumed to be V/V/psi but can be altered as long as a consistent set is maintained throughout the subcircuit and the input voltage (analog of pressure) is specified correctly.

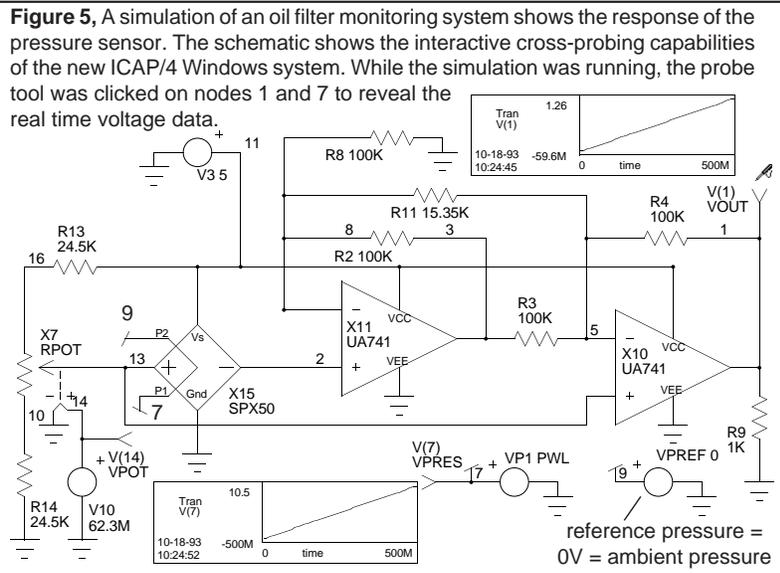
All of the generic parameters are available from a typical data sheet. For example, the SX01 pressure sensor from SenSym has the following typical characteristics;

Parameters	Sensitivity = 4 mV/V/psi	Zero Pressure VOS = 20mV
measured at 25°C	Input Resistance = 4.65k Ω	Output Resistance = 4.65k Ω

Thus, a typical call line to the generic model would be:

```
X1 1 2 3 4 DUCER {SENS=4E-3 VOS=-20E-3 RIN=4.65K ROUT=4.65K}
```

There are three kinds of sensors that can be modeled with the new subcircuit. Differential devices measure one pressure with respect to another pressure. P1 is typically the high pressure input with P2 the low. For the differential transducer model shown in Figure 4, P1 should always remain above P2. Gage devices measure pressure with respect to an ambient pressure. In this case, pressure is measured at one input while the other is set at a desired reference pressure. Absolute devices



measure pressure with respect to an internally isolated vacuum chamber. For these devices, the P2 pressure terminal is removed and replaced internally with a DC voltage source of 0.0005V representing 0.0005 psia. This will have to be adjusted if other units are used.

A typical application is shown in Figure 5. The SenSym SPX50 is used to produce a simple low cost oil filter monitor. The sensor monitors the pressure drop across a typical oil filter to determine its replacement status. A clean filter would typically have a 4 psid pressure drop, however, as dirt accumulates, the pressure drop across the filter increases. At a drop of 7psid, the filter should be changed. Slight modifications to the SP05 model were made to allow for topological differences in the SPX50 used above. During the simulation the pressure was swept from 0-10psi with resulting output at node 1 of .36mV to 1.2V. The output of the sensor went from 0 to 80mV over the 10psi range. Note: the sensor offset pot, RPOT, and the full-scale gain adjust resistor, R11, were both interactively tweaked to find their proper values without ever leaving the IsPICE4 simulator.

Model Limitations

Generally, a sensor is operated in a specific pressure range. Since this range is not normally exceeded, some of the nonlinearities have not been modeled. The pressure transducer models the linear behavior of a given device. The model gives the correct response time through the use of delay elements. (R7,C1,E1 and R8,C2,E2 as seen in the subcircuit). Nonlineari-

ties associated with the pressure to voltage conversion, hysteresis, repeatability, and stability are not modeled. Temperature dependencies (offset/span) are also not modeled but could be easily added. The applied pressure is not limited, hence, if the applied pressure exceeds the rated Maximum Over Pressure the transducer will operate as if it were a legal condition.

