



Copyright © Intusoft, All Rights Reserved
(310) 833-0710 Fax (310) 833-9658

New Keyless Network SPICE

Do You Hate Dongles? Is your computer networked? If the answer is yes, then Intusoft has a key new simulation package for you. Or should we say, keyless. Dongleless network versions of both the ICAP/4Windows and ICAP/4Lite packages are available as of June 27, 1995. With the network version, the ICAP/4 software is installed on your server. Any client machine on the network can then run the ICAP/4 software. Special provisions have been made for clients to have their own private model libraries, symbols, and preferences.

Continued on pg. 7

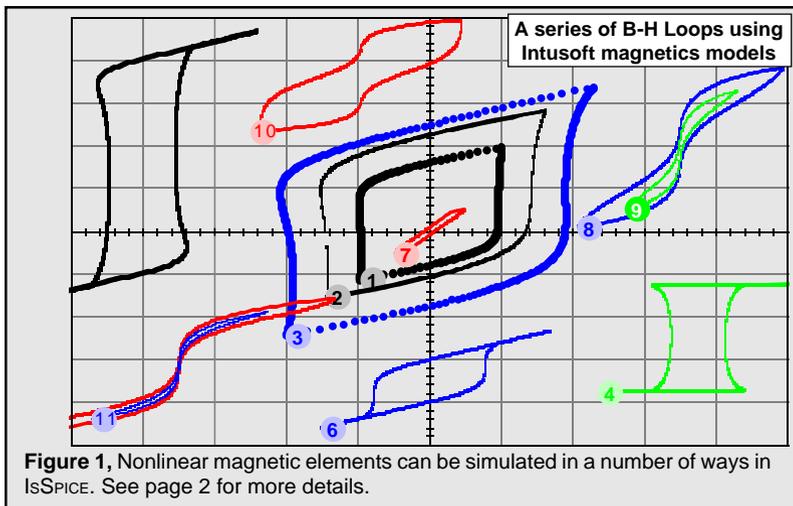
In This Issue

- 2 **Simulating Nonlinear Magnetics**
- 7 Mac Competitive Disc.
- 8 Summer Sale: ICAP/4
- 9 Training The Old Way
**** SMPS Designers ****
- 10 **NEW PWM Library**
- 12 SGS Timer & Maxim Op-Amp Models
- 14 Simulating In The 90's

Summer Sale ICAP/4Lite Xtra, \$995 'til 9/95

If you've been thinking about getting some simulation software, may we suggest a special offer. This summer Intusoft is making you an unbeatable deal on SPICE.

Continued on pg. 8



Modeling Nonlinear Magnetics

Many models for magnetic cores exist [1]. Some have been added to SPICE as built-in models while others use a subcircuit approach. In terms of modeling, however, it's important to understand what SPICE behavioral modeling can offer vs. direct coding methods in order to determine at what point an AHDL implementation is appropriate. Behavioral constructs in SPICE today are plentiful and powerful. As an example, let's take a look at two subcircuit based magnetic core models that use SPICE 2G and SPICE 3 behavioral modeling.

A saturable reactor is a magnetic circuit element consisting of a single coil wound around a magnetic core. The presence of a magnetic core drastically alters the behavior of the coil by increasing the magnetic flux and confining most of the flux to the core. The magnetic flux density, B , is a function of the applied MMF, which is proportional to ampere turns. The core consists of a number of tiny magnetic domains made up of magnetic dipoles. These domains set up a magnetic flux that adds to or subtracts from the flux set up by the magnetizing current. After overcoming initial friction, the domains rotate like small DC motors, to become aligned with the applied field. As the MMF is increased, the domains rotate one by one until they are all in alignment and the core saturates. Eddy currents are induced as the flux changes, causing added loss.

Modeling the physical process performed by a saturable core is most easily accomplished by developing an electrical analog of the magnetic flux. This is done by integrating the voltage across the core and then shaping the flux analog with nonlinear elements to cause a current to flow proportional to the desired function. Intusoft has created a SPICE 2 compatible subcircuit model that accurately simulates nonlinear core behavior including saturation, hysteresis, and eddy current losses [2, 3].

The popular Jiles-Atherton magnetics model, used in some SPICE programs, is based on existing ideas of domain wall motion, including flexing and translation [4]. While a good description of core behavior, the Jiles Atherton model is difficult to use unless you have access to parameters like the pinning energy per volume, thermal energy, interdomain coupling, and domain flexing value. Invariably, the model can only be constructed by trial simulations and tweaking of the model parameters; if you know how. In contrast, the Intusoft model is much easier to use and accepts commonly available electrical or physical data sheet parameters.

