Intusoft Newsletter

Personal Computer Circuit Design Tools

April 1994 Issue

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

NEW Low Cost Version Makes SPICE Affordable for ALL

Many current and potential users of SPICE based tools have asked for a slimmed down version of the ICAP/4 system. Whether it be for home use, school work, or less intensive simulation tasks, the new low cost ICAP/4Lite package is the simulation tool you have been waiting for. ICAP/4Lite is a reduced feature version of our ICAP/4Windows system and its advantages are clear. ICAP/4Lite (list price: $595) comes complete with an integrated schematic entry front-end, model libraries (> 500 parts), the popular

continued on Page 2

In This Issue

Interactive SPICE Eases Sweeping

Parameter sweeping is one of the main strengths of circuit simulation. It allows the designer the freedom to insert a range of component values into a circuit and see what happens. These what-if trials can be invaluable when trying to understand a design.

continued on Page 3

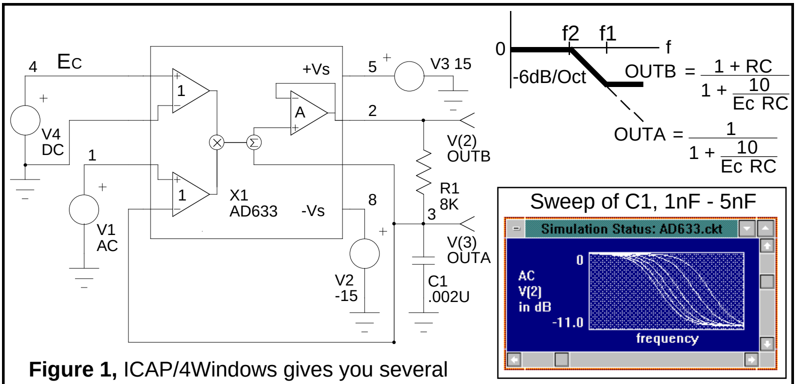


Figure 1, ICAP/4Windows gives you several ways to sweep parameters depending on your needs. As an example, the Analog Devices AD633 analog multiplier is setup as a voltage controlled low-pass filter. Here, and on page 3, R1, C1, and V4 are swept to examine the circuit performance.

ICAP/4Lite: *Simulates Great, Less Costly*

Continued from page 1

"SPICE Cookbook" reference book, and a Windows version of IsSpice3 (32-bit, extended RAM, SPICE 3F.2 based) that allows **UNLIMITED SIZE** circuits. ICAP/4Lite is superior to other low cost products, such as the small analog Pspice® version from Microsim ($1495), which does not include schematic entry and is limited to circuits with less than 100 transistors (16-bit, DOS 640K memory, SPICE 2G.6). In addition, when compared to other low cost software vendors Intusoft provides technical support you can count on and an upgrade path to higher performance tools.

The main differences between the ICAP/4Windows system and the ICAP/4Lite system are summarized in the following table:

	ICAP/4Windows	**ICAP/4Lite**
Integrated Schematic Entry (SpiceNet)	Yes	Limited to 1 page, no symbol editing, symbols provided
Works with third party schematics	Yes	Yes
Graphical Data Processor (IntuScope)	Yes	No (Data display and analysis is performed in IsSpice)
SPICE Simulation	Yes	Yes
Interactive	Yes	No
Simulation Scripts	Yes	Yes
Real Time Waveform Display	Yes	Yes
All SPICE 3 Analyses	Yes	AC, DC, and TRANSIENT only
All SPICE 3 Elements	Yes	No Mesfet, MOS Levels 4-6, Lossy T-lines, or SPICE 3 switches
Berkeley SPICE 2/3 compatible	Yes	Yes
Circuit Size	Unlimited	Unlimited
Model Libraries	> 5000	> 500
Behavioral Models	Yes	Yes
Fully Supported	Yes	Yes
Windows/Windows NT	Yes	Yes
List Price	**$2595**	**$595**

Price and Availability

With the introduction of ICAP/4Lite Intusoft continues its tradition of bringing high performance CAE tools to every engineer at an AFFORDABLE price. If price has been holding you back, then ICAP/4Lite is for you. ICAP/4Lite will be available June 1, 1994 for **$595.** Note: Look for ICAP/4Lite at your local retail software outlet.

Exact Feature Comparison

For those interested in finding out more about the low cost ICAP/4Lite system and how it compares with other Intusoft ICAP systems, as well as with CAE tools from other vendors, please fax a request to (310) 833-9658 including your fax number. A summary comparison will be faxed back to you. In addition, a summary comparison will be posted on the Intusoft forum (Go Caddven) on CompuServe.