Other techniques for the development of new pressure transducers, such as the X-Ducer™ made by Motorola, use a single piezoresistive element to sense the shear stresses. This single-element transverse voltage strain gauge can be viewed as the mechanical analog of a Hall effect device discussed next.

Hall Effect Sensors

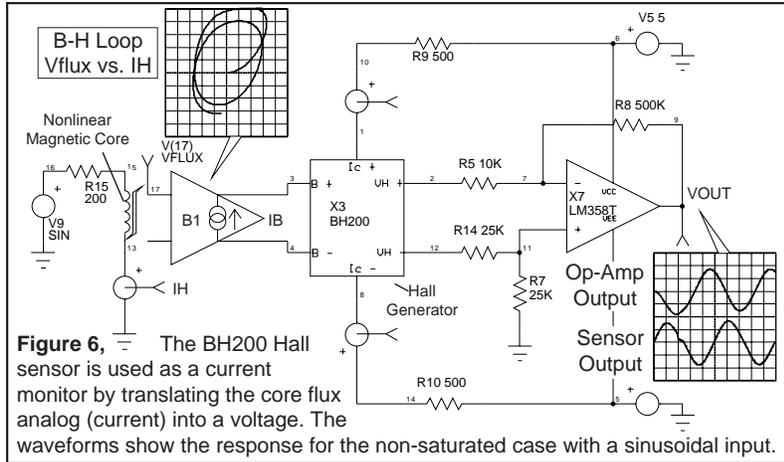
Edwin H. Hall discovered in 1879 that a magnetic field would skew equipotential lines in a current carrying conductor. This effect is observed as a voltage perpendicular to the direction of the current in the conductor, hence the name “Hall effect”. A Hall generator is a 4 terminal device that produces a voltage, V_{Hall} , proportional to the product of the input current, I_c , the magnetic flux density, B , and the sine of the angle between B and the Hall generator ($V_{Hall} = K * I_c * B$). By holding I_c constant, V_{Hall} becomes a direct function of B . If both I_c and B are variable, V_{Hall} is proportional to the product of the two functions. With B and I_c held constant, V_{Hall} becomes a function of the angle between B and the plane of the Hall sensor’s active area.

SPICE modeling is an extremely powerful tool for predicting the performance of analog circuits that utilize Hall sensors, as well as for testing the operation of different sensors. The Is_{SPICE} subcircuit used here was provided by F. W. BELL, makers of Hall generators. Table 2, at the end of the article, shows an example subcircuit. The subcircuit models the following effects:

Input Resistance	Output Resistance	Magneto Resistance
Magnetic Sensitivity	Inductive Pickup	Linearity
Offset Voltage	Temp. Coefficient of the Offset and the Output	

Max., min., and typical models are available. The parameters that are altered in these models are the input resistance, output resistance, and sensitivity. All temperature coefficients and the misalignment voltage are set to their maximum values. All other parameters are set to their nominal values.

To test the Hall model, a current monitoring circuit was set-up. In the real circuit, the Hall sensor resides in a small gap in a flux-collector toroid in order to measure the current. For the simulation, the Intusoft core model conveniently provides a flux test



point out of the nonlinear core model. The B1 element converts the flux into a current ($IB = V(17) \cdot B_m / V_n$), where B_m and V_n are core model parameters. The current into the sensor ($B+ - B-$) represents the magnetic field level with $1A = 1KGauss$. The circuit was simulated with the core saturated and unsaturated. The results are shown in Figures 6 and 7. Over 30 SPICE models of pressure and Hall effect sensors are included on the newsletter subscriber's floppy disk.

