

Intusoft Newsletter

Personal Computer Circuit Design Tools

October 1994 Issue



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

New AHDL Based On 'C'

SPICE simulators of today are clearly NOT the same as they were 10 years ago. Due to improved development tools and the change from Fortran to C, SPICE developers have been able to extend Berkeley SPICE by adding new models, analyses, and graphical user interfaces. Beginning with a release this December, Intusoft will offer a true Analog Hardware Description Language (AHDL) that allows users to develop SPICE models (code models) using C. This release incorporates extensions of Georgia Tech's XSPICE integrated with our IsSPICE4 simulator. This will include an integrated event driven Digital Mixed Mode simulator and more than double the number of SPICE

In This Issue

- 2 New 'C' AHDL
- 8 New 'C' Code Models
Speeds 4046 PLL Sims
- 9 3Ø IGBT Inverter
- 11 ICAP/4Lite Upgrades
New Compuserve
Section for Intusoft
Power Mac SPICE
- 12 The Modeling Corner:
AD & Comlinear Parts
- 12 CP Clare Optocoupler
- 14 Simulating In The 90's

continued on page 2



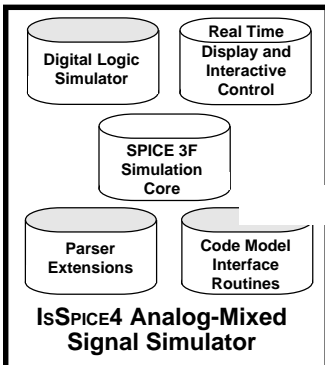
C Code for Model
+ Interface File
Specification

Debugging Information

Accessor
Macros and
Functions

Tools
Compiler, Makefile
Examples

Code Model Development Kit



Windows NT
Visual C++

User Defined
Code Models

Runtime link

Code Models
Supplied with
IsSPICE4

Figure 1, New techniques allow end users to easily add C code subroutines to IsSPICE4. Items with shaded tops are new components.

New AHDL Based On 'C'

*continued
from page
1*

primitives. Many important analog models will be added. In the following months, we will make the AHDL available in a "Code Model" Development kit. Our implementation of this C code based modeling technique uses DLLs that can be accessed with an options line in the SPICE netlist. This allows the simulator itself to be customized based on the needs of a given simulation and allows users to exchange code models. The update for the December release will be priced at \$295. There will be no change in the ICAP/4Windows price. Those buying ICAP/4Windows after October 9 will receive a free update certificate.

This major breakthrough uses a C language code generator to convert an easy to read text file (Interface File Specification) into the data structures required by `IsSPICE4`. The interface file is a table you prepare that describes the model ports and parameters. The code generator then produces the necessary data structures and initializes them based on your table entries. All you see is a model template that will use C macros to access the SPICE data structures. The task of describing the electrical behavior (code model body) is greatly simplified by supplied macros and functions that handle the details of interfacing a model to SPICE. The complexities of SPICE are hidden; the XSPICE additions do most of the work for you.

The code modeling methodology removes the need for you to learn the internal workings of the SPICE simulator. Built into the XSPICE additions are the procedures for reading the input netlist, parsing it into the model and instance data structures, setting up the matrix entries, etc. The result is an incredibly simple interface that reduces the effort in making a SPICE model from months to days.

Our first release will include:

- An event driven logic simulator, inside `IsSPICE4`, providing native mixed mode simulation. A full set of 21 NEW digital primitives (logic gates, flip-flops, RAM, State Machine, etc.) with access to timing information.
- Over 15 NEW analog and hybrid (mixed analog/digital) primitives created with the Code Modeling toolkit.

In the 1st quarter of 1995, Intusoft will introduce a Code Model Development kit consisting of a compiler and other miscellaneous tools (Described below). This C code modeling

Modeling Methods Available In IsSPICE4

SPICE 2G	SPICE 3	Code Models
Passive Elements	Math Equations	More Analog Primitives
Semiconductors	If-Then-Else	Digital Primitives
Polynomials Equations	Table Models	C Code Subroutines
	Boolean Expressions	Laplace Equations
	Switches/Lossy Lines	

A primitive is a basic SPICE building block. It is described by a single statement and, in some cases, a .MODEL statement. For example:

Diode - D1 1 0 Dmod *plus* .Model Dmod D (Is=10P N=1.75)

Boolean Expressions B1 1 0 V= ~ (V(1) & V(2))

Mathematical Expressions - B0 1 0 V= V(1)* SIN(V(2)) + ABS(V(3,4))

Procedural Function (If-Then-Else) - B3 2 0 V=V(CTRL) < 0 ? V(LO) : V(HI)

Digital - A4 [1 2] 3 Nand *plus* .Model Nand D_nand (Rise_Delay=10n)

Table 1, IsSPICE4 supports a variety of procedural, behavioral, digital and analog constructs for modeling.

capability completes the AHDL picture for IsSPICE4 giving it all the features and portability of even the most powerful AHDL languages.