Parameter Sweeping Made Easy

Continued from page 1

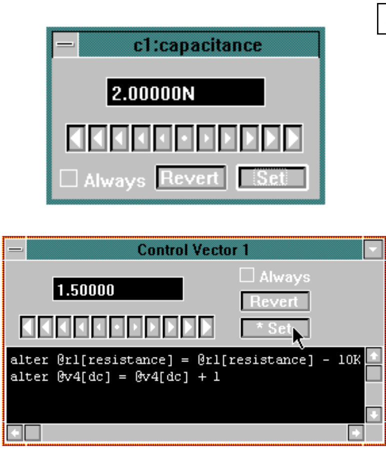
The new ICAP/4Windows system gives you three main ways to sweep parameters:

- Interactive sweeping in the IsSpice4 environment
- Simulation Scripts
- Batch Style IsSpice4/IntuScope combination

Thought it is possible to combine these methods we will treat them individually in order to better distinguish their features. The table below gives a brief summary of each method. The circuit in figure 1 was used to explore the different parameter sweeping methods. With interactive parameter sweeping (figures 1 and 2) you simply choose the device or model parameters you want to sweep from a list. Parameters can be arbitrarily varied one at a time or in groups. When the value is

Sweep Method	***Characteristics***
Interactive Sweeping (Inside IsSpice4)	Fast, interactive, easy to use and setup, arbitrary sweep values and increments, only display multiple waveforms (curve families) or simulation data values, one analysis at a time
Simulation Scripts (Inside IsSpice4 or via netlist)	Procedural language, arbitrary number and type of analyses, arbitrary sweep values and increments, automated analysis loops with *Simulation Breakpoints*, build curve families in IntuScope for post processing, save and display simulation data values and simple measurements (mean, RMS)
Batch Style (IsSpice4 in combination with IntuScope)	Complex measurements on any simulation data, automated summary of measurements from several simulations, requires separate IntuScope keystroke macro and netlist alterations

Simulation data refers to node voltages, device currents, device power dissipation, and computed device parameters such as BJT Vbe, and FET gm.



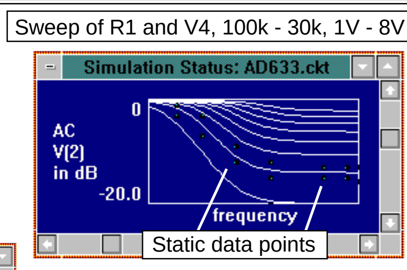


Figure 2, With IsSpice4 you can interactively sweep parameters singly or in groups. The Points feature can be used to set up static data points in a graph window. Once placed you can simply "dial in" the circuit values to match the performance you want.

```
stop when frequency > 50K when db(v(2)) < -15
repeat
ac dec 20 100 100k
alter @v4[dc]=@v4[dc] - .3
alias vdb2 db(v(2))
sendplot vdb2
end
```

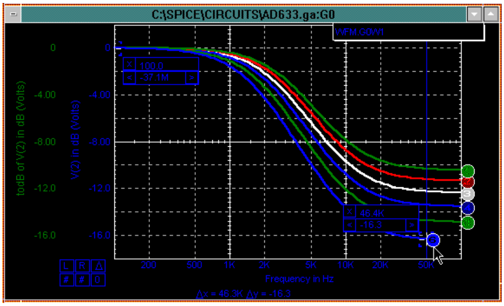


Figure 3, The Simulation Scripts feature of IsSpice4 allows complex sweeps to be performed like a test procedure. The script above sets a breakpoint when the frequency is > 50K and OUTB is < 15dB. Then a series of simulations are performed, with V4 varied after each analysis, until the condition is met. The graph shows the curve family results as V4 is swept from 3V to 1.7V, when the condition is met.

changed, the simulation is repeated and the results displayed. This is the fastest and easiest way to see how a particular parameter affects your circuit. Figure 3 shows a more powerful sweeping method, Simulation Scripts. Scripts allow you more versatility in the variety of analyses and tests you can perform. Curve families can also be sent to IntuScope for post simulation analysis. There are over 60 different script commands. The last method uses the IntuScope data processing program to analyze and record data from the output of each simulation. In this case, the 3dB frequency and the attenuation at 10kHz were measured for each sweep step of V4. When the entire sweep is complete IntuScope conveniently presents the summarized results.

Many other sweeping scenarios are possible, especially when the above methods are combined. No matter what your needs may be, ICAP/4 systems gives you the power and versatility to study your designs in the most cost effective manner.

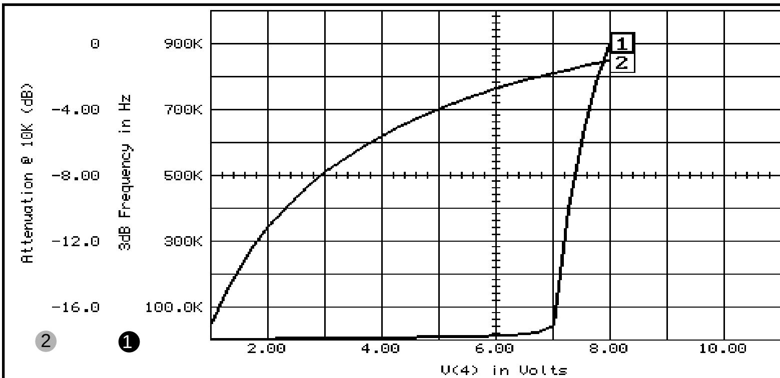


Figure 4, Batch style parameter sweeping allows IntuScope to take advanced measurements after each simulation is run.