- [1] SensSym Solid-State Sensor Handbook 1992, SensSym Inc., 1244 Reamwood Ave. Sunnyvale, CA 94089, 408-744-1500
- [2] Motorola Master Selection Guide, Sensors section, 1992
- [3] F.W. Bell Hall Generators Catalog, 6120 Hanging Moss Rd., Orlando FL 32807, 407-678-6900
- [4] EDN Design Ideas Special Issue, Vol. 1 July 7, 1988, pg. 39

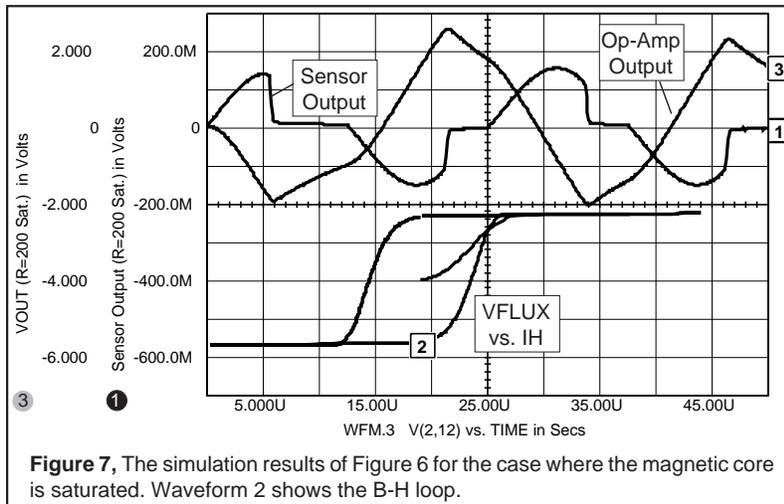


Table 2, ISpICE3 subcircuit model for the BH200 Hall Effect sensor from F.W. Bell, Inc. The syntax uses SPICE3 conventions to take advantage of the resistor model statement. For SPICE2 programs, move the resistor temperature coefficients on to the resistor call line, "RGUO 41 100 1 TC = 1.E-2".

```
.SUBCKT BH200 2 100 17 4 31 101
* Node 2: +Ic Hall Control(Bias) Current Input
* Node 100: -Ic Hall Control(Bias) Current Input
* Node 17: +Vh Hall Output
* Node 4: -Vh Hall Output
* Node 31: +B Magnetic Field Input
* Node 101: -B Magnetic Field Input
FUO 100 41 VIST 1.E-3
RGUO 41 100 1 RMOD10M
.MODEL RMOD10M R(TC1 = 1.E-2)
EUO 9 5 41 100 1
E1 3 11 50 100 0.05
E2 13 4 50 100 0.05
VIST 2 7 0
VOM 15 3 0
FNON 100 190 POLY(1) VIB 0 6.38E-5 0 7.465E-8
RG 190 100 1 RMOD800U
.MODEL RMOD800U R(TC1 = -8.E-4)
EMAGC 37 100 31 1 1.E3
CMAG 37 38 1.E-11
RMAG 38 100 4.695
EXX1 301 100 POLY(2) 71 100 190 100 0 0 0 10.0
EXX2 302 301 38 100 1
RXX1 302 100 1
ENON 17 15 302 100 1
HR11 73 100 POLY(1) VIB -0.33862 0.108319
MI1 73 63 70 70 TYPE1
HG11 63 70 VIB 1
.MODEL TYPE1 NMOS LEVEL=1 VTO=3.126
HR12 74 100 POLY(1) VIB -0.34121 -0.10838
MI2 74 64 70 70 TYPE2
HG12 64 70 VIB -1
.MODEL TYPE2 NMOS LEVEL=1 VTO=3.148
RMAG1 70 100 1 RMOD1_5M
.MODEL RMOD1_5M R(TC1= 1.5E-3)
HIST 71 100 VIST 1
RIST 71 100 1
EV1 7 6 POLY(2) 71 100 70 100 0 0 0 -0.5
EV2 8 100 POLY(2) 71 100 70 100 0 0 0 -0.5
HRO1 93 100 POLY(1) VIB -0.24846 0.109796
MO1 93 95 80 80 TYPE3
HGO1 95 80 VIB 1
.MODEL TYPE3 NMOS LEVEL=1 VTO=2.2629
HRO2 94 100 POLY(1) VIB -0.24709 -0.10988
MO2 94 96 80 80 TYPE4
HGO2 96 80 VIB -1
.MODEL TYPE4 NMOS LEVEL=1 VTO=2.2487
RMAGO 80 100 1 RMOD1_5M
HVOM 81 100 VOM 1
ROM 81 100 1
EV3 11 10 POLY(2) 81 100 80 100 0 0 0 -0.5
EV4 12 13 POLY(2) 81 100 80 100 0 0 0 -0.5
R1 6 5 1 RMOD1_5M
R2 5 8 1 RMOD1_5M
R3 10 9 0.85 RMOD1_5M
R4 5 12 0.85 RMOD1_5M
RB 31 1 1
VIB 1 101 0
RDUM 101 100 1.E8
GFST 100 50 POLY(2) 31 1 71 100 0 0 0 1
RST 50 100 1 RMOD800U
.ENDS
```