Note: The toolkit is supported for Microsoft Visual C++ 1.1 under Windows NT. This setup was chosen because it is the only current viable 32-bit development alternative under Windows. Although code models must be developed under NT, the resulting DLL can be used with IsSPICE4 running under Windows 3.x on any compatible platform.

In future newsletters, we will explore the new analog and digital primitives created using the C code modeling approach. Here we discuss the foundation; C Code model development.

Developing C Based SPICE Models

Often, the designer has a very clear idea of how a particular behavior can be modeled in a programming language like "C", but has no idea how to go about adding a new device model to SPICE. Those determined enough to examine the SPICE source code quickly come to the conclusion that the effort involved does not justify the potential benefits. Macro-modeling, on the other hand, may not provide the required model flexibility or speed.

The new Intusoft Code Modeling kit provides an easy way for an engineer to add C code based subroutines to SPICE without having to deal with the details of interfacing with the SPICE software and data structures. What would have taken MONTHS to develop and integrate with SPICE can now be done in a matter of DAYS.

Figure 2 (Interface File Specification), The Port_Table describes the model's nodal connections. The Parameter_Table describes the parameters that will be available in the code model's .MODEL statement:

NAME_TABLE:

C_Function_Name: cm_gain
 Spice_Model_Name: gain
 Description: "A simple gain block"

PORT_TABLE:

Port_Name:	in	out
Description:	"input"	"output"
Direction:	in	out
Default_Type:	v	v
Allowed_Types:	[v,vd,i,id,vnam]	[v,vd,i,id]
Vector:	no	no
Vector_Bounds:	-	-
Null_Allowed:	no	no

Types of signals that can be connected to this element (voltage, current, digital, etc.)

PARAMETER_TABLE:

Parameter_Name:	in_offset	gain	out_offset
Description:	"input offset"	"gain"	"output offset"
Data_Type:	real	real	real
Default_Value:	0.0	1.0	0.0
Limits:	-	-	-
Vector:	no	no	no
Vector_Bounds:	-	-	-
Null_Allowed:	yes	yes	yes

Figure 3 (Code Model Body):

```
void cm_gain(ARGS) /* structure holding parms, inputs, outputs, etc. */
{
    Mif_Complex_t ac_gain;

    if(ANALYSIS != MIF_AC) {
        OUTPUT(out) = PARAM(out_offset) + PARAM(gain) *
            ( INPUT(in) + PARAM(in_offset));
        PARTIAL(out,in) = PARAM(gain);
    }
    else {
        ac_gain.real = PARAM(gain);
        ac_gain.imag = 0.0;
        AC_GAIN(out,in) = ac_gain;
    }
}
```

Code Models are SPICE primitives, like BJT's or resistors. They can be written in a matter of days in contrast to the months required for making new SPICE models.

Figure 4 (Sample IsSPICE4 Netlist):

```
A SIMPLE AMPLIFIER CIRCUIT
.OPTIONS CML=GAIN.DLL ; Calls user created Code Model DLL
.TRAN 1E-5 2E-3
VIN 1 0 0.0 AC 1.0 SIN(0 1 1K)
CCOUPLE 1 IN 10UF
RZIN IN 0 19.35K
AAMP IN COLL GAIN_BLOCK ; New IsSPICE4 primitive gain element
.MODEL GAIN_BLOCK GAIN (GAIN = -3.9 OUT_OFFSET = 7.003) ; New .Model
RZOUT OUT COLL 3.9K
RBIG COLL 0 1E12
.PRINT TRAN V(1) IN COLL OUT
.END
```

The benefits of developing models using C code have not been widely recognized. This is mainly due to the limited programming and simulator support and extensive knowledge of SPICE that is required. The Code Model toolkit alleviates these difficulties. It assists the user in writing, compiling, linking, and using C code subroutines with `IsSPICE4`. The toolkit works with additions to the simulator core to tell the simulator how to parse the extended circuit description and model statements and how to call C code that defines the model's behavior (Figure 1). The code modeling architecture is based on the XSPICE program distributed by the Georgia Tech Research Corp., a division of the Georgia Institute of Technology [3].