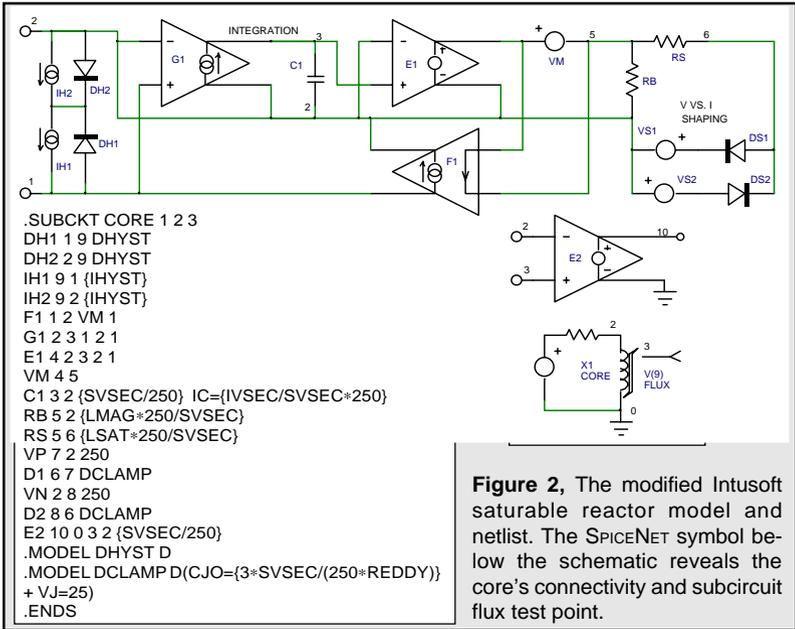


Figure 2, The modified Intusoft saturable reactor model and netlist. The SPICENET symbol below the schematic reveals the core's connectivity and subcircuit flux test point.

- SVSEC** Volt-sec at Saturation = $B_{SAT} \cdot A_E \cdot N$
- IVSEC** Volt-sec Initial Condition = $B \cdot A_E \cdot N$
- LMAG** Unsaturated Inductance = $\mu_o \mu_r \cdot N^2 \cdot A_E / L_M$
- LSAT** Saturated Inductance = $\mu_o \cdot N^2 \cdot A_E / L_M$
- IHYST** Magnetizing I @ 0 Flux = $H \cdot L_M / N$
- REDDY** Eddy Current Loss Resistance

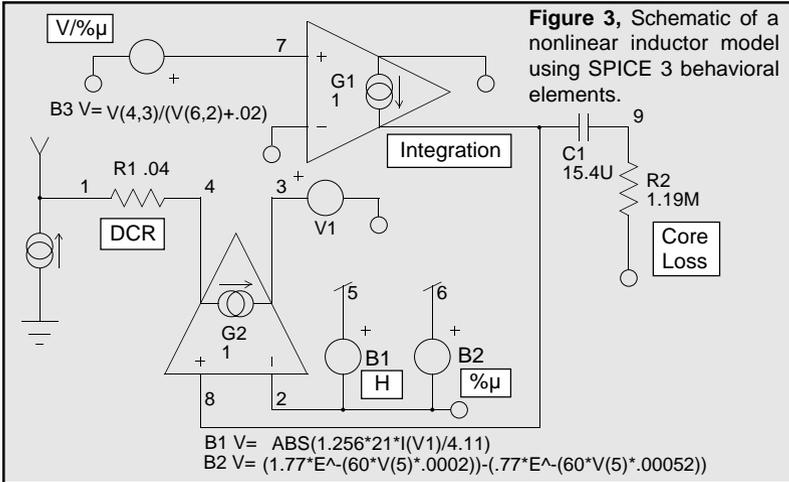
Table 1, Description of the model parameters for the modified Intusoft saturable core subcircuit shown in Figure 1.

Svsec and **Ivsec** are based on peak flux values. **Lmag**: For an ungapped core, $L = L_M$, (total path around core), For a gapped core, $\mu_r = 1$, $L =$ gap length. $A_E =$ core area, m^2 , **Lsat**: Use core dimensions but with $\mu_r = 1$, **Reddy**: Equals Lmag reactance when permeability vs freq. is 3dB down.

The original Intusoft model had one shortcoming; it did not include low frequency hysteresis. A modified version, shown in Figure 2, solves this with the addition of 4 elements to the input. Magnetizing current associated with low frequency hysteresis is provided by current sinks IH1/IH2. With no voltage across terminals 1 and 2, these currents circulate through their respective diodes, and the net terminal current is zero. When voltage is applied, the appropriate diode starts to block and its current sink becomes active [3]. The model takes into account frequency dependent losses associated with eddy currents and transient widening of the B-H loop (Figure 1, Table 1) caused by magnetic domain angular momentum. It provides excellent results for most applications, including magnetic amplifiers.