Intusoft NEWS: CompuServe Information Service

Intusoft maintains a bulletin board service under the CADD/CAM/CAE Vendor forum on CompuServe. A variety of technical articles, application notes, demos and free SPICE models are posted there. To connect with Intusoft you can navigate through CompuServe's computing support menus to reach the CADD/CAM/CAE Vendor forum or type "Go CADDVEN" at any ! point prompt. Then select the "All CADD/CAM/CAE" section.

New items posted this month include: **Multidec** (a utility to create SPICE models of multiple coupled lossy transmission lines), SPICE models from Philips Semiconductors for ALS, FAST, and ABT logic families, plus a paper on SPICE models for fuses.

The Intusoft Modeling Corner

In this issue of The Intusoft Modeling Corner we bring you the latest models from several Asia-Pacific vendors.

New Pacific Rim Parts Join Library

Thanks to the Intusoft representatives in Japan, Taiwan, and Singapore we were able to obtain data books for a variety of Asia-Pacific discrete devices (some in English no less!). Taking advantage of the windfall, Intusoft tech support was able to create new models for diodes, JFETs, BJTs, and power BJTs. Therefore, enclosed on the floppy disk are over **250 models** from the following vendors: Toshiba, Mitsubishi, Hitachi, Fujitsu, Matsushita, Fujidenki, Yunizon, Nippon, and Sanyo.

The newsletter floppy disk also contains 30 new models for op-amps, MOS-controlled thyristors, power Mosfets, and a power Mosfet driver IC (HIP2500).

```
.MODEL Q2SC2705 NPN (IS=5.07F NF=1 BF=162 VAF=220 IKF=40M ISE=3.6P NE=2 BR=4 NR=1
+ VAR=20 IKR=60M RE=.63 RB=2.52 RC=.252 XTB=1.5 CJE=13.2P VJE=1.1 MJE=.5 CJC=4.26P
+ VJC=.3 MJC=.3 TF=795P TR=552N) ;Toshiba 2SC2705 150V 50mA 200MHz NPN BJT
.SUBCKT Q2SC2366 1 2 3 ; Fujitsu 400V 40A NPN Power BJT
Q1 1 2 3 QPWR .67
Q2 1 4 3 QPWR .33
RBS 2 4 5
.MODEL QPWR NPN (IS=1.47P NF=1 BF=58.5 VAF=360 IKF=13.3 ISE=3.11N NE=2
+ BR=4 NR=1 VAR=20 IKR=20 RE=11.5M RB=46.3M RC=4.63M XTB=1.5
+ CJE=2.03N VJE=6 MJE=.3 CJC=1.31N VJC=.22 MJC=.2 TF=10.6N TR=6.8U)
.ENDS
.MODEL Q2SB767 PNP (IS=50.7F NF=1 BF=403 VAF=161 IKF=.3 ISE=12.5P NE=2 BR=4 NR=1
+ VAR=16 IKR=.45 RE=78.3M RB=.313 RC=31.3M XTB=1.5 CJE=183P VJE=1.1 MJE=.5 CJC=59.1P
+ VJC=.3 MJC=.3 TF=1.32N TR=921N) ;Matsushita 80V .5A 120M PNP BJT
```