Creation of a new model requires 4 steps:

- Creation of the directories to hold the appropriate files
- Definition of the model interface specification (Figure 2)
- Definition of the C subroutine to describe the model's behavior (Figure 3)
- Building and linking of the model

The interface file specification describes the model's connections and parameters. For developing the code model body, the toolkit provides a comprehensive set of macros and functions to simplify the model development process and to insulate developers from the underlying SPICE algorithms and data structures. For example, macros to access information in the simulator (voltages, parameter values) and functions for smoothing, model state storage functions, integration and convergence message handling, breakpoint handling, complex math and more are provided.

The developer uses a makefile to compile and link the code model. Compiler and linker diagnostics aid in the debugging process as they would for any application being developed under Microsoft Visual C++. The result of a successful build is a code model library (DLL) that can be accessed (dynamically linked) with the `IsSPICE4` executable at runtime. An optional command line switch causes `IsSPICE4` to pause after dynamically loading code model libraries. This allows the user to set breakpoints, if desired, in the code model. Once a breakpoint is encountered, Visual C++ provides the ability to examine variables, step through the execution, and otherwise support debugging.

To ease model development, the toolkit shields the developer from most of the simulator's internal operation. This alleviates problems that were once the hurdles to SPICE model development. Before the simulation `IsSPICE4` automatically:

- Parses the code model call lines in the `IsSPICE4` input netlist, verifies the right number of connections, checks the input parameters, etc.)

- Sets up the model data structures, including supplying default values for parameters not specified on the .MODEL line. Sets up the circuit description data structures for each code model instance.
- Creates equations in the matrix for voltage sources. Sets up the matrix pointers used during simulation to load the matrix.

During the simulation `IsSPICE4`:

- Iterates through all models and instances of a specified code model device type, fills in the inputs for the model, calls the model, and then uses the outputs and partials returned by the model to load the matrix.
- Monitors conversion and truncation errors in code models.

Once the user is satisfied that the model is performing correctly, the code model library can be copied for use on other workstations running `IsSPICE4`. These steps are easily learned and only require a few minutes to perform once the model description is written. Figure 4 shows a SPICE netlist that makes use of the sample gain code model.

Advantages of Code Modeling

Since the XSPICE interface for C code modeling is publicly available, all models written for it are transportable to any other simulator that supports the code model interface. Intusoft's version accesses the C code models via a Windows DLL. This alleviates the need to recompile the SPICE executable each time a new model is added. Addition of the DLL feature allows models to be easily distributed and copied by developers and users and copy protected by the programmer if required. Trading and/or buying code model libraries provides the means to greatly expand the capabilities of SPICE, even for those not interested in developing their own code models.

The C code modeling approach has additional advantages:

- Allows new primitives to be added to the simulator core more efficiently than any other direct coding method
- Allows arbitrarily complex functions and behavior to be added to SPICE
- Provides higher level primitives for better system level simulations
- Provides a full range of digital primitives, as well as ability to create custom digital elements
- Allows user-defined node/data types
- Primitives are added to SPICE via a Windows DLL; Users can add to the existing DLL or create separate DLLs

Code models support:

- Any number of inputs or outputs
- Scalar, vector, real, integer, ground reference, differential, digital (12-

- states), resistance, conductance, voltage and current port types
- Unlimited number of model parameters of type real, integer, complex, string, or boolean
- Model parameter defaults and constraint checking
- Model information and message output.

Some examples of the new IS_{SPICE4} analog code model primitives are:

- System blocks: Laplace equations, limiters w/smoothing, hysteresis, slew rate
- New Models: Magnetic core, zener diode, controlled oscillators, analog switch

In summary, code modeling extends IS_{SPICE4}'s capabilities to allow efficient simulation of mixed-signal, board, and systems-level circuits. It provides a practical alternative to macro-modeling and the widely scattered AHDL languages by allowing end-users to easily add new SPICE primitives. It uses the commonly available C language and shields the user from virtually all internal simulator related concerns through specially developed macros and functions.