Simulating Nonlinear Inductors

The Magnetics molypermalloy powder (MPP) core is widely used in power conversion circuits. These cores are ideal for use



in power inductors and for flyback power transformers. The cores are available in several types. The most popular are currently the 55xxx and 58xxx series. The 55xxx series has a maximum flux density of 7000 gauss and the 58xxx series has a maximum flux density of 15000 gauss, both of which are significantly higher than ferrite. Core loss of both materials are significantly higher than ferrite, and the 58xxx material has a much greater core loss than the 55xxx series.

The MPP material provides a fairly “soft” B-H loop which is ideal for applications where it is desirable to create a swinging inductance that allows a significant decrease in inductance as a function of the inductor current. This can be beneficial in applications where optimum transient response is required or to minimize the preload current of a switching power supply.

For the model to be useful, it must correctly represent the initial inductance, the incremental inductance, which represents the inductance under a DC biased condition, and the core loss.

```
.SUBCKT MP55135 1 2 10 {N=1 DCR=.01 IC=0}
R1 1 4 {DCR}
V1 3 2
G2 4 3 8 2 1
G1 2 8 7 2 1
C1 8 9 {N^2*62*1n} IC={IC} ; {} = N^2*AL = L
R2 9 2 {48.2M*300^1.451/(62.*N^2)}
B1 5 2 V=ABS(1.256*{N}*I(V1)/0.817) ; 1.256 * NI/L, .817=LM
B2 6 2 V=(1.77*E^(300*V(5,2)*.0002))- (.77*E^(300*V(5,2)*.00052))
B3 7 2 V=V(4,3)/(V(6,2)+.02) ; 300 in B2 is the U(avg. permeability)
B4 10 0 V=V(6,2)*{N^2*62*1n} ; Inductance Test Point
.ENDS
```

Figure 4, SPICE 3 subcircuit listing for the nonlinear inductor model. N is the number of turns, DCR is the series resistance, and IC is an optional initial condition. The models uses the mathematical equation feature of SPICE 3.

Figure 5,
Inrush current for an inductor and the MPP 58121 and 55121 cores from Magnetics.

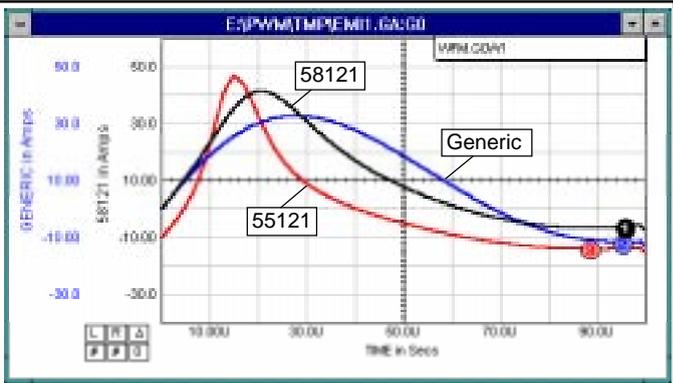


Figure 3 shows the core subcircuit while Figure 4 shows a sample netlist and key core parameters.

Initial Inductance, DC Bias, Core Loss, & Model Operation

The initial inductance is simplified by a core parameter, AL (inductance/1000 turns). The inductance in microhenries is simply computed as $N^2 * AL$. The DC bias inductance and frequency response are provided by the manufacturer in the form of graphs for each permeability available in each of the MPP materials. Using linear least squares, and some trial and error, a continuous function (difference of two exponentials) was derived which correctly calculates the % of initial permeability as a function of the DC bias in Oersteds and the initial permeability U_i . A continuous function was also derived for the loss as a function of the initial permeability U_i .

For 55xxx material: $\%U_i = 1.77e^{-0.0002\mu H} - 0.77e^{-0.005\mu H}$

For 58xxx material: $\%U_i = 1.25e^{-0.0001\mu H} - 0.25e^{-0.005\mu H}$

55xxx material: $f = 3.3E9 * e^{-1.451}$, 58xxx material: $f = 3.3E8 * e^{-1.09}$.

V1 senses the current through the inductor. B1 uses this current to calculate the magnetizing force of the inductor in Oersteds. The magnetizing force is used by B2 to calculate the percentage of initial inductance. G1 along with C1 ($=N^2 * AL=L$) and B3 perform an integration whose result is passed to G2. G2 generates the inductor current. R2 adds a zero at the -3dB frequency of the MPP material to model the core loss.