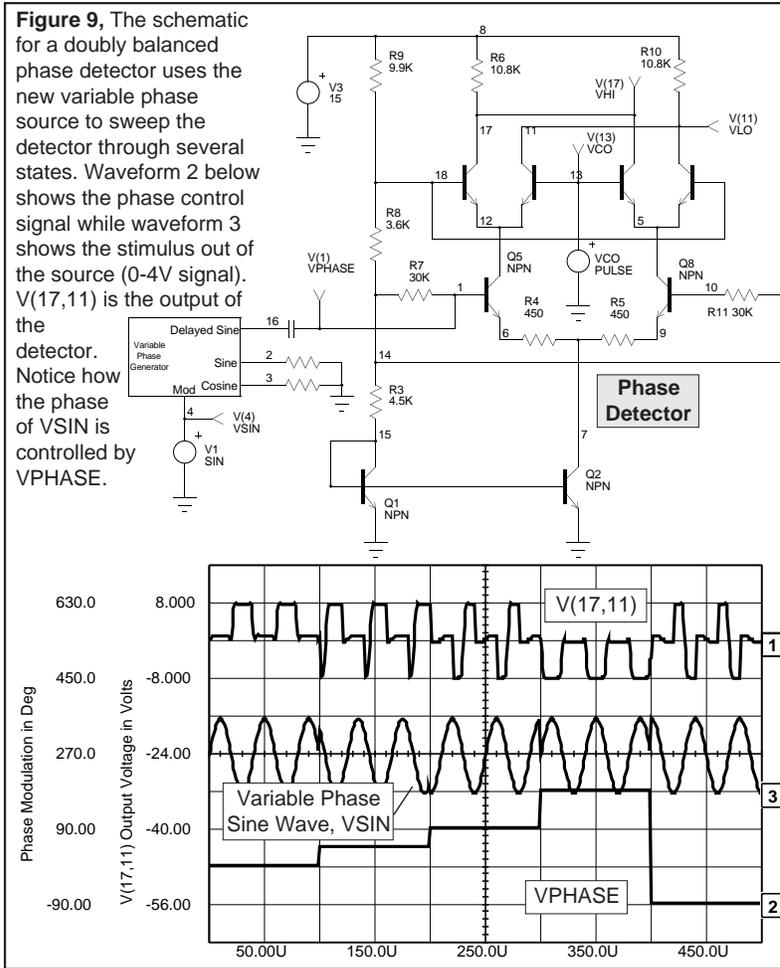
Sample Pacific Rim Models

Stimulating Circuits: Variable Phase

The SIN wave generator in Berkeley SPICE does not have a phase delay, only an initial time delay. A simple way to create

```
.SUBCKT PHASE2 2 4 5 10 {VGEN=1 FREQ=1K VOFFSET=0}
* NODE 2 is the delayed Sine wave output
* NODE 4 is a reference Sine wave
* NODE 5 is a reference Cosine wave
* NODE 10 is phase angle variation in degrees
*VGEN is the amplitude in volts, FREQ is frequency in Hz
C1 1 0 {1/(6.28319K * FREQ)} ; Use IC={VGEN} for UIC
R1 1 0 1E6
I1 0 1 PULSE {VGEN * 1U} 0
* I1 is used for the initialization of C1
E1 5 0 0 1 1
G1 0 1 0 4 1M
BE2 2 0 V={VOFFSET} + (V(5) * SIN(V(10) * 3.1415926/180) +
+ V(4) * COS(V(10) * 3.1415926/180))
R5 5 0 1MEG
R4 4 0 1MEG
V1 4 0 SIN 0 {VGEN} {FREQ}
.ENDS
```

Figure 8, The subcircuit for a phase modulated voltage generator. The subcircuit uses a variation of the 3 phase generator model included with Intusoft's standard model libraries. First a reference sine wave is integrated to get a cosine wave. Then, the two are combined using the in-line equation in BE2. V(10) is the voltage equal to the desired phase delay.



a variable phase stimulus is to combine cosine and sine waveforms. The subcircuit in Figure 8 does this very efficiently using the IsSPICE3 in-line equation feature. To test the operation, the source was connected to a phase detector as shown in Figure 9. The stimulus waveform was modulated through several phase changes and the output of the detector observed.

Philips Produces Models for 3V BiCMOS

Philips has released SPICE models for its 3-Volt line of BiCMOS and MOS products including the HLL, LVC, and LVT families. The new models join previously released models for the ALS, FAST, and ABT logic families.