[1] "Code-level Modeling in XSPICE", Fred. L. Cox III, William B. Kuhn, Jeffery Murray, and Stephen Tynor, Proceedings of the 1992 Intl. Symp. on C&S, San Diego, CA, 5/92, IEEE 0-7803-0593-0/92

[2] "SPICE as an AHDL", Charles Hymowitz, Penwell Publishing, Computer Design Mag. Analog & Mixed Signal Conference, July 1994

[3] XSPICE Users Manual, Georgia Research Corporation, Georgia Institute of Technology, Atlanta Georgia 30332-0800

Code Models Help Simulate 4046 PLL

Phase lock loops can consume a great deal of simulation time due to the many cycles required. Therefore, model efficiency is important. To illustrate the advantages of using code model elements, a PLL circuit similar to the 4046 was simulated.

Two cases were simulated for the circuit in Figure 5. One where the VCO was modeled using a SPICE 2 subcircuit, using 10 elements, and the second using a single VCO code model. Both performed about the same, but the code model provides superior flexibility with regards to the control voltage to output

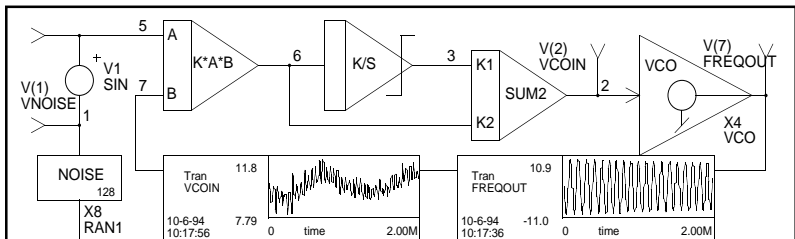
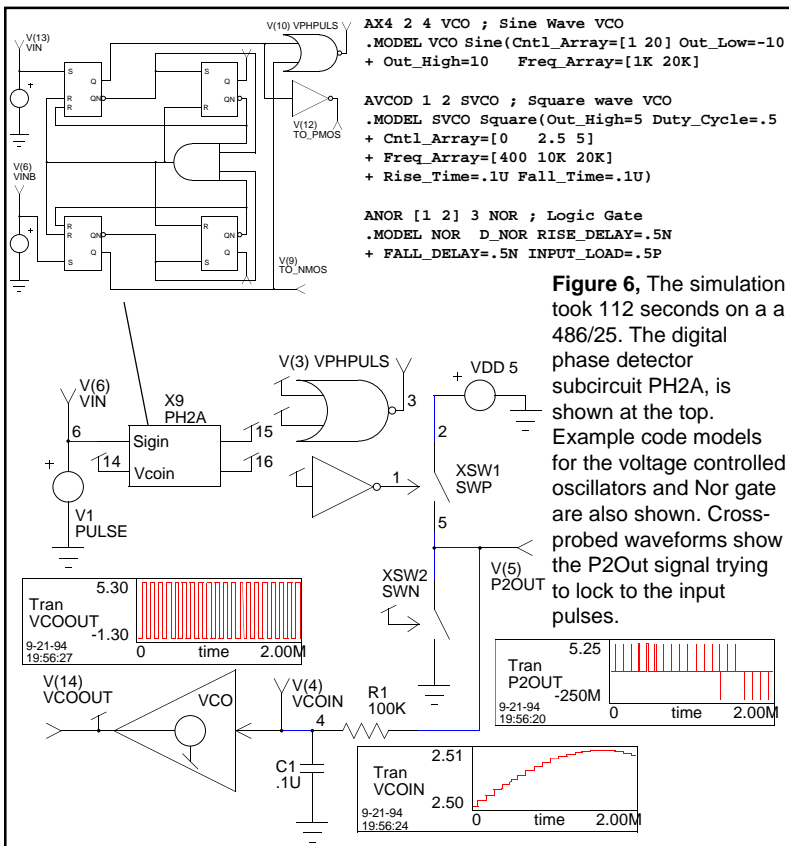


Figure 5, A phase lock loop simulation with noise. The blocks can use either SPICE 2 subcircuits or the new code models. While the simulation speed is comparable for analog code models, their added versatility makes them superior.

Code Models Help Simulate PLL

frequency transfer function. For example, the VCO code model in Figure 6 contains Cntl_array and Freq_array parameters. The input VCO voltage is used as an independent variable in a piecewise linear curve described by the array pairs. From the curve the output frequency value is determined. It is easy to see that array sizes of 2 for both arrays will yield a linear variation of Vin vs Freqout. More complex relationship can be very easily created using this table-type approach. This feature could be added to the SPICE 2 VCO but at the cost of even greater complexity.