Example - Inrush Current

A simple circuit was created to compare the results of a generic inductor and the Magnetics 55xxx and 58xxx models. The results of the simulation, in Figure 5, show that the inrush current is 25% greater with a 58121 MPP core versus the generic inductor model and 50% greater with a 55121 MPP core. In many applications this could cause concern over additional stress on the capacitors, long term reliability degra-

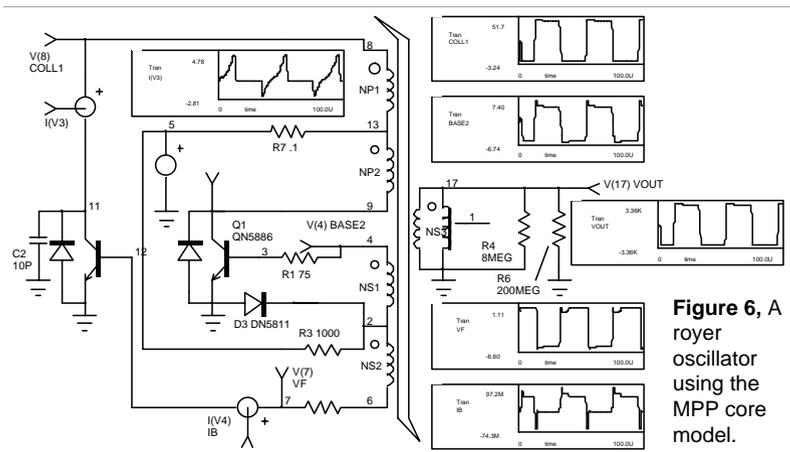


Figure 6, A Royer oscillator using the MPP core model.

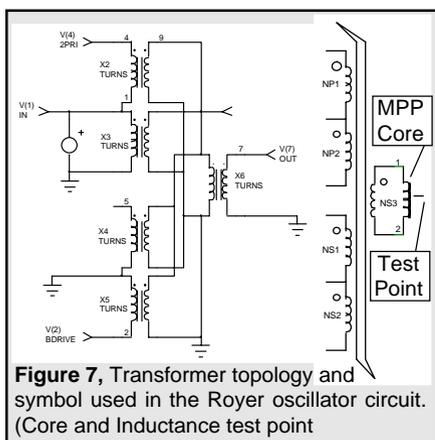


Figure 7, Transformer topology and symbol used in the Royer oscillator circuit. (Core and Inductance test point)

ation, relay contacts and connector pins which could be damaged by these currents. Similar results can be seen in characteristics such as output ripple and EMI filter attenuation.

As a typical example, the MPP core was used in a Royer oscillator circuit shown in Figure 6. The transformer posed an interesting simulation challenge. The resulting structure, created from individual transformers [2], is shown in Fig 7.

Many engineers have asked Intusoft technical support if SPICE can support nonlinear magnetics. The preceding work should give the user a good indication of some of what SPICE and behavioral modeling techniques can accomplish.

Thanks to Steve Sandler, **Analytical Engineering Services** (602) 917-9727 for his contributions to this article and for the creation of the nonlinear inductor model.

References

- [1] Survey of Magnetic Core Models, Peter Lauritzen, Margarita Takach, APEC Conference Proceedings 1995
- [2] "Improved Spice model simulates transformer's physical processes", L.G. Meares, Charles E Hymowitz, EDN, August 19, 1993
- [3] "An Electrical Circuit Model for magnetic Cores", Lloyd Dixon, Unitrode Corp., Oct 1994
- [4] "Theory of ferromagnetic hysteresis", D.C. Jiles and D.L. Atherton, Journal of Magnetism and Magnetic Materials, 1986

New Keyless Network Version

*Continued
from pg. 1*

The software is licensed for a specific number of copies that "float" on your network (Windows or Novell). This can save your department a great deal of money since individual copies do not have to be installed on specific machines. And network simulation performance is as good as that of stand-alone versions.

Do You Hate Dongles? - If you hate having a protection key on the back of your computer, then the ICAP/4 network version is for you. A protection key IS NOT REQUIRED. A key isn't even required on the server!* **This makes Intusoft the ONLY CAE vendor delivering SPICE 3 without a key!!**

ICAP/4 owners can update their current systems. The price of the network version is equal to the maximum number of copies that you want to be able to run at any one time, plus 1, times the cost of a single system ($N+1 * \text{List Price}$). Thus, a 3 user license is four times the single license price. Contact Intusoft or your local dealer for further pricing and information.

** May not be available in some countries.*

Mac Users... Competitive Discount!

