Intusoft Newsletter

Personal Computer Circuit Design Tools

#43 August 1995 Issue

intusoft

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

Windows 95 Guarantee

Many of you may be wondering what impact Windows 95 will have on your SPICE software. The answer is NONE. And Intusoft is willing to back up its words with a guarantee. If you own the latest ICAP/4 Windows version 7.51 or later Intusoft will guarantee that it will work with Windows 95. Actually the same ICAP/4Windows software will run on either Windows 3.1x, Windows95, or WindowsNT. So feel free to upgrade your Windows at any time. The NT platforms we support are x86, Digital Alpha, and MIPS.

In This Issue

New Keyless Network Easy To Justify

The new keyless network version of ICAP/4Windows offers some major benefits over standalone versions. Because its network friendly ICAP/4Windows can "float" on your network. While the software is stored on the server, the

Continued on pg. 10

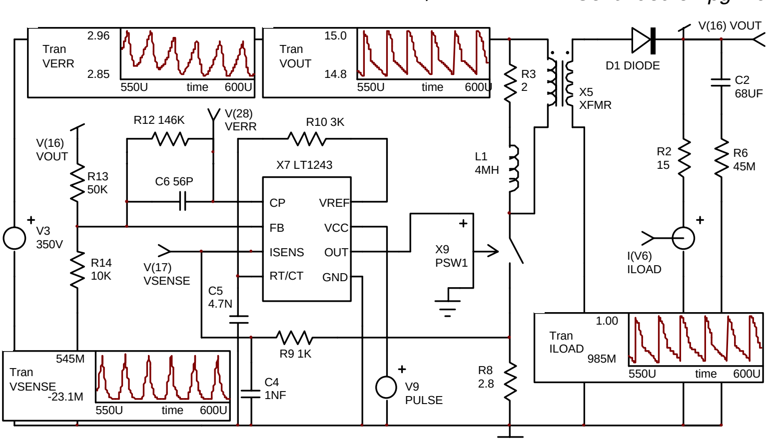


Figure 1, Intusoft's New Power Supply Designer's Library allows you to simulate forward converters like this one using the LT1243. See page 2 for more details.

Simulating SMPS Designs

In the June 1995 newsletter we introduced a new SPICE model library for power supply designers. The library contains a comprehensive set of nonlinear models for popular Pulse Width Modulation (PWM) ICs. Here we continue our review with a few examples of what the models allow you to accomplish.

Applications today are much more demanding, requiring increases in switching frequency, higher efficiency and lower standby current. State space models simply do not reveal many important factors which influence these performance characteristics. As shown in Table 1, the models accurately account for many characteristics including prop. delay, switching speed, drive capability, and operating current.

The features included in the models allow you to directly compare the performance of components from different vendors. You can also analyze the effects of different implementations such as peak current mode control, hysteretic current control, low voltage and low operating current to name a few. The new library has over 400 models:

- **Models for various PWM ICs including those from Unitrode, Linear Technology, Siliconix, and Cherry Semiconductor**
- **'Unified' state space model for forward, and flyback converters topologies plus state space models for several specific ICs**
- **Nonlinear Magnetic Cores and Transformers**
- **Power Mosfet Drivers**
- **Motor controller IC (UC1637)**
- **Power factor correction IC (UC1854)**

The price of the Power Supply Designer's Library is $395. The models are compatible with ICAP/4 systems on DOS, NEC, Windows, Macintosh and the Power Macintosh. ICAP/4Lite and ICAP/4Lite Xtra systems must have the IsSpice4 upgrade in order to run the new models.

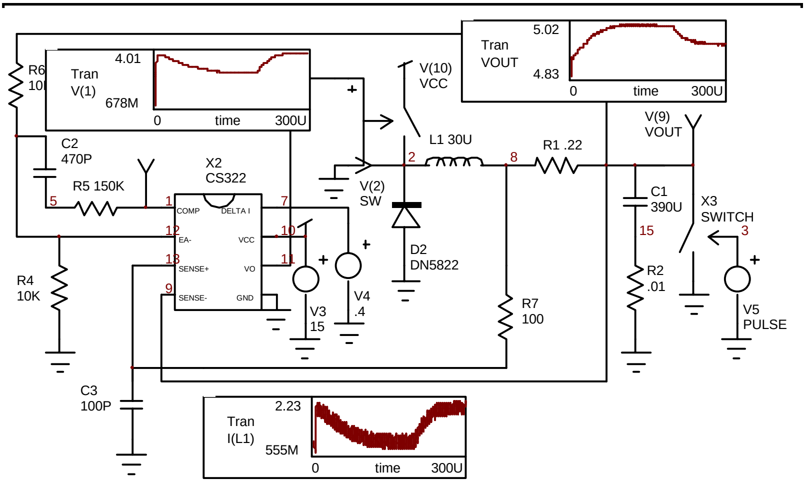
Hysteretic CMC and Buck Regulators

Figure 2 shows a current mode converter example using the Cherry CS-322. The circuit uses hysteretic control, which is different from constant frequency or constant off time control. In constant frequency control, the instant of turn-on is always independent of the closed loop dynamics of the supply thereby limiting its regulation performance. However, because hysteretic control can change both edges of the switching waveform to keep a constant inductor current hysteresis, it has no mode on instabilities; no slope compensation is required. In Figure 2, the load resistance was varied from 5ý to 2.5ý. The