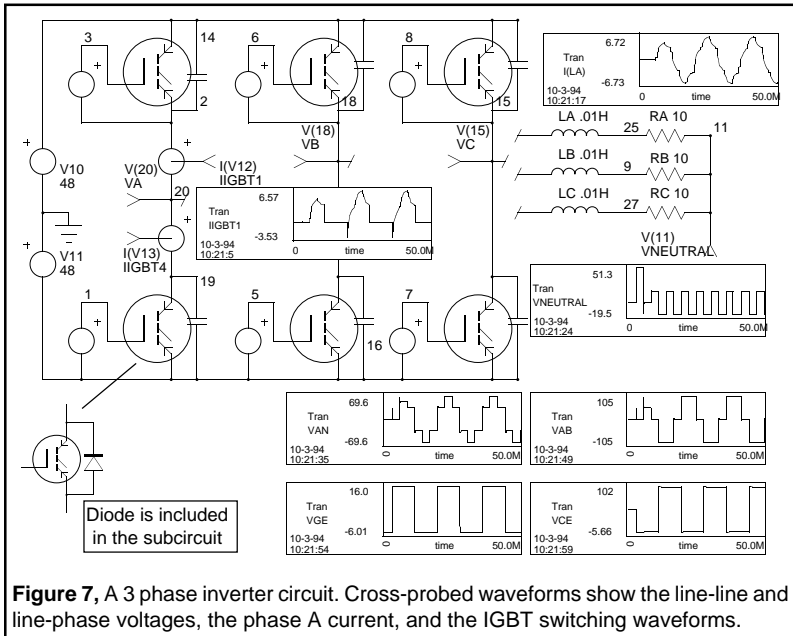
A digital phase detector, Figure 6, can be modeled using transistor, Boolean Logic expressions or by the new digital code model primitives. Since the digital primitives are simulated using a built-in event driven logic simulator, their simulation efficiency is much greater than any other modeling method. It simulation took less than 11 seconds on a Pentium/60.



IGBT Model Simulates 3 θ Inverter

The ICAP/4 package contains an advanced model for IGBT devices. To illustrate its use, a 3 phase inverter with simplified motor load was simulated (Figure 7). Unlike behavioral models used by other SPICE vendors the Intusoft IGBT exhibits more important effects such as accurate switching losses and nonlinear turn-on/turn-off. Behavioral modeling of power semiconductors is possible with IS_{SPICE4} and can greatly speed a simulation. In the case of IGBTs, however, behavioral modeling can hide some important information. The model, shown in Figure 8, is SPICE 2G.6 compatible and can be used on almost any SPICE simulator. It was made with the $SPICEMOD$ modeling program. $SPICEMOD$ allows users to create their own models from typically available data sheet parameters.

The model allows examination of both circuit and IGBT related design issues. For the inverter circuit, Figure 7 shows the line-line and line-neutral quantities, as well as the IGBT switching waveforms. In Figure 9, the effect of varying the load inductances (LA , LB , and LC) is displayed. The control circuitry has been simplified so as not to unnecessarily complicate the simulation. An anti-parallel diode, included in the actual Fuji IGBT module, is hidden in the IGBT subcircuit (DFW). Issues such as parallel IGBT operation, circuitry for overcurrent/short



circuit protection, and various snubber configurations can also be explored.