Circuit Initialization

Problem: You have entered a simple inverting op-amp circuit such as the one shown at the bottom of Figure 5. But by accident you have reversed your op-amp inputs. It may be because the symbol was wrong or the subcircuit node list was typed in incorrectly. In any case, you find the circuit works the same in BOTH inverting and non-inverting configurations! What's going on?

The problem you have encountered is one of **proper circuit initialization**. For example, if we take the following initial simulation conditions, referring to figure 5, (V1=.1V, V2=0V initially then a slow ramp to 1V, Power Supplies ON, .TRAN .1s 5s) IsSpice gives the correct answer. Let's see why.

Using a slightly simplified case of an inverting amplifier we can solve the KCL equations for Vo as shown in Figure 6. IsSpice is designed to solve these equations, just as you would by hand. Clearly, another solution exists in which the output goes to positive infinity for linear models or the positive op-amp rail for non-linear models. To obtain this solution, the simulation can not be started under the assumption that the power and signals are already present. When we make this assumption, IsSpice is free to create a negative initial condition at the output. This could never occur in reality because the summing junction input is always positive. To get IsSpice to represent real life, it is necessary to apply power and signals as a function of time (as in real life), beginning with the device turned off so that all internal nodes are initialized to zero. Moreover, the circuit must be modeled like a real op-amp, in that it has a sense of temporal ordering. That is, the input must occur before the output. This requires at least one pole in the transfer function, and that the

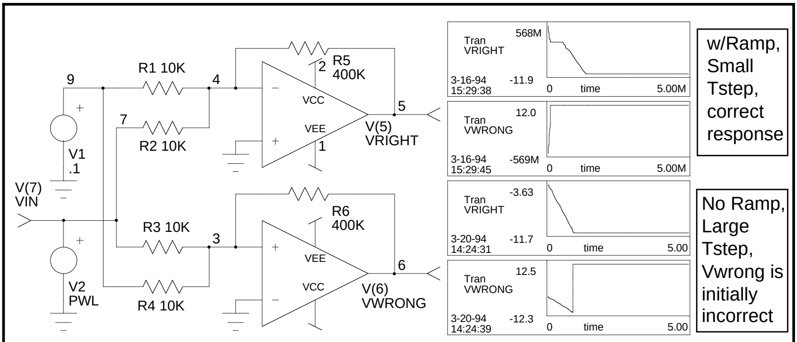
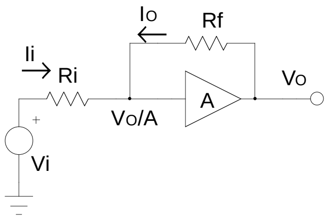


Figure 5, By properly initializing the circuit (ramping the power supplies, adjusting the time step (TSTEP)), the proper op-amp response is obtained.



$$i_o + i_i = 0$$

$$\frac{v_o - \frac{v_o}{A}}{Rf} + \frac{v_i - \frac{v_o}{A}}{Ri} = 0$$

$$v_o\, Ri\left(1 - \frac{1}{A}\right) + v_i\, Rf - \frac{v_o}{A}\, Rf = 0 \longrightarrow v_o = \frac{-v_i\, Rf}{Ri\left(1 - \frac{1}{A}\right) - \frac{Rf}{A}}$$

let $|A| \rightarrow \infty$ then $v_o = \dfrac{v_i\, Rf}{Ri}$

Figure 6, Using a simplified inverter we can solve the KCL equations.

time steps in the simulation are small enough to "expose" the pole. The pole will lower the op-amp gain for small time steps so that an output solution cannot be found for negative outputs. To shorten the IsSpice time steps we can change the .TRAN from "`.1s 5s`" to "`.1ms 5ms`". To ramp the power supplies, change "`VCC 10 0 12V`" to "`VCC 10 0 PULSE 0 12 0 .1m`". It is best to give the power supply a realistic ramp time, in this case .1ms. The simulation results are shown in Figure 5.

This may seem pretty complex for a novice SPICE user; however, it's a standard technique when the initial operating point is suspect. We strongly recommend that SPICE users have a general idea of the circuit behavior they expect. Simple errors in construction of a SPICE netlist could give wildly erroneous data. The user must be prepared to debug a SPICE simulation, just as they must debug a breadboard after it has first been constructed. At least when SPICE netlists are constructed improperly you don't get parts flying off a smoldering breadboard. Instead of ducking, you may feel like throwing the computer out of a window. But debugging is a fact of life in circuit design and the techniques for debugging a simulation must be learned, just like the techniques for debugging a breadboard. It turns out the steps for running a simulation aren't that different from getting a breadboard to run; that is, take a series of small careful steps, checking each section of circuitry before connecting the next. The learning curve is pretty easy, and the increase in design productivity can be very large.