Has the vendor of your Macintosh SPICE software recently abandoned you? Some vendors may have decided to stop selling Macintosh software, but not Intusoft. So if you're using a product that's now unsupported Intusoft would like to make you a special offer. For current owners of PSpice® for the Macintosh* you can take advantage of a **20% discount** when you order an ICAP package for the Macintosh. But don't wait too long. This offer is valid only for a limited time (**until August 4, 1995**).

All ICAP systems come with integrated schematic entry. You don't have to enter netlists by hand. Intusoft also offers compatibility with PSpice specific syntax. Generally, models and circuits may be run as is. In any case, Intusoft offers FREE PSpice model translation and assistance on adapting your existing circuit simulations. With the native code version, ICAP/4PM, Intusoft supports ALL Macintosh platforms including the Power Macintosh!

Prices and Limited Time Offer

Due to added support and development costs associated with the Macintosh and Power Macintosh line of computers, the price of Macintosh products will be going up August 4, 1995. So now is the time to purchase simulation tools and support that will last you for years to come.

*Offer does not extend to evaluation, student, or demo versions. Other competitive products of equal or greater value may also qualify. Contact Intusoft or your local dealer for more information. Product serial # and a photocopy of distribution disk #1 must accompany order. Offer good until 8/4/95.

ICAP/4Lite Xtra: Summer Sale

Continued
from pg. 1

The new **ICAP/4Lite Xtra** includes ICAP/4Lite, the INTUSCOPE and Model Library Lite upgrades, plus the vendor IC libraries (total value > \$1680) **for a single low price of \$995** until 9/95. ICAP/4Lite Xtra gives you more power than any other system at double its price! And unlike the competition, you can upgrade to the full ICAP/4Windows system for the price difference.

ICAP/4Lite, ICAP/4Lite Xtra, and ICAP/4Windows are available in node locked or keyless network versions*. ICAP/4Lite owners can update their current systems to ICAP/4Lite Xtra for the price difference, \$400. Please contact Intusoft or your local dealer for further pricing and information. Available Now!

Table 2, ICAP/4Lite Xtra represents a great opportunity to obtain a price-performance combination that's hard to beat.

Simulation Environment

• Unlimited Circuit Size

- AC, DC, Transient, Temperature and Parametric analyses
- Digital simulation using Boolean expressions and exact transistor gates
- Mixed Technology Simulations (physical, mechanical, non-electrical)
- Interactive analyses and parameter sweeping with ICL Scripts
- Real time waveform display of ALL voltages, currents, & power dissipations
- Advanced SPICE 3F Convergence Algorithms

Elements and Models

- SPICE 2G & SPICE 3F (Behavioral, Scripts, Breakpoints) elements
- Library Size (Analog parts) >4500, (Digital parts) >175
- Includes all standard parts and Japanese & European models, plus nonlinear magnetics, SCRs, IGBTs, PWMs (state space), MOVs, Mechanical elements, Lamps, Fuses, Thermistors, Pressure Sensors, Photo/Laser Diodes and more, plus over 1300 IC/Op-amp models
- Behavioral Modeling: Math expressions, If-Then-Else, Table models
- Special Signal Generator Library

Graphics and Post Processing

- Integrated schematic entry program
- Operating point voltage display directly on the schematic
- Simulates directly from schematic with dialog setup
- Full INTUSCOPE (Graphical Waveform Processing & Analysis) Power
- Multiple windows and Curve Families with independent scaling
- Schematic compatibility with full version and across operating systems

General

- Network version with NO Protection Dongle available* (key required for stand-alone version only)
- FREE bimonthly newsletter & modeling service for registered customers
- 30 day money back guarantee & **Low Cost Upgrade Path**

*May not be available in some countries

Training The Old Fashioned Way

Questions To Ponder: Why has Microsim™ hired RCG Research (an independent consulting firm) to hold training classes for PSpice®? Why doesn't Microsim offer its own training?

Answer: Well, there are many possible explanations. But it should tell you something important. At Intusoft, at least, customer support doesn't end with your software purchase.

Intusoft has offered SPICE training classes for over 8 years. Our classes have been held at conventions, like Wescon and Electro, at institutions like UCLA and DeVry, and at companies like McDonnell Douglas and Philips. Over 400 students have benefited from our classes not to mention thousands who have learned simulation with our popular SPICE reference books.

Why does Intusoft offer training and other vendors don't? Because at Intusoft we're engineers like you. We just happen to know SPICE inside and out and care enough to want to impart that knowledge to you! While not everyone takes the training class, every Intusoft user benefits from the experience we gain in giving the class, and the personal feedback we receive, through SUPERIOR documentation and technical support.