ICAP/4 libraries contain over 90 IGBT models from Harris, IXYS, Fuji, I.R. and others. Intusoft was the first vendor to introduce a Berkeley SPICE compatible model in 1991. Interested parties can download several sample IGBT models from Intusoft's CompuServe forum. **Simulation Speed Note:** On a Carrera Cobra 275 (275MHz Digital Alpha AXP PC) the 166 element inverter circuit runs in under **6 seconds!!**

```
.SUBCKT I6MBI30F 71 72 74; Connections: Collector Gate Emitter
M1 81 82 83 83 MFIN L=1U W=1U
Q1 83 81 85 QOUT
DSD 83 81 DO
DBE 85 81 DE
RC 85 71 26.2M
RE 83 73 2.62M
RG 72 82 40.1
CGE 82 83 2.83N
CGC 82 71 1P
EGD 91 0 82 81 1
VFB 93 0 0
FFB 82 81 VFB 1
CGD 92 93 1.2N
R1 92 0 1
D1 91 92 DLIM
DHV 94 93 DR
R2 91 94 1
D2 94 0 DLIM
LE 73 74 7.5N
DLV 94 95 DR 13
RLV 95 0 1
ESD 96 93 POLY(1) 83 81 19 1
MLV 95 96 93 93 SW
DFW 74 71 DMBI30 ; anti-parallel diode
.MODEL SW NMOS (LEVEL=3 VTO=0 KP=5)
.MODEL QOUT PNP (IS=3.24P NF=1.2 BF=5.1 CJE=3.21N
+ TF=75N XTB=1.3)
.MODEL MFIN NMOS (LEVEL=3 VMAX=400K THETA=53.3M
+ ETA=2M VTO=4.5 KP=1.61)
.MODEL DR D (IS=324F CJO=85.8P VJ=1 M=.82)
.MODEL DO D (IS=324F BV=600 CJO=2N VJ=1 M=.7)
.MODEL DE D (IS=324F BV=14.3 N=2)
.MODEL DLIM D (IS=100N)
.MODEL DMBI30 D (IS=34.5N RS=59.2M N=1.75 BV=600
+ IBV=1M CJO=1.23N VJ=.75 M=.333 TT=432N)
.ENDS
```

Figure 8, An IGBT subcircuit for the FUJI I6MBI30F 600V 30A 75ns device. The model was made with SPICEMod, a program that makes SPICE 2G.6 compatible models from data sheet parameters. All of the IGBTs in the inverter used this model.

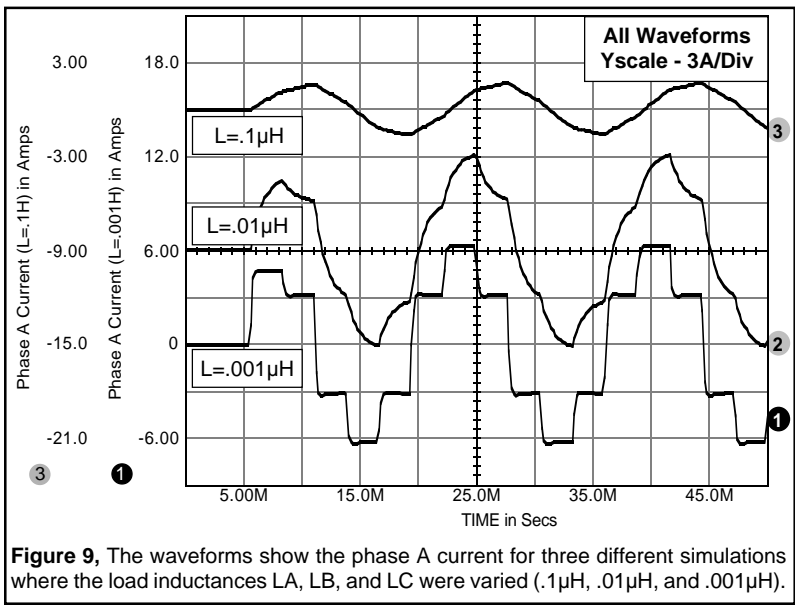


Figure 9, The waveforms show the phase A current for three different simulations where the load inductances LA, LB, and LC were varied (.1µH, .01µH, and .001µH).

ICAP/4Lite Upgrades A Hit

For the hundreds of you who bought the ICAP/4Lite program we thank you for making its launch so successful. ICAP/4Lite can be obtained through your local technical bookstore, your local dealer, or directly from Intusoft. All systems contain a \$150 discount coupon applicable towards any ICAP/4Lite upgrade. The coupon is good until 12/15/94. Upgrades are available for the schematic, model, simulator, or post processing portions of ICAP/4Lite.

The most popular upgrade so far is for the INTUSCOPE graphical post processor. INTUSCOPE allows you to fully analyze the ISpICE output using: **waveform cursors, advanced waveform math and measurements, powerful macro capability, and greatly enhanced scaling features.** The INTUSCOPE upgrade is the best way to improve the power and efficiency of your Lite simulation system.

For current Pspice® or ICAP/4 owners the new ICAP/4Lite system is priced at \$395, **a saving of \$200** over the list price of \$595. Working demonstration disks are available. Other competitive products of equal or greater value may also qualify; demo, student or evaluation versions do not. Contact Intusoft or your local dealer for more information. Offer good until 1/15/95.