Intusoft on CompuServe Information Service

There is also an application note on solving SPICE convergence problems on the Intusoft CompuServe CAD/CAE/CAM forum (GO CADDVEN for CompuServe users). The Intusoft CompuServe address is **71564,3147**. For Internet e-mail use: "71564.3147@compuserve.com".

Modeling Vacuum Tubes

Part II

In the February 1994 issue of the *Intusoft Newsletter*, we introduced a very accurate triode vacuum tube model that makes use of the IsSpice4 behavioral modeling capability (in-line equations). In part II of this article we will extend the development to include models for pentodes or tetrodes.

A Realistic Pentode Model

Pentodes users have had more problems then triode enthusiasts when trying to simulate their favorite component, since no realistic models have existed up to now. In references [1,2] equations are given for the plate-to-cathode current flow as:

$$I_K = K_T\left(\frac{E_B}{\mu} + E_C\right)^{\frac{3}{2}} \text{; triodes} \quad I_K = K_T\left(\frac{E_{C2}}{\mu} + E_C\right)^{\frac{3}{2}} \frac{2}{\pi} \arctan\left(\frac{E_B}{10}\right) \text{; pentodes}$$

While the triode equation can be solved with the SPICE 2 style polynomial [1], both equations are more easily simulated using the in-line equation feature of IsSpice4. For example:

B1 1 2 I = KT ∗ (EC2/μ + EC)^1.5 ∗ 2/pi ∗ ATAN(EB/10)

where KT and μ are constants and EC2, EC, and EB are node names. Unfortunately, these equation are only approximations lacking sufficient accuracy with respect to positive grid voltage, grid current, and high plate currents for many applications.

The model for the more realistic pentode in Table 1 was derived from the triode model in part I, but takes into account the specific properties attributable to several grids and their interactions. The model takes into account the law for the G2 (second grid) current, the effects of the G2 voltage on the anode current and the G1 current (in the case of positive G1 voltage). Even the effects of the virtual cathode (quick reduction of the cathode current when Va goes below 40 Volts) have been modeled. The pentode subcircuit is made up of interelectrode capacitances, a heater subcircuit which can be excluded, and the main PENT1 subcircuit. The PENT1 portion is parameterized allowing virtually any tube to be modeled. The parameters are defined in Table 2.

Figures 7 and 8 show the results for a sinusoidal input (8V 2kHz) and the characteristic output curves (IK versus VA, for several VG1 voltages), which are obtained with the new pentode model for a typical set of pentode parameters.