UC1842/3/4/5 Model Performance Comparison

Parameter	Condition	Type Spec	IsSpice4
Reference			
Output Voltage	1mA	5V	5V
Load Regulation	1-20mA	6mV	6.2mV
Oscillator Section			
Initial Accuracy		52kHz	52.1kHz
Amplitude		1.7Vp-p	1.72Vp-p
Discharge Current Standard		10mA	9.8mA
Error Amplifier			
Input Voltage		2.5V	2.5V
AVOL		90dB	89.1dB
Unity Gain Bandwidth		1mHz	1mHz
Output Sink Current		6mA	6.07mA
Output Source Current		-0.8mA	-0.8mA
Vout High		6V	5.64V
Vout Low		0.7V	0.72V
Current Sense			
Gain		3V/V	3V/V
Maximum Input Signal		1V	1V
Delay To Output		150nS	151nS
Output Section			
Output Low Level	20mA	0.1V	0.13V
	200mA	1.5V	1.43V
Output High Level	20mA	13.5V	13.56V
	200mA	13.5V	13.43V
Rise Time		50nS	35nS
Fall Time		50nS	50nS
Undervoltage Lockout			
Start Threshold	UC1842/4	16V	16V
	UC1843/5	8.4V	8.4V
Min Operating after Turn-On	UC1842/4	10V	10V
	UC1843/5	7.6V	7.6V
PWM Section			
Maximum Duty Cycle	UC1842/4	97%	97%
	UC1843/5	48%	49%
Minimum Duty Cycle		0%	0%
Total Standby Current			
Start-up Current	UC1842/4	0.5mA	0.45mA
	UC1843/5	0.5mA	0.51mA
Operating Current	UC1842/4	11mA	11.1mA
	UC1843/5	11mA	11.5mA

Table 1, The PWM models use a mixed mode modeling approach. They combine switches, behavioral models, and logic gates in order to provide a full set of features, simulation efficiency and excellent accuracy as shown above.



converter using the Cherry CS-322 Hysteretic Current mode controller. The switching frequency is 300kHz.

CS-322 simulates very quickly; the entire 300µs simulation took only 181.9s on a Pentium/90 even with TMAX (maximum timestep) limited to 200ns. Gear integration was used to speed the simulation while Tmax was set to 200n in order to maintain accuracy. In Figure 3 we see the result when the load is short-circuited by changing the switch resistance from 5ý to 1mý in 1µs.

Figure 4 shows a 5V-to-3V, 300mA, 1MHz buck regulator example as described in the Siliconix Low Voltage DC-to-DC Converter design guide. The design uses the high speed Si9145 switchmode controller and low on-resistance (P-channel 20mý Si4435, N-channel 100mý Si9952) Little Foot® power Mosfets. Traditionally, SMPS simulations, especially with this level of detail, can take a long time to simulate. New modeling techniques, however, have brought the simulation

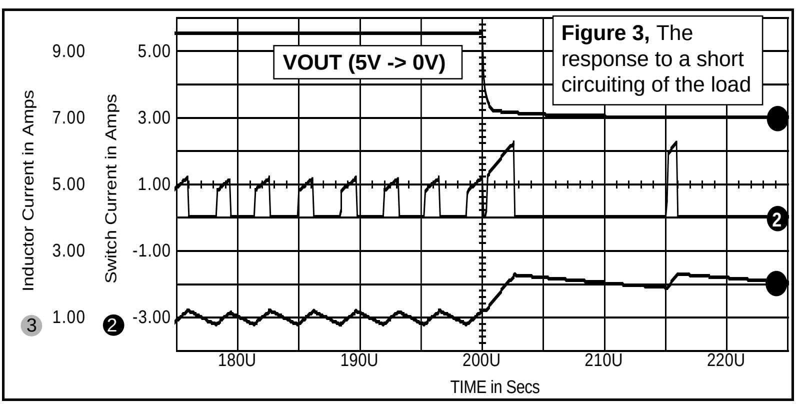
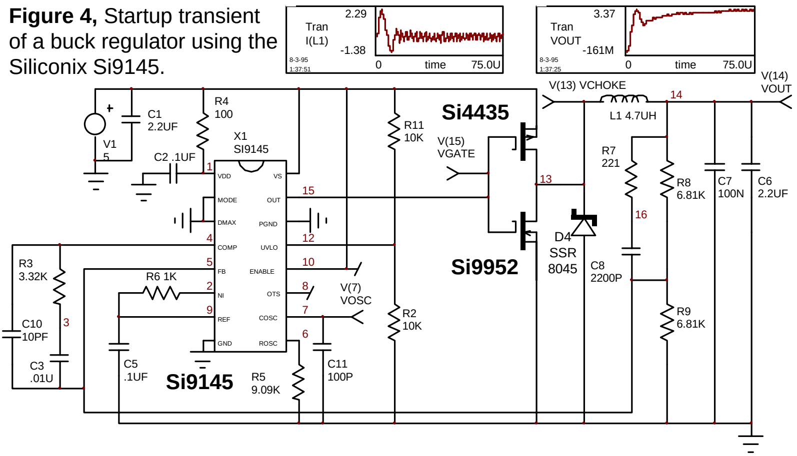


Figure 3, The response to a short circuiting of the load

Figure 4, Startup transient of a buck regulator using the Siliconix Si9145.



time down. This simulation (.tran 50n 75n 0 10n) takes only 450s on a 486/25. Starting the simulation presented some problems. The .tran UIC keyword is used to initialize the internal logic in the 9145. Unless the VDD capacitors (C1/C2) are initialized with ICs (IC=5V) the VDD pin will start low and ramp to 5V and the switching action will not begin.

New Simulation Tutorial Book for SMPS Designers

For those interested in pursing more in-depth SMPS simulation techniques and modeling, Intusoft will be releasing a book on the subject entitled "**SMPS Simulating with SPICE 3**" in the near future. Stay tuned to the newsletter for more information.