New Compuserve Section For Intusoft

CompuServe has provided Intusoft with its own section under the CAD/CAM/CAE forum. Instead of being listed under the All CAD/CAM/CAE section (library #1), Intusoft will have its own library and message sections. This provides much easier access to all of Intusoft's postings and easier access to technical support. Type Go CADDVEN at any ! prompt to get to the CADD/CAM/CAE forum and look for Intusoft.

New ICAP For Power Macs Being Readied

A native Power Macintosh version of ICAP/4 will be available in the 4th quarter. The new version will run on any Power Macintosh computer. Currently, all 68xxx systems are supported including the Powerbook series and other Macintosh computers that do not have a coprocessor. At present, Power Mac users can take advantage of ICAP/4MNC, the no coprocessor version of ICAP/4. Even though the coprocessor is emulated, ICAP/4MNC on a Power Macintosh still runs as fast as the coprocessor version does on a Macintosh IIci. ICAP/4MNC (For Macintosh Powerbooks and Power Macs) is available NOW for \$1575.

Pspice is a registered trademark of Microsim Corp.

The Intusoft Modeling Corner

In this issue of The Intusoft Modeling Corner we bring you an application note using the dual opto-isolator model from C.P. Clare.

In addition to several opto-isolator models, over 55 new op-amp models from Analog Devices and Comlinear are also included on the *Intusoft Newsletter* disk for subscribers. The new op-amps include current and voltage feedback op-amps such as the CLC402, CLC426, and OP113, buffers such as the CLC109 and CLC111, voltage references such as the AD588 and AD680, and instrumentation amps such as the AMP04.

The *Intusoft Newsletter* floppy disk is available on a subscription basis and includes models supplied to Intusoft by various hardware vendors, models associated with the articles in the newsletter and models created by Intusoft Tech. Support staff.

Simulating A Dual Opto-Coupler

How do we select candidates for new models? Most of the input comes from our customers. In this issue, we present a model requested by ICAP/4 users at C. P. Clare Corp.[1] They asked Intusoft to provide a simulation model, which their customers could also use, for a new part - the LOC110 dual output, linear opto-isolator. Intusoft is the only SPICE vendor to create models for its customers FREE of charge.

The LOC110 features dual phototransistor outputs, one of which is a monitor that is used in a feedback loop to compensate for any non-linearity or temperature effects in the primary output. The feedback allows DC, as well as AC signals, to be transmitted with isolation. Some typical uses of this part are in power supplies, medical sensors, and telephone or modem circuitry.

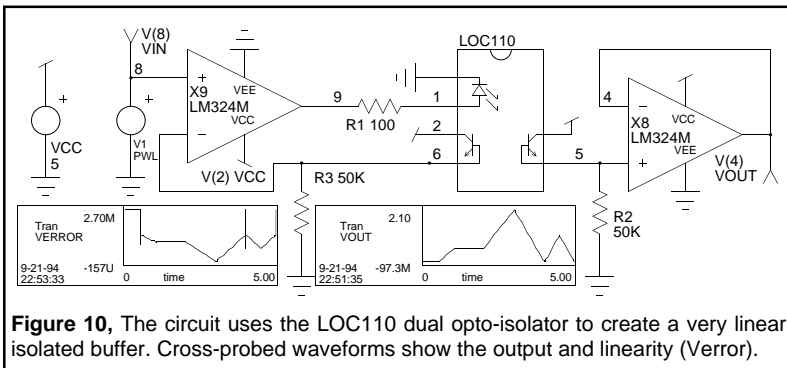


Figure 10, The circuit uses the LOC110 dual opto-isolator to create a very linear isolated buffer. Cross-probed waveforms show the output and linearity (Verror).

```

.SUBCKT LOC110 1 2 3 4 5 6
D1 13 1 DOPTO
.MODEL DOPTO D(N=2 RS=.7 CJO=40P IS=2.5E-12)
G1 6 5 10 0 .007
G2 3 4 8 0 .007
H1 14 0 V1 1
H2 15 0 V1 1
R4 14 10 1K
R5 15 8 1K
C1 8 0 500P
C2 10 0 500P
V1 2 13 0
.ENDS

```

Figure 11 The IsSPICE subcircuit for the LOC110 dual opto-isolator made by C.P. Clare.