Table 1, Forward and reverse conditions are treated in the pentode model, as well as saturation. Model parameters for a typical tube are shown below.

```
.SUBCKT ELXXXX    1     2        3         4          5    6
*                     Anode Grid2 Grid1 Cathode  F    F'  COPYRIGHT EXCEM, 1993
X1 1 2 3 4 10 PENT1 {SFS=0.7 VBIG=-0.9 VBIA=-1.3 MUG2=17 MUA=15000 RMU=0.5
+ VMU=-20 SFMU=1.6 K=5.4E-3 RK=0.08 VK=-20 SFK=1.6 SIGMA1=0.05
+ ALPHA1=5.2 SFG1=3.5 SIGMA2=0.12 ALPHA2=0.06 SFG2=2.3 VCCR=0.58 SFVC=0.33}
X2 5 6 10 HEAT1 {INOM=0.15 VNOM=6.3 LAMBDA=1 RCOOL=3 TCTE=10 TNOM=1150 INIT=100
+ W=2.045 ISAT=0.690}  ; Heater can be replaced with 2 statements "RF 5 6 42" & "VH 10 0 99m".
C2 1 2 1.5P
C3 3 1 0.5P
C4 2 3 1.6P
C5 3 4 4P
C6 3 5 4P
.ENDS
******************
.SUBCKT PENT1    A  G2  G1  C  ISAT   ; COPYRIGHT EXCEM, 1993
B1 15 0 V = V(G1) - V(C) < -1U ?  {K} * (1 + {RK} *
+ ((V(G1) - V(C)) / {VK})^{SFK}) / (1 + ((V(G1) - V(C)) / {VK})^{SFK}) : {K}
B2 16 0 V = V(G1) - V(C) < -1U ?  (1 + {RMU} * ((V(G1) - V(C)) / {VMU})^{SFMU}) /
+ (1 + ((V(G1) - V(C)) / {VMU})^{SFMU}) : 1
E1 17 0 16 0 {MUG2}
E2 18 0 16 0 {MUA}
B4 9 0 V = V(G1) - V(C) - {VBIG} + (V(A) - V(C) - {VBIA}) / (V(18) + 1U)+(V(G2) - V(C)) / (V(17)+1U)
B6 10 0 V = V(9) > 1P ? V(15) * V(9)^1.5 / (V(ISAT) + 1P) : 0
B7 12 0 V = V(10) < {SFS} ? V(10) * (V(ISAT) + 1P) :
+ (V(ISAT) + 1P) * ({SFS} + (V(10) - {SFS}) * {1-SFS} / ({1 - 2 * SFS} + V(10)))
B8 14 0 V = V(G2) - V(C) + {MUG2 / MUA} * (V(A) - V(C)) > 0.1M ?
+ V(G2) - V(C) + {MUG2 / MUA} * (V(A) - V(C)) / {ALPHA1} : 0.2M
B9 28 0 V = V(G1) - V(C) > {VBIG + 10U} ? V(14) > 0.1M ? ((V(G1) - V(C) - {VBIG} +
+ {SIGMA1^(1/SFG1)} * V(14)) / (V(G1) - V(C) - {VBIG} + V(14)))^{SFG1} : 0
B10 8 0 V = V(G1) - V(C) < 0 ? V(28) * (({VBIG+10U} + V(C) - V(G1)) / {VBIG+10U}) : V(28)
B11 21 0 V = V(A) - V(C) > {VBIA + 0.1M} ? (V(A) - V(C) - {VBIA}) / {ALPHA2} : 0.2M
B12 32 0 V = V(G2) - V(C) > {VBIG+10U} ? V(21) > 0.1M ?
+((V(G2) - V(C) - {VBIG} + {SIGMA2^(1/SFG2)} * V(21)) / (V(G2) - V(C) - {VBIG} + V(21)))^{SFG2} : 0
B13 22 0 V = V(G2) - V(C) < 0 ? V(32) * (({VBIG+10U} + V(C) - V(G2)) / {VBIG+10U}) : V(32)
B14 23 0 V = V(22) - {SIGMA2} > 1P ? V(12) * (1-{VCCR} * (V(22) - {SIGMA2})^{SFVC}) : V(12)
B15 G1 C I = V(8) * V(23)
R15 G1 C 100MEG
B16 G2 C I = (1 - V(8)) * V(22) * V(23)
R16 G2 C 100MEG
B17 A C I = (1 - V(8)) * (1 - V(22)) * V(23)
R17 A C 100MEG
.ENDS
```

See Table 2 for notes on the B elements and parameters. See Feb. '94 issue for information on the heater model.

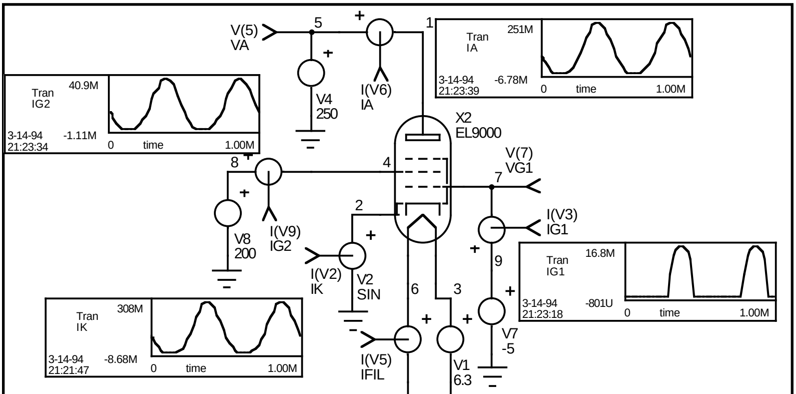


Figure 7, Cross-probed waveforms show the pentode tube response to sinusoidal input (8V, 2kHz) at the cathode.

Table 2, The generic parameters for the pentode subcircuit (PENT1) reveal the tremendous versatility of the model.

The Triode Parameters Are:

SFS	Shape factor of the saturation law.
VBIG	Contact potential of the grid G1 (above this value grid current may start to flow).
VBIA	Contact potential of the anode.
MUG2	Amplification factor for G2 at slightly negative G1 voltage.
MUA	Amplification factor for A at slightly negative G1 voltage.
RMU	Reduction factor for MU at very negative G1 voltage.
MU	Grid voltage for mid-range MU (negative).
SFMU	Shape factor for MU reduction law.
K	Perveance at slightly negative G1 voltage.
RK	Perveance reduction factor at very negative G1 voltage.
VK	Grid voltage for mid-range perveance (negative).
SFK	Shape factor for perveance reduction law.
SIGMA1	Effective cross-section of G1 relative to the anode and G2.
ALPHA1	Grid G1 current amplification factor.
SFG1	Shape factor of the grid G1 current law.
SIGMA2	Effective cross-section of G2 relative to the anode.
ALPHA2	Grid G2 current amplification factor.
SFG2	Shape factor of the grid G2 current law.
VCCR	Virtual cathode current ratio.
SFVC	Shape factor of the virtual cathode current law.