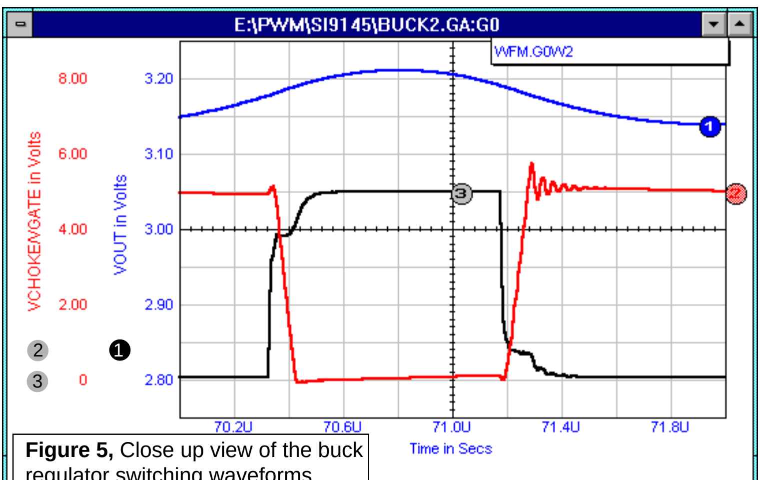


Figure 5, Close up view of the buck regulator switching waveforms.

Thanks to Steve Sandler, Analytical Engineering (602) 917-9727 for his contributions to this article and the Power Supply Library. Steve is a design consultant specializing in SMPS simulation, modeling, and design.

Simulating Pulse Code Modulation

There are applications, such as uninterruptible power supplies (UPS) that convert a DC input voltage to a sinusoidal AC output voltage. The basis of the conversion is very similar to the conversion of a DC input to a DC output voltage. One of the more difficult aspects of DC to sinewave conversion is obtaining a regulated, low distortion sinewave reference. The goal is to provide a variable frequency and amplitude with low harmonics and a zero DC term.

While pulse width modulation is a possibility, another method is to utilize a string of ones and zeros. When repeated this string will possess a Fourier series consisting of a fundamental and some harmonics. By picking all of your ones and zeros correctly, you can force most of the lower harmonics to zero and still provide a variable amplitude output that is both microcontroller friendly and free of a high frequency carrier.

The following demonstrates an unusual task for IsSpice4. The circuit in Figure 6 simulates a single bit pulse code representation of a sinewave. The same circuit is easily extended to any number of phases. The fundamental problem was generating a bit pattern for the sinewave reference. The pulse generator, V1, is used as the clock. Since we will generate 256 bit values this clock is 256 times greater than the output frequency. Flip-flop X1 latches the data between clock pulses. V2 is a sinewave used for a reference. R1, R5, C1 and C2 filter the pulse coded waveform and reconstruct the sinewave. B1 is a behavioral If-Then-Else comparator that sets the output bit high if the output is lower than the sinewave reference value, or sets the output bit low if the output is higher than the sinewave reference value. B2 level shifts the bit values to a zero-one format.

Looking at the cross-probed waveforms, you can see the bit patterns and the sinewave output at each filter stage. Note that more sophisticated filters could produce lower distortion, as could more values in the data table. The output listing from Bits_Out, node V(9) in the IsSpice4 output file, is a series of ones and zeros and is the bit pattern representing the sine wave.

Using A State Machine To Generate A Sine ROM

We could use the circuitry in Figure 6 to generate the pulse codes for other parts of a simulation, however, it is much more efficient to code the digital output into a ROM. Fortunately, IsSpice4 includes a state machine model that can store and play back the pulse code data. Table 2 shows a partial listing of the state machine input file. The data in the Output column is from

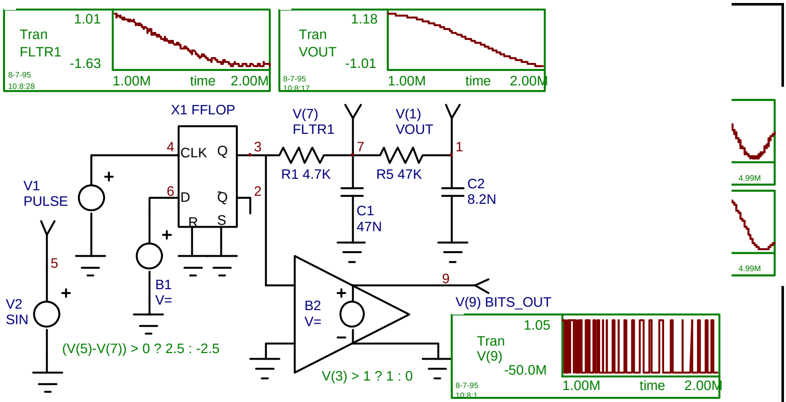


Figure 6, Circuit used to generate a sine wave and the 1 bit code representing the sine wave. A portion of the sine wave and bit stream has been cross-probed.

Figure 6 node 9. Figure 7 uses the state machine to create a model for an 8 bit counter with the 256 bit ROM. The clock frequency must be 256 times the desired sinewave frequency. The Tstep value of 9.766u in the simulation's .Tran statement (.Tran 9.766u 4.99M) is important since this is the period of the sample. Selecting a different number requires a Tmax value which slows down the simulation. Note that by changing the clock frequency we can change the sine wave frequency.

The runtime performance of the state machine is far superior to the Figure 6 method (Figure 6: 79.15 seconds, State Machine circuit: 14.50 seconds, Pentium/75) and its improved performance will become significant over the course of many runs. In addition, the IsSpice4 state machine is a C code model that is separate from the simulator. The model's source code is available to those who might wish to expand its functionality and input formats. For example, it could be possible for the state machine to accept ABEL or JEDEC descriptions.

Three Phase Sine Reference

As an extension to the single phase case, Figure 8 demonstrates a three phase sine wave reference circuit. A 6 stage shift

State	Output	Transition		
0	0s	0	->	1
1	1s	0	->	2
2	1s	0	->	3
3	0s	0	->	4
4	1s	0	->	5
5	1s	0	->	6
...	...	...	...	...
253	0s	0	->	254
254	1s	0	->	255
255	1s	0	->	0

Table 2, The input to the state machine generated by Figure 6 and used in Figure 7. The Output is the state machine output at the particular state. The Transition determines which state the machine will move to depending on the input. In this case, the input is held at 0 and the machine simply progresses state by state until state 255 where it then repeats.