The input current drives a light emitting diode (LED), which is optically coupled to the two base-collector transistor junctions used as photodiodes. The output photodiode current is typically converted into a voltage with a resistor and then buffered to provide the required drive current. The second photodiode monitors the LED and provides feedback to control the LED drive current. A typical application is shown in Figure 10 using an inexpensive LM324 single-supply op-amp for the output buffer and servo amplifier functions. The output current is converted into a voltage through the 50k Ω resistor and then buffered by the op-amp follower. The other photodiode current is converted to a voltage through another 50k Ω resistor. This latter voltage is compared with the input voltage by use of another LM324, with negative feedback assuring the voltages track. Figure 10 shows the simulated performance of the LOC110 for an arbitrary input waveform and the error (calculated by subtracting the output voltage from the input voltage). Note: this error calculation includes any errors from the op-amp.

The macro model for the LOC110, shown above, contains diode D1 at the input. The current through this diode is monitored by voltage source V1. This current is converted to a pair of voltages by the current controlled voltage sources H1 and H2. The frequency response of the opto-isolator is set by two RC networks, R4/C2 and R5/C1. The output of the R/C networks is then converted into the final output currents by a pair of voltage controlled current sources, G1 and G2. The typical current gain of 0.07 (ratio of diode current to output current) is set by the gain coefficients of G1 and G2. The model for the LOC110 is included on the *Intusoft Newsletter* floppy disk for subscribers.

[1] C. P. Clare Corp., Solid State Products Division, 8 Corporate Place, 107 Audubon Road, Wakefield, MA 01880.
Tel: (617) 246-4000, FAX: (617) 246-1356

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
- **PRESPICE** Model Libraries/Netlist Editing
- **ISSPICE** SPICE Simulation/Advanced Analyses
- **INTUSCOPE** Data Processing and Analysis

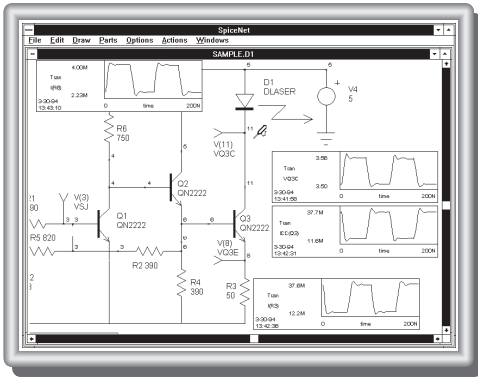
With ICAP/4 you can cut through the toughest circuit design problems with ease, create better products with more functions and higher yields, and explore new concepts. From power to RF to mixed mode, ICAPS allows you to analyze and predict the performance of all types of circuits.

Intusoft has been a leader in full featured design tools since our first product, ISSPICE, was released over eight 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

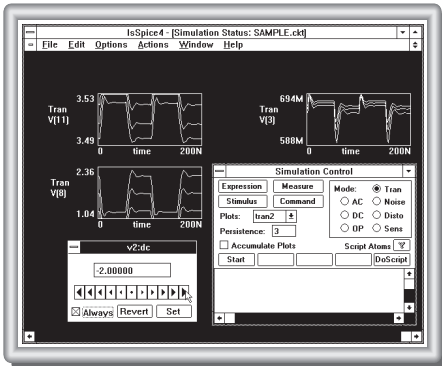


down menus, you can input and simulate your first design in less than an hour. SPICE4 has a direct interface to IS-SPICE4 allowing waveforms to be interactively cross-probed directly on the schematic. Additionally, the circuit operating point voltages can be updated as component values are changed.

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 and Macintoshes. It supports unlimited size circuits, waveform cross-probing, real time waveform display, *Simulation Scripts and Breakpoints*, and mixed mode simulation.
- IsSPICE3 provides the same analysis, model, and real time waveform support as IsSPICE4 except that it runs on DOS/Macintosh systems and is not interactive (SPICE2 batch style).
- IsSPICE runs on any PC under DOS or Macintosh. Circuit size is limited to about 200 components. It is based on SPICE 2G.6. IsSPICE3/4 for the PC do **NOT** require a coprocessor, but one is strongly recommended. All other versions require a coprocessor.

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

Additional IsSPICE3/IsSPICE4 Features

- Real-time waveform display of voltages, currents and device parameters
- GaAs Mesfets, MOSFETs Levels 4, 5, and 6, Lossy T-Lines, voltage/current ctrl'd Switches, and Boolean logic expressions. 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.

Interactive IsSPICE4 Features

- Sweep parameters one at a time or in groups
- Add or delete waveforms to the real-time display
- Interactively run any analyses without having to restart the simulator
- Start, Stop, Pause, Change, or Resume any analysis on demand

Compatibility

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