The Intusoft training class is customized and includes no more than 6-8 people. This makes it possible to concentrate on your personal simulation problems. So, if you need the combination of a solid SPICE simulator and a company committed and able to support you, then maybe its time you checked out Intusoft.

Abbreviated Intusoft Training Seminar Schedule

Morning

SPICE History, SPICE 2 and SPICE 3 Syntax
Interactive vs Batch Simulation
Hands-On Example Simulation: RLC circuit
Schematic Entry and Data Post Processing
Hands-On Example: Common Class problem chosen from a list
How To Effectively Simulate; Convergence and Problem solving

Afternoon

Basics of Semiconductor Modeling; Modeling a Diode
Subcircuit Macromodeling with Targeted Examples
Hands-On Example Simulation: Personal design problem
Mixed Mode (Analog-Digital) Simulation
Evaluating Vendor Models; How to spot a bad model
Monte Carlo Analysis or AHDL C code Modeling
Additional Example Simulations if time permits

New Library Powers SMPS Designers

Intusoft has introduced a new SPICE model library for power supply designers. It is the first SPICE library to contain a comprehensive set of nonlinear switching level models for popular Pulse Width Modulation (PWM) ICs. The new library represents a major breakthrough for SMPS designers.

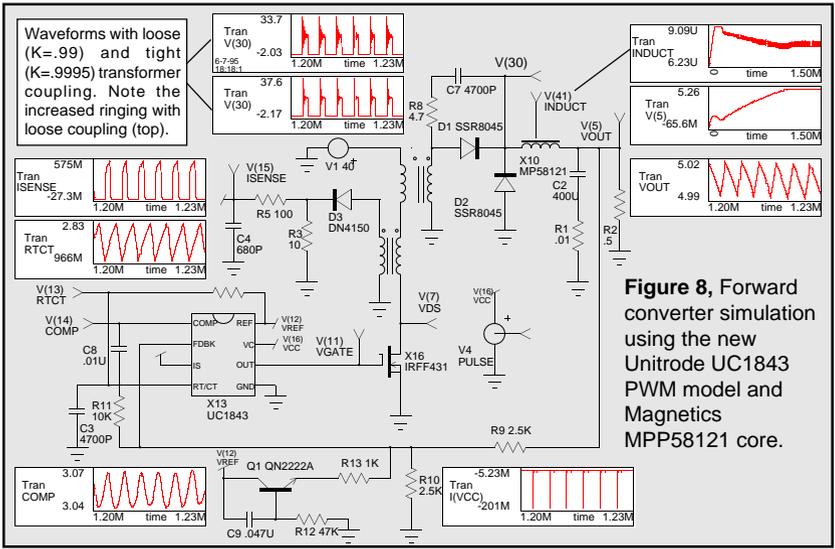
Previous models were based almost exclusively on state space average models which don't show many important nonlinear effects such as propagation delay, switching speed, component stress, sampling effects, and the total performance capability of the IC. The library does include a unique "unified" state space PWM model which works for AC, DC, and transient analyses in both continuous and discontinuous modes. However, the switching level models give designers a capability they have never had before; the ability to plug in a model, representative of the actual IC, and simulate the switching performance accurately and realistically. The library has over 400 models including:

- **Over 20 voltage and current-mode PWM IC models with full switching accuracy; Unitrode (UC1842/3/4/5, UC1524/5/7 and UC1823/5), Linear Technology (LT1242/3/4/5), Siliconix (Si9145), and Cherry Semiconductor (CS322/324).**
- **'Unified' state space model for forward, buck, boost, and flyback converter topologies and for several PWM ICs**
- **Nonlinear Magnetic Cores; Magnetics MPP cores, F material, R material, and several European cores**
- **Transformers; a variety of structures (pulse, auto, multiple windings, multiple cores)**
- **Voltage regulators; TL431, LM136**
- **Mosfet/IGBT Drivers; Teledyne TC426/7/8, Intl. Rectifier IR2110**
- **Motor controller IC; UC1637**
- **Power factor correction IC; UC1854**

Some PWM IC models have multiple levels of complexity. This allows the engineer to select the appropriate model features of interest and thus control the simulation efficiency. The models utilize analog behavioral elements and special Boolean logic elements unique to Intusoft's IsSPICE4. Together they greatly reduce the model's simulation runtime, an important factor in SMPS analysis.