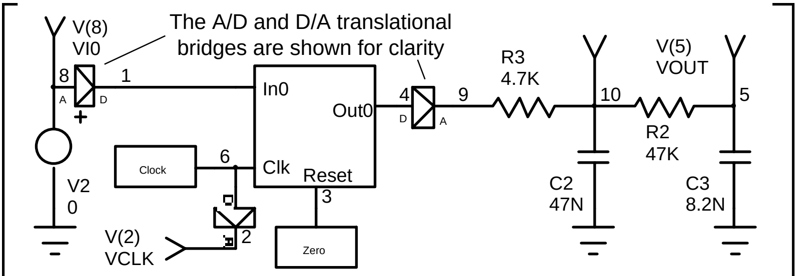


Figure 7, This circuit generates the same sine waves shown in Figure 6, however, here a ROM (state machine) is used to driver the filter.

register is used to generate 3 quasi-square waves which are exactly 120 degrees apart. Each waveform has a conduction angle of 120 degrees. The 120 degree quasi-square waveform has the advantage of having no third harmonic content; the first significant harmonic is the fifth. Each quasi-square wave is filtered by a second order active low pass filter. The quasi-square waves are created by averaging 2 square waves which are phase shifted by 60 degrees. The sinewave output distortion could be further reduced by using a higher order active filter, reducing the corner frequency of the existing filters, or replacing the quasi-square waveform with a more sophisticated waveform to eliminate several more harmonics. Care must be taken in the placement of the filters, since component tolerances could easily alter the phase angles between phases; something that could be investigated with the IsSpice4 Monte Carlo analysis. As in the previous case, a state machine model could be created to hold the waveform's pulse codes.

References

[1] "The Quest for magic Sine Waves", Don Lancaster, Circuit Cellar Ink, #59, 6/95

[2] "SMPS Simulation With SPICE 3", Steve Sandler, forthcoming from Intusoft

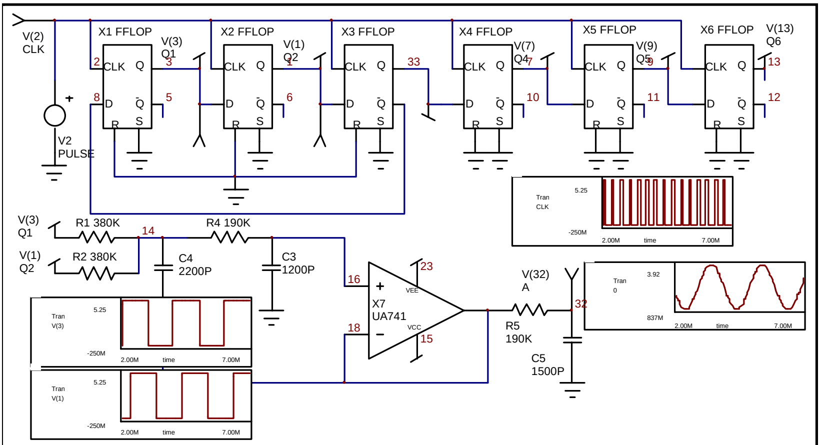


Figure 8, Portion of the circuit used to generate a 3 phase sine wave. The taps at the flip-flops are used to drive the active filters. Only one of the 3 phases is shown.

Summer Sale: ICAP/4Lite Xtra

Intusoft's ICAP/4Lite Xtra is our affordable "basic" version of ICAP/4Windows. ICAP/4Lite Xtra includes all the features of our popular ICAP/4Lite product and adds a full featured graphical waveform post processor and model libraries with over 6000 parts. ICAP/4Lite Xtra also provides an integrated schematic entry front-end and a version of the IsSpice4 simulator with unlimited circuit size. With all these features, it's hard to believe that ICAP/4Lite Xtra is only $995 (list price). ICAP/4Lite users can even upgrade to ICAP/4Lite Xtra for the $400 price difference. Keyless network versions of ICAP/4Lite and ICAP/4Lite Xtra are also available. The ICAP/4Lite Xtra offer is valid until September 30, 1995. Please contact Intusoft or your local dealer for further pricing and information.

Below are some answers to commonly asked questions about ICAP/4Lite Xtra that will help you in your evaluation process.

Questions and Answers To Help You Evaluate ICAP/4Lite Xtra

What size of circuit can I simulate? Circuit size is unlimited. There are NO built-in node or component restrictions. The only limit is memory.

What analyses can I perform? You can do AC (frequency response), DC, Transient, Fourier, Temperature, and Parametric analyses. What you can't do is noise, sensitivity, Monte Carlo, and Circuit Optimization analyses. Support is included for all SPICE 2 elements plus behavioral modeling constructs. Some of the higher level SPICE 3 elements (lossy transmission lines, GaAs Mesfets, MOS BSIM models) are not supported.

How do you compare to the competition? Our SPICE is based on Berkeley SPICE 3F.4 and XSPICE from Georgia Tech. This, along with our own extensive enhancements, gives us a number of advantages in the convergence, mixed mode, interactive simulation, AHDL, and modeling areas. Our schematic and post-processor are very tightly integrated with IsSpice4 to provide you with the most easy to use simulation environment possible.

How big is your library? ICAP/4Lite Xtra contains over 6000 models including those for standard electrical parts, Japanese & European parts, plus devices the other guys forgot to model: nonlinear magnetics, SCRs, IGBTs, PWMs (state space), MOVs, mechanical elements, thermistors, sensors, photo/laser diodes and more.

How do I enter my design? Do I have to type a SPICE netlist? You don't have to enter a netlist or deal with virtually any SPICE syntax intricacies. All entry is done graphically with our SpiceNet schematic program. With the IntuScope program you can display and analyze all of your simulation results utilizing FFTs, integration, differentiation, waveform math operations. You can even view and compare data from several output files and measured laboratory data all at the same!

ICAP/4Lite Xtra Questions and Answers, *continued*

Can I use another schematic package to drive the simulation? Yes.

Can I use the SPICE models supplied by hardware vendors? Yes.

Can I view and edit your models and add my own models? Yes.

Can I run your software on my company's network? Yes, there is a network version. In addition, NO Protection Dongle is required for the network version.

How is your support? Intusoft's support is the absolute best in the simulation industry. We are the ONLY company to offer a FREE SPICE modeling service. Other benefits include: a Free bimonthly newsletter, CompuServe/Internet sites with posted models and technical articles and support that is free from service or yearly maintenance charges.

New SPICE is Keyless and Floats

Continued from pg. 1

simulations run on your local machine. The network performance determines how quickly the software starts. The runtime performance is determined by the speed of your PC.

The network version allows any engineer on the network to have access to the circuit design tools. When one is finished another engineer can run simulation without having to have the software installed locally. And best of all, no client machine is required to have a protection dongle. The keyless version offers an enormous cost benefit, because a separate copy of the software isn't needed for each engineer on the network.

Keyless Network versions* of the ICAP/4Windows, ICAP/4Lite and ICAP/4Lite Xtra are available now. The price of the network version is equal to the maximum number of copies to be run at any one time, plus 1, times the cost of a single system (N+1 ∗ List Price). ICAP/4 owners can update their current systems for the price difference.

**** May not be available in some countries.***

SPICE 2 Book Available

For those interested in learning about analog simulators based on Berkeley SPICE 2 Intusoft has a special book available. Its called "SIMULATING WITH SPICE". The 276 pages comprehensively cover learning and using the universally accepted version of SPICE; 2G.6. It includes an easy to follow tutorial for novice users, a dozen example problems with helpful hints and an advanced techniques section with several application notes. A complete SPICE syntax reference guide with examples is also included. Best of all, this great reference book is available for just $15 (plus $10 S&H, U.S.A.) while they last.

Microsim Makes False Claims

This letter is in response to a number of false claims made by Microsim™ about their SPICE simulator Pspice® and about Intusoft's simulator IsSpice. This letter has been sent to Microsim and is reproduced here.

To: Mr. W. Blume, President Microsim
Date: 8/7/95

For several months Microsim has been advertising that "Microsim is the only company with native mixed analog/digital simulation for PCs" (See ad of June 8, 1995 in EDN). This is simply not true. In October, 1994 Microsim Corp. purchased our ICAP/4Windows software. As a registered user and Intusoft Newsletter subscriber you should have known about our native mixed mode simulator from our November 1994 newsletter and the numerous ads we have placed stating that IsSpice4 (release date 12/94) is a native mixed mode simulator.

In addition, you have been distributing an erroneous product comparison to your foreign distributors, claiming "Intusoft makes no modifications to SPICE 3E.1 and relies on Berkeley for any improvements made to the program" (that is a quote). We know that this material exists because we have obtained a "Microsim Distributor Manual" directly from one of your dealers.

Firstly, IsSpice has never been based on 3E.1. Secondly, we have never simply shipped a copy of Berkeley's version of SPICE. We have made extensive changes to Berkeley SPICE, including <u>significant</u> feature enhancements; for example, the Boolean and if-then-else syntax was introduced in 1992, and was the first ever introduced in a SPICE 3 derived simulator. Many of the bug fixes were even sent back to U.C. Berkeley where they have been incorporated in the current 3F release.

Intusoft demands that you stop these advertising practices immediately and prepare a truthful product comparison for your foreign distributors.

We have both benefited from the pioneering work of U.C. Berkeley. Intusoft adopted the more recent SPICE 3 as our simulator baseline while PSpice is derived from the older SPICE 2 release. Comparing either product with its public domain counterpart is misleading at best. In view of your past mis-statements, we believe that it is necessary for us to review and approve of the actions you plan to take to correct this unfortunate situation.

L. G. Meares - President, Intusoft

The Intusoft Modeling Corner

In this edition of the modeling corner we present models from **SSDI** and **Intusoft Tech support**. The *Intusoft Newsletter* subscription disk contains new models from Analog Devices, plus the models applicable to the switching converter example in Figure 10; a TL431 voltage reference, over 100 TIP Power BJT models, and a number of schottky diodes. Over 40 Asian-Pacific BJT and FET models are also included.

SSDI Provides IsSpice4 Rectifier Models

Solid State Devices Inc. (Irvine, CA, (714) 670-7734) has created models of their SRMx, SDR937, and SSR8045 (0.4V drop @ 20A) power diodes. SSDI specializes in Power semiconductors. The SPICE model netlists are shown in Figure 9. The diode models were made using average measurements from 10 randomly selected parts of each type. More models are forthcoming and will be available on future *Intusoft Newsletter* floppies for subscribers.

How Accurate Are The Models?

The accuracy of the models is controlled by the SPICE parameters. Forward voltage, including temperature effects, is characterized by the IS, N, RS, XTI, and EG parameters. The following table provides the actual measured and simulated data along with the error. All of the simulations were performed using the ICAP/4Windows software.

IF (A)	Simulated VF	Measured VF	%Error
5	0.739	0.738	-0.042
10	0.787	0.786	-0.116
20	0.838	0.838	-0.044
50	0.92	0.92	0.040
100	1.002	1.002	-0.079

```
.MODEL SRM1UF D (IS=4.9E-5 RS=.77M N=2.45 TT=65.6N BV=303.5
+IBV=100U M=.252 CJO=669.7P VJ=.75 XTI=4) ; 100V 20A

.MODEL SRM3UF D (IS=9.7E-5 RS=.74M N=2.78 TT=65.6N BV=486
+IBV=100U M=.102 CJO=211.1P VJ=.75 XTI=4) ; 300V 20A

.MODEL SRM5 D (IS=.0210 N=7.5 RS=.71M TT=1.344E-7 BV=1111
+IBV=100U M=.348 CJO=467.4P VJ=.75 XTI=28) ; 500V 20A

.MODEL SRM6UF D (IS=59.6M RS=.63M N=11.5  TT=129.2N BV=1237
+IBV=100U M=.410 CJO=709.1P VJ=.75 XTI=58) ; 600V 20A

.MODEL SRM5SOFT D (IS=4.95E-7 RS=1.1M N=1.96 TT=11.52U BV=1128
+IBV=100U M=.39 CJO=678.3P VJ=.75 XTI=4) ; 500V 20A

.MODEL SSR8045 D (IS=26M RS=3.57M N=2 BV=50 IBV=5M
+CJO=2.86N VJ=.75 M=.333 TT=14.4P ; 45V 40A

.MODEL SDR937 D(IS=8.359E-6 N=2.458 RS=1.93M CJO=600P M=0.6 VJ=0.34
+ IBV=100U BV=674 TT=30N EG=1.15) ; 700V 100A
```

Figure 9, SSDI has created several new IsSpice4 diode models for their line of power devices.

High Efficiency Step-Down Converter

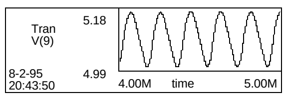
```
.SUBCKT TL431     7  6  11
*                 K  A  FDBK
.MODEL DCLAMP D (IS=13.5N RS=25M N=1.59
+ CJO=45P VJ=.75 M=.302 TT=50.4N BV=34 IBV=1M)
V1 1 6 2.495
R1 6 2 15.6
C1 2 6 .5U
R2 2 3 100
C2 3 4 .08U
R3 4 6 10
G2 6 8 3 6 1.73
D1 5 8 DCLAMP
D2 7 8 DCLAMP
V4 5 6 2
G1 6 2 1 11 0.11
.ENDS
```

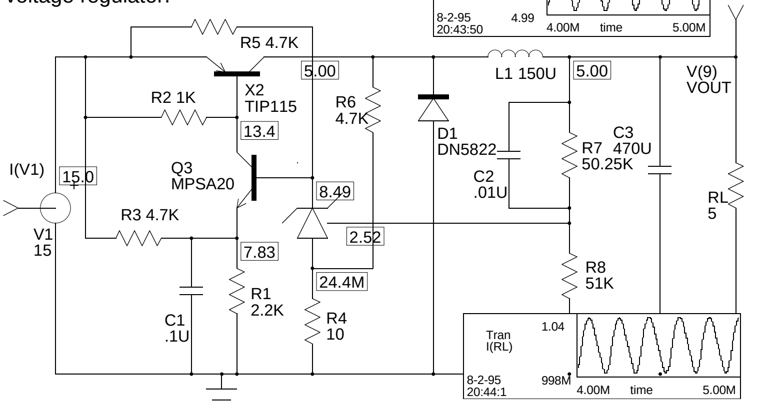
Figure 10, The SPICE 2 model for the TL431 regulator.

The TL431 programmable precision reference (Figure 10) may be used to implement a low cost stepdown switching converter. The TL431 performs the function of both a voltage reference and a voltage comparator, all contained within a three pin package. The programmable output voltage feature permits a wide range of output voltages by changing one resistor value. The test circuit (Figure 11) achieved a simulated efficiency of over 70% for a 5 volt output. Other simulated results are shown in the figure, along with measured results from the Motorola Linear IC data book. Initial conditions were set on the output LC filter elements. The circuit did not require these initial conditions to start the circuit switching, but did require that the maximum transient time step be specified. The IsSpice4 simulation revealed that the converter operation is very sensitive to parasitics which explain some of the differences between simulated and measured results.

Test	Conditions	Simulated Results	Measured Results
Line Regulation	Vin=10-20V Io=1A	200.2mV	53mV
Output Ripple	Vin=10V Io=1A	41.83mVp-p	50mVp-p
Output Ripple	Vin=20V Io=1A	125.58mVp-p	100mVp-p
Efficiency	Vin=15V Io=1A	70%	82%

Figure 11, Low cost, high efficiency step-down converter using the TL431 voltage regulator.





Simulating In The 90's

Running SPICE is no longer the same as it was when the program was first introduced. Schematic entry has replaced card decks, interactive simulations that now take only seconds have replaced day long mainframe runs, and real time graphical data processing has replaced crude line printer plots. This is the state of simulation today. Changes brought about by new software products, such as ICAP/4 , have had a profound effect on how engineers simulate circuits.

ICAP/4 is an integrated simulation system that includes 4 modules, each one performing a different function.

- **SpiceNet** Integrated Schematic Entry
- **Models** SPICE & HDL Model Libraries
- **IsSpice** Native Analog Mixed Mode Simulation
- **IntuScope** Data Processing and Analysis

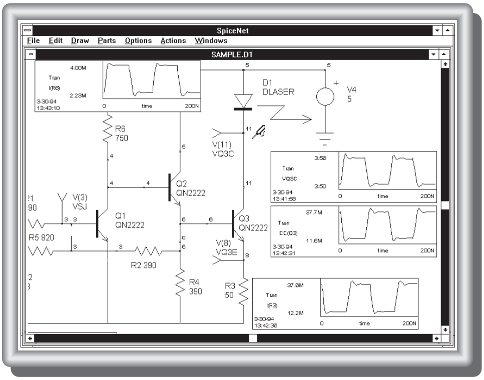
With ICAP/4 you can cut through the toughest circuit design problems with ease, create better products with more functions and higher yields, and explore new concepts. From power to RF to mixed mode, ICAPS allows you to analyze and predict the performance of all types of circuits.

Intusoft has been a leader in full featured design tools since our first product, IsSpice, was released over ten years ago. Our family of software, integrated under the ICAP/4 environment, reduces engineering and manufacturing costs, increases yields, and slashes repair, testing, and design time. The following sections contain detailed information about the ICAP/4 system. They will assist you in discovering the power of ICAPS.

SpiceNet: Integrated Schematic Entry

Description: SpiceNet is a schematic entry program that is designed to be an interactive front-end to IsSpice4. It greatly eases the burden of creating a SPICE netlist by generating a complete netlist, ready for simulation, directly from the schematic. Unlike other schematic packages, which are geared for digital circuits or PCB layout, SpiceNet supports all facets of SPICE. SpiceNet alleviates the editing and syntax headaches allowing you to spend your time creating a better design instead of debugging typos.

Benefits: Schematic entry with SpiceNet is designed to be faster than pencil and paper. Most components can be placed on the schematic with a single keystroke. And with all its functions on pull

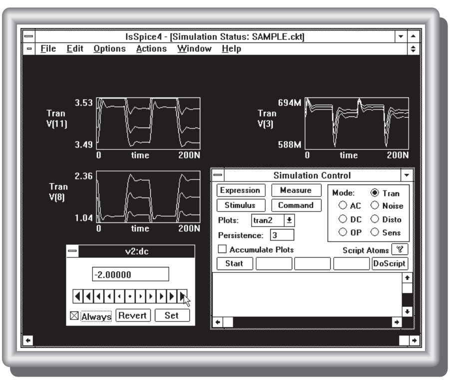


down menus, you can input and simulate your first design in less than an hour. SpiceNet has a direct interface to IsSpice4 allowing waveforms to be interactively cross-probed directly on the schematic. Additionally, the circuit operating point voltages can be updated as component values are changed.

Schematic Entry Features

- Produces a complete SPICE netlist, no editing necessary
- Runs a simulation directly from the schematic
- Interactively cross-probe waveforms by clicking on a node or device
- Change values and resimulate directly from the schematic
- Place parts by part number or from a list
- A Preferred Parts Menu can be defined by the user
- On-line symbol editor plus pre-made symbols for every model
- Automatic subcircuit maker
- Special easy to use pop-up dialogs for SPICE control statements
- Compatible with any SPICE simulator
- Multiple page schematics, Edit several schematics at one time
- Schematics are compatible between versions and platforms
- Cut and paste between different schematics
- Report quality graphics: supports all Windows and Macintosh Chooser output devices

IsSpice: Analog/Mixed Signal Simulation



Description: The new *Interactive* IsSpice4 program provides a quantum leap in performance over other SPICE simulators. It allows you to explore circuit performance by interactively running different analyses and sweeping any circuit variable. Analyses include AC, DC, Transient, pole-zero, noise, sensitivity, Fourier and distortion analyses. Circuit temperature variations are available for all analyses and individual elements.

Benefits: The advanced features of IsSpice4 allow all types of analog and mixed mode applications to be simulated like: switch mode power supplies, mixed signal ASICs, RF communication systems, interconnect problems, control systems, and mixed domain (mechanical/physical) systems. There are several IsSpice versions, described next, that vary in speed, circuit size, operating system, and built-in model/analysis support.

- IsSpice4 is 32-bit version of SPICE 3F.2 for Windows, Windows NT (x86, Mips, Alpha) and Macintoshs. It supports unlimited size circuits, waveform cross-probing, real time waveform display, AHDL models, simulation scripts and breakpoints, and is a native mixed mode simulator.*

- IsSpice3 provides the same analysis, model, and real time waveform support as IsSpice4 except that it runs on DOS, Macintosh, and Power Mac systems and is not interactive.

IsSpice (Analog/Mixed Mode Simulator) Features

Analysis and Built-in Models (All Versions)

- Elements: Resistors, Capacitors, Inductors, Coupled Inductors, Transmission Lines, Diodes, BJTs, JFETs, MOSFETs (Level 1,2, and 3), Subcircuits, Independent/Dependent sources (SPICE2 polynomials)
- AC, DC, transient, noise, Fourier, distortion, temperature, DC sensitivity
- Monte Carlo Analysis: Statistical yield analysis of circuit performance
  - Randomly vary circuit parameters to test performance
- Circuit Optimization/Performance Analysis: Circuits can be optimized based on a user defined objective function.

Additional IsSpice3/IsSpice4 Features

- Real-time waveform display of voltages, currents and power dissipation
- Elements: GaAs Mesfets, MOSFETs Levels 4, 5, and 6, Lossy T-Lines, voltage/current ctrl'd Switches, and Boolean logic expressions.
- Analyses: AC sensitivity and Pole-Zero analyses, Temperature variations on individual elements
- Behavioral Modeling: In-line Equations, Table models, If-Then-Else
- Simulation Scripts: a robust scripting language that allows simulation breakpoints and loops of different analyses to be run as a test procedure.

Additional Interactive, AHDL & Mixed Mode IsSpice4 Features

- Interactively run analyses without having to edit the netlist or restart the simulator, Add, delete, or rescale waveforms on the real-time display
- Native Mixed Mode: IsSpice4 includes a 12 state digital logic simulator and models with timing information *****
- Sweep parameters one at a time or in groups with great ease
- Start, stop, pause, change, or resume any analysis on demand
- Use C code subroutines & AHDL models based on XSPICE *****
- Elements: Digital logic gates, Flip-Flops, Latches, State Machine, Freq. Div., RAM, Sampled-Data Filters, Nonlinear VCOs, Laplace Equations *****
- Supports interprocess communication and control via shared memory

Compatibility

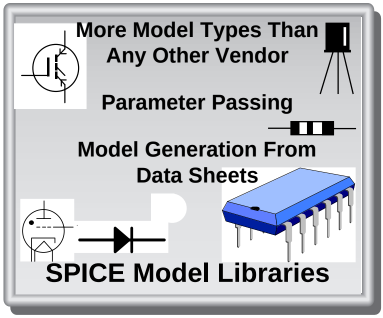
- <u>Works with ALL popular schematic entry programs</u>
- Accepts Berkeley SPICE 2G.6 or 3F.2 syntax; Outputs SPICE 2 and 3

Adding C Subroutines To IsSpice4 *

The Intusoft Code Modeling Software Development Kit (CMSDK) allows you to add your own C code subroutines to IsSpice4. The C code is the basis for XDL, a new HDL (Hi-level Description Language). XDL models are like traditional SPICE models, except they are created by you. The CMSDK is required for model development, however, any IsSpice4 program can then use the newly developed models.

*** Windows Only**

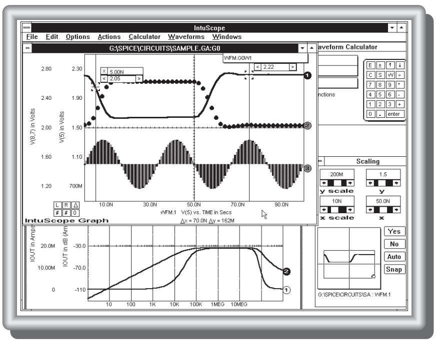
SPICE Model Libraries



Description: ICAP/4Windows and Macintosh include an extensive array of over **<u>6000 models</u>**.
Benefits: The model libraries contain a wide variety of models including diodes, zeners, BJTs, Darlingtons, op-amps, comparators, transformers, nonlinear magnetics, JFETs, SCRs, IGBTs, Triacs, power MOSFETs, PWMs, SC filters, analog behavioral models, digital logic gates, switches, opto-isolators, transmission line models, crystals, vacuum tube models and more. Over 100 "Generic Template" models that convert data sheet parameters into SPICE parameters are also included. Models are stored in ASCII text files that can be viewed and edited. A complete list of models is available. **Note:** The vendor supplied IC libraries (over 1300 models), the RF Device Library (over 300 models) and the Power Library (over 400 models) are available separately.

Model Generation From Data Sheets: A separate program (SpiceMod 2.0) is available for creating SPICE models from data sheet parameters.

IntuScope: Graphical Waveform Processing



Description: IntuScope is an interactive graphical data processing program especially designed to display and analyze IsSpice output data. IntuScope can display waveforms from any Berkeley SPICE compatible program, as well as user generated data files.

Benefits: IntuScope is more than just a SPICE post processor. It is a very powerful data processing system. It displays data as waveforms and contains a comprehensive set of waveform processing functions and operations.

Data Analysis Features

- Displays all circuit voltages, currents, power dissipations and more
- Accepts output from any SPICE program or user generated data files
- Can save any displayed waveform for use as circuit stimulus
- 32-bit version allows large waveforms to be displayed and analyzed
- Various scaling formats include linear, semilog, histogram, and probability
- Multiple graphs with multiple independent scales
- Waveform Operations: RMS, Pk-Pk, Mean, Max, Min, cursors
- Add, subtract, multiply, and divide waveforms
- Math Functions: trigonometric, log, power, e^x, algebraic
- Advanced Waveform Functions: Integrate, differentiate, FFT, polynomial regression, filtering, gain/phase margin prop delay, rise/fall time
- Report quality output similar to SpiceNet

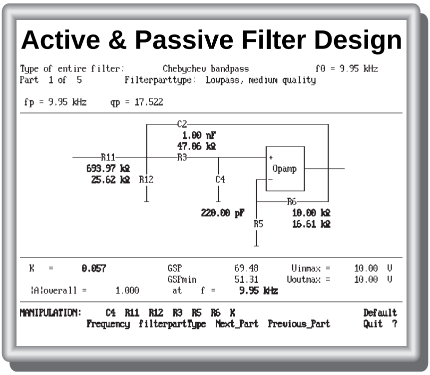
Additional SPICE Related Products

RF Device Library version 3.0
Description: This is a special SPICE model library for those users performing simulations at or above 200MegHz. It contains models for over 300 different RF devices including bipolar transistors, FETs, MMICs, GaAs Mesfets, PIN Diodes, and RF beads.
Benefits: The RF library allows any SPICE program to simulate high frequency circuits using linear and nonlinear AC, DC, and Transient analyses. This capability was not available before because of the lack of quality subcircuit based models. All models are characterized up to their published s-parameter data.

SPICE Reference Books
"SPICE Applications Handbook, 2nd Edition" **- Collections of past *Intusoft Newsletters*, 6/86 - 2/94 (34 in all!).**
"A SPICE Cookbook" - Over 100 practical circuit examples encompassing a wide array of topics (RF, Power, Filters, Digital) and how they were simulated with SPICE.

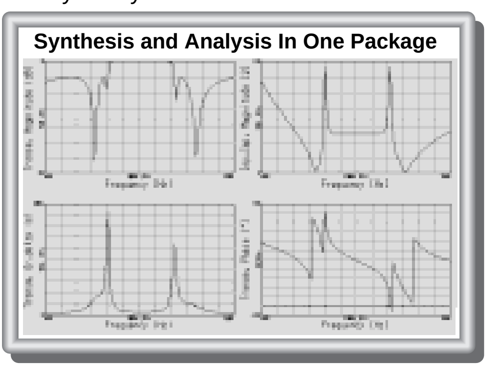
FilterMaster: Filter Design

Active/Passive Filter Design
Description: The FilterMaster Design Series is a set of PC-based programs used for the synthesis, and analysis of analog LC (lumped element) and active RC filters. Low-pass, high-pass, bandpass, and band-stop filters can be synthesized. Available approximations include: Elliptic (Cauer), Butterworth, Chebyshev, Inverse Chebyshev or, Bessel (for low-pass filters), and two general amplitude approximations.



Active & Passive Filter Design

Benefits: The FilterMaster Design Series includes both synthesis, as well as analysis capabilities, allowing filter topologies and characteristics to be easily compared for the optimal results. Once a filter is designed, it can be transferred directly onto your SpiceNet schematic and simulated with IsSpice.


Synthesis and Analysis In One Package

Special Interfaces
- Interface to the SpiceNet schematic entry program allowing inclusion of designed filters directly onto your schematic.
- Direct output of subcircuit and stand-alone SPICE netlists.
- Output of component tolerances for use with Monte Carlo statistical yield analysis (passive only).