Model Notes: B7 contains an arbitrary saturation law modeled by the shape factor SFS to match the available (?) data. SFS should be between 0 and 1, and the lower SFS, the sloppier the saturation law. V(15) is the effective perveance. V(16) is the factor used to establish both effective MU coefficients. V(17) is the effective MUG2 and V(18) is the effective MUA. (B14, V(23)) When the virtual cathode is present, this factor describes the decrease in cathode current (see Terman p. 192). The model describes only the static behavior of the pentode, and neglects secondary emission. It is assumed that G2 is always very positive with respect to the cathode. The heater parameters are described in part 1 of this article (Feb. 1994 issue).

Figure 9 shows the use of a pentode in an cascode amplifier circuit similar to the one which appeared in part I of this article and used a dual triode combination.

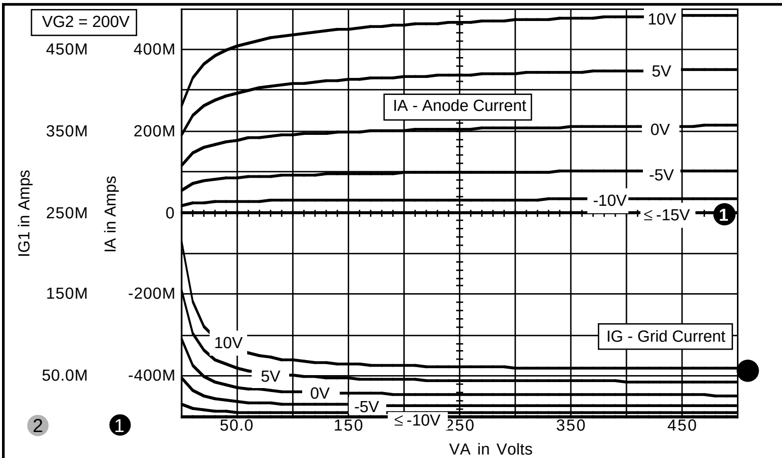


Figure 8, DC characteristics for a typical pentode. The graph clearly shows the nonlinear effects of anode and grid current versus anode and grid G1 voltage.

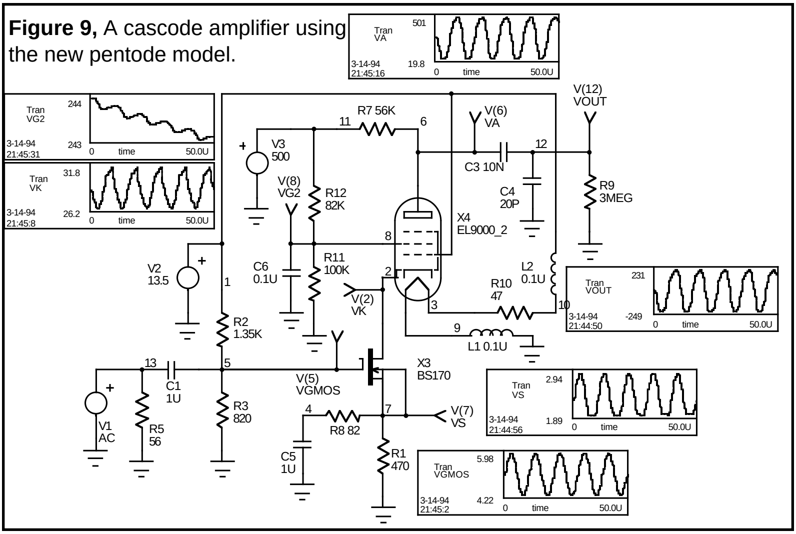


Figure 9, A cascode amplifier using the new pentode model.

The response of this amplifier to a sinusoid of 100 kHz with an amplitude of 1.6 Volts pk-pk gives an output amplitude of about 430 Volts pk-pk. As in the case of the triode version, there is a slight distortion at this level, however with different characteristics. VGMOS is the signal at the input and VA is the signal at the output of the pentode.

Waveforms for the pentode's cathode (VK) and grid VG2 are also shown. The non-linear shape of the cathode's voltage is very evident. The strong feedback that creates this particular waveshape should be given credit for the almost linear response obtained at the output. The curve of the Figure 10 gives the frequency response of the pentode amplifier.

In order to verify the models an amplifier similar to Figure 9 was constructed and simulated with good correlation. The amplifier used a 2kV beam tetrode and 1kV-1.2kV power supply to deliver a peak-to-peak output signal of about 960 Volts. This amplifier was designed for the measurement of the transfer

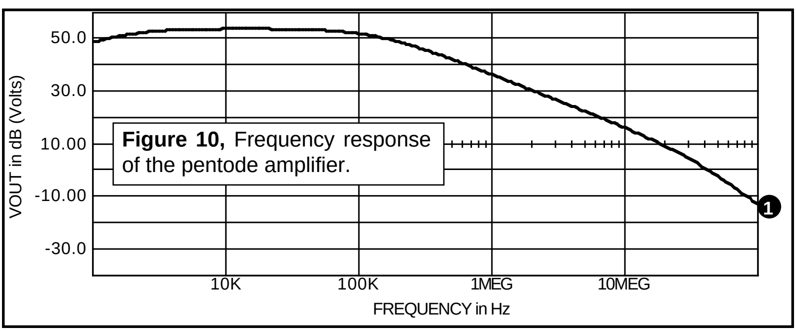


Figure 10, Frequency response of the pentode amplifier.