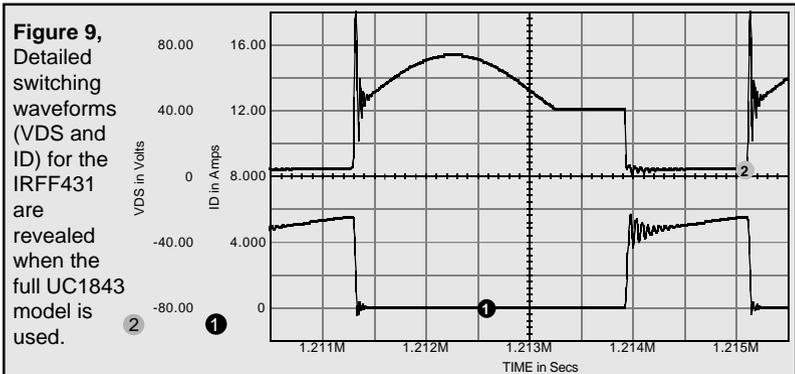
The price of the Power Supply Designer's Library is \$395.

The library will be available JUNE 26, 1995. The models are compatible with IsSPICE3 and IsSPICE4, as well as 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.



Forward Converter Using the UC1843

Figure 8 shows a forward converter example using the Unitrode UC1843 and Magnetics MPP58121 nonlinear core models from the new library. The PWM can be simulated with or without the output drivers fully modeled. The IRFF431 can be replaced with a non-ideal switch. These changes provide useful results (see VOUT & V(41), 0 -1.5Ms) in a simulation time of 208s on a Digital Alpha 190. With the full models in place we can further examine the operating current into VCC, under voltage lockout threshold, the propagation delay through the PWM, or in this case, the detailed Mosfet performance (Fig. 9) and drive capability of the PWM. These are the types of quantities that can't be measured with state space models. The steady state switching waveforms for the PWM are shown above along with the VOUT ripple (20mVp-p). Note the test point in the MPP model showing the inductance variation during the simulation.



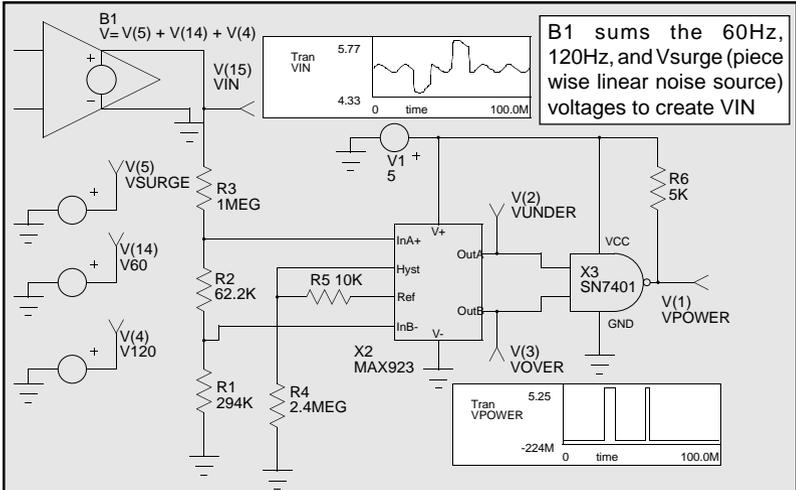


Figure 11, Maxim's new release includes over 24 new op-amp and comparator models. The MAX923 ultra-low power dual comparator (with hysteresis and a built in voltage reference) is used here to create a voltage window detector.

Several typographical errors were noted in the original model [1], as well as cases where the model did not match the vendors' block diagram. Intusoft has corrected these errors and tested the models. The corrected subcircuit netlist for the timer model is shown in Figure 10 along with a typical astable simulation. If, in the course of running the models as listed in [1], you encounter any problems, please feel free to contact Intusoft technical support. You can also get a copy of the June newsletter floppy which has the tested listings.

New Maxim Op-amps and Comparators

Maxim has provided Intusoft with an addition to their SPICE model offerings. Models are included for op-amps (MAX427 , 473, 478, and 492 families), transconductance amps (MAX435 family) and comparators (MAX921, 922, 923, 924, and 941 families). We chose the MAX923 (low power, low cost comparator) for a simple but illustrative example. Figure 11 shows an undervoltage/overvoltage window detector with its Power OK, under, and over voltage outputs. The design procedure is outlined in [2]. Of particular simulation interest is how the VIN waveform was constructed. VIN is the summation of 3 separate sources. The summing is performed by the IS_{SPICE4} B element (B1), which allows mathematical expressions containing voltages and currents from anywhere in the circuit. Many kinds of stimulus waveforms can easily be created in this manner [3].