admittance of shielded cable in the 100 kHz to 10 MHz frequency range. With IsSpice and these new vacuum tube models audio engineers can evaluate their designs more accurately and realistically then ever before. Those tube enthusiasts who are interested in vacuum tube SPICE models for specific parts should contact Intusoft directly.

References

[1] , C. Hymowitz, L. Meares, “SPICE Applications Handbook 2nd Edition”, Intusoft, March 1994

[2] Scott Reynolds, “Vacuum Tube Models For SPICE Simulations”, Glass Audio, April 1993

[3] Frederick E. Terman, “Electronic and Radio Engineering”, McGraw-Hill, 1955

Tube models were created by: EXCEM 12, Chemin des Hauts de Clairefontaine 78580 MAULE FRANCE Tel 33 (1) 34 75 13 65, Fax 33 (1) 34 75 13 66

ICAP/4 Goes *POWER PC*

The ICAP/4Macintosh system, including schematic entry, model libraries, interactive IsSpice4 simulation, and data processing will be available for the Power PC ™ processor running under the Power Macintosh System 7 environment. The “native” version will be available mid-summer 1994. The price will be the same as the ICAP/4Macintosh system ($2595). Updates for current users of the ICAP/4Macintosh system are $99.

New Spice Applications Handbook Released

Since June of 1986, Intusoft has been providing free technical information on SPICE via the *Intusoft Newsletter*. A compilation of all of the back issues of the SPICE newsletter is now available. The handbook entitled “Spice Applications Handbook, 2nd Edition”, contains all of the past issues of the *Intusoft Newsletter*; 16 from the first volume plus 18 more. A total of 34 newsletters dating from 6/86 to 2/94. Over 60 technical articles are included covering a wide range of applications.

The handbook also contains valuable SPICE simulation tips and techniques, device modeling, and actual SPICE models for a wide variety of components. A floppy disk is included with the handbook and contains all of the models, SPICE netlists, and associated SpiceNet schematics from all of the newsletters. The floppy disk provides easy access to all of the SPICE information, eliminating the need to re-enter complex netlist information. The price of the Spice Applications Handbook, 2nd Edition, along with the floppy disk, is $49.95. Until May 31, 1994, the price is $25.95 for those who have purchased Volume 1 since 10/93.

The Intusoft Modeling Corner

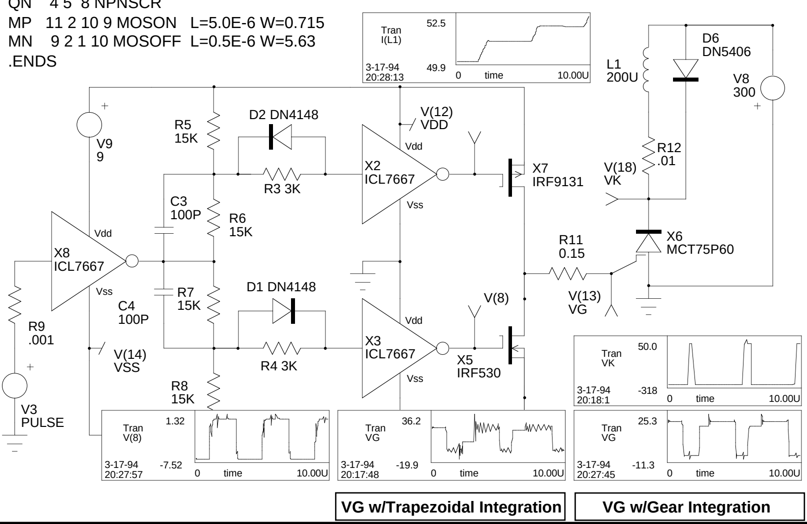
In this issue of The Intusoft Modeling Corner we bring you a short app. note on the new MCT models from Harris. In this month's newsletter floppy we have included new IC models from Analog Devices and National Semiconductor. The total includes over 41 devices for AD op-amps, multipliers, and buffers. From National there are over 40 models for CMOS, BJT, JFET, current feedback, and transconductance op-amps.

IsSpice4 Enhances Harris MCT Simulation

MCT's are an exciting new power semiconductor that combines thyristor current and voltage capability with MOS gated turn-on and turn-off. To provide support for the new device, Harris has created a SPICE model shown in the IsSpice4 simulation below. Two simulation hurdles discussed in the Harris data book are solved by features in the IsSpice4 simulator. First, Harris notes that SPICE simulations are subject to spurious numerical oscillations. This is often due to the default trapezoidal integration method. This problem can be overcome here, and for many switching type

```
.SUBCKT MCT75P60 1 2 3 ; Terminal Connections:  Anode Gate Cathode
.MODEL MOSOFF NMOS(VTO=2.0 NSUB=1E18 UO=500 TOX=7E-8)
.MODEL MOSON  PMOS(VTO=-2.0 NSUB=8E13 UO=200 TOX=7E-8)
.MODEL PNPSCR PNP(BF=9.1 VAF=2000 IKF=600 IS=1.14E-10 TF=42.9E-9 CJE=9.47E-9
+ VJE=1.0 CJC=362E-12 MJC=0.5 VJC=0.96 NF=1.18 MJE=0.5)
.MODEL NPNSCR NPN(BF=2.75 VAF=2000 IKF=600 IS=1.14E-10 TF=207E-9 CJE=1.25E-9
+ VJE=0.66 CJC=362E-12 MJC=0.5 VJC=0.64 NF=1.18 MJE=0.5)
RVPP  1 12 30E-6
RLP  12 10 96.1E-6
RVN   4  9 747E-6
RVNP  8  7 2.5E-3
RVP   5 11 31.7E-3
VDROP 7 3  0.18
QP    5 4 12 PNPSCR
QN    4 5  8 NPNSCR
MP   11 2 10 9 MOSON   L=5.0E-6 W=0.715
MN    9 2 1 10 MOSOFF  L=0.5E-6 W=5.63
.ENDS
```

Figure 11, Simulation of the gate drive circuitry for an MCT using a power MOSFET driver (ICL7667) and the Harris MCTV75P60 (75A, 600V) P-Type MOS controlled thyristor. IsSpice4 cross-probed waveforms reveal the advantages of Gear Integration.



simulations, by using GEAR integration during the transient analysis. Gear integration, with Reltol set to .01, avoids these numerical oscillations while maintaining comparable accuracy. With Gear integration the simulation in Figure 12 ran in under 20 seconds on a Pentium. With trapezoidal, the runtime was between 2-10 times longer depending on the conditions, sometimes ending in convergence failure. Both Gear and Reltol are standard IsSpice options. While the Harris solution of adding snubbers is a good idea, reducing TMAX unfortunately slows the entire simulation down.

V(1,3) is the Anode to Cathode voltage.

The second problem is that the model does not check to see if the SOA has been exceeded. This check can be easily added with the ***Simulation Breakpoints*** feature. Since all the current in the MCT passes through the resistor RVNP we can issue the statements "**Stop when @RVNP[I] > 75A" and "Stop when V(1,3) > 600V**". If these ratings are ever exceeded, the simulation will automatically pause and issue a warning. At that time the designer can decide to abort the simulation or go on and watch the circuit behavior.

While the MCT device and the SPICE model are further detailed in the 1994 Harris MCT/IGBT data book, the exact netlist is not given. Hence, it is shown here for completeness and included on the newsletter subscriber's disk. Harris notes that in the future the MCT model may be updated to include a superior MOS capacitor model. It is suggested that interested parties review the behavioral MIS capacitor model described in the Nov. 1992 Intusoft Newsletter.

Stimulating Circuits: Square Wave VCO

The Intusoft library includes a voltage controlled oscillator made from a series of integrators (G, R, and C elements). The subcircuit is parameterized allowing the VCO output to vary in amplitude and frequency per volt of input control. Shown below is the VCO subcircuit with a small addition. At the output (V(2)), the simple but powerful B element has been added. The behavioral element uses the If-Then-Else syntax to convert the 0-1V VCO signal to an arbitrary binary output level. A simulation sweeping the input voltage from 1 to 2V is shown in Figure 12.

```
.SUBCKT VCOSQ  3 30 {FREQ=1MEG VLO=0 VHI=5}
RE1 2 0 1K
E1 2 0 22 0 1
G1 22 0 POLY(2) 1 0 3 0 0 0 0 0 1
R1 22 0 1E9
C1 22 0 {.15915/FREQ} IC=0
G2 1 0 POLY(2) 22 0 3 0 0 0 0 0 -1
R2 1 0 1E9
C2 1 0 {.15915/FREQ} IC=1
I1 0 1 PULSE 1E-9 0
R3 0 3 1MEG
B1 30 0 V=V(2) > .5 ? {VHI} : {VLO}
.ENDS
```

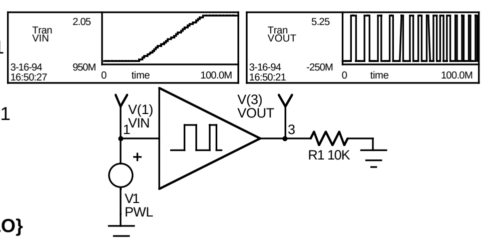


Figure 12, By combining the standard Intusoft VCO generator with an If-Then-Else behavioral model, you can create a variable frequency square wave source.