[1] Standard Linear Circuits Macromodels, SGS-Thomson Microelectronics, 6/94

[2] Maxim 1995 New Releases Data Book Vol. IV pg. 3-57"

[3] Stimulating Circuits: Test Generators, Intusoft Newsletter, November 1994.

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

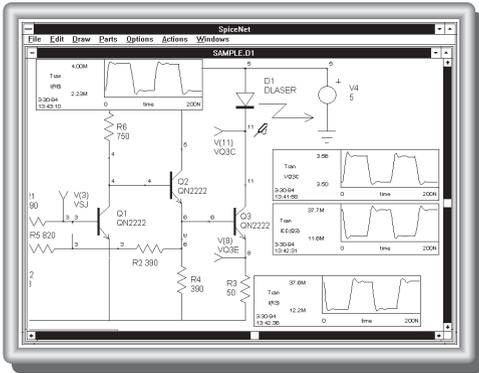
With ICAP/4 you can cut through the toughest circuit design problems with ease, create better products with more functions and higher yields, and explore new concepts. From power to RF to mixed mode, ICAPS allows you to analyze and predict the performance of all types of circuits.

Intusoft has been a leader in full featured design tools since our first product, IS_{SPICE}, 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 IS_{SPICE}4. 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

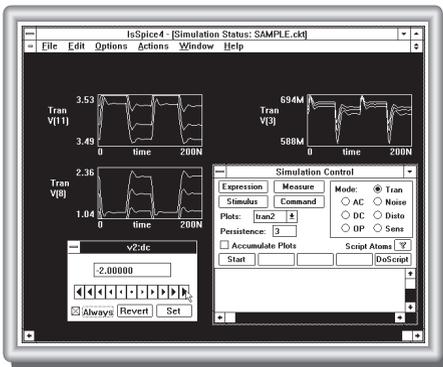


down menus, you can input and simulate your first design in less than an hour. SPICE4 has a direct interface to IS-SPICE4 allowing waveforms to be interactively cross-probed directly on the schematic. Additionally, the circuit operating point voltages can be updated as component values are changed.

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 Macintoshes. 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

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

Adding C Subroutines To IsSPICE4*

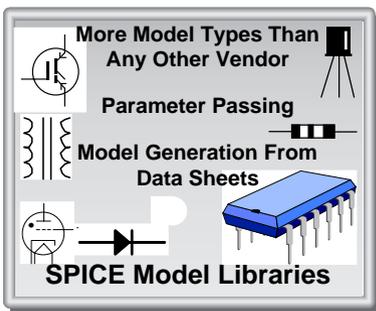
The Intusoft Code Modeling Software Development Kit (CMSDK) allows you to add your own C code subroutines to IsSPICE4. The C code is the basis for XDL, a new HDL (Hi-level Description Language). XDL models are like traditional SPICE models, except they are created by you. The CMSDK is required for model 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 **6000 models**.

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.

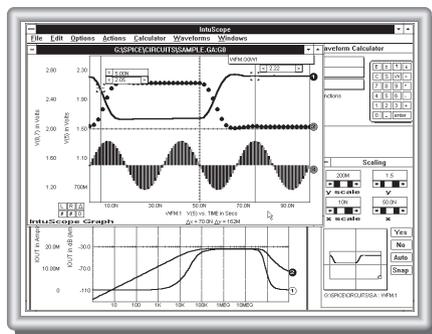


Model Generation From Data Sheets: A separate program (SPICEMOD 2.0) is available for creating SPICE models from data sheet parameters.

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 200MHz. 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

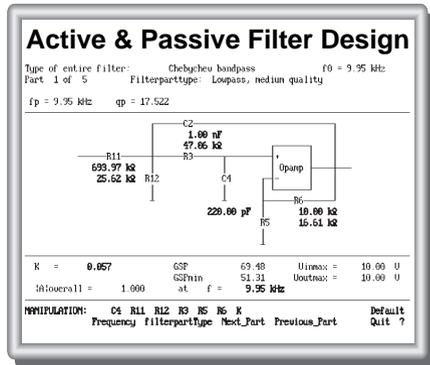
"SPICE APPLICATIONS HANDBOOK, 2nd Edition" - Collections of past Intusoft Newsletters, 6/86 - 2/94 (34 in all!).

"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

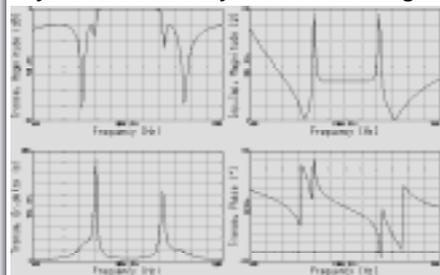
Active/Passive Filter Design

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.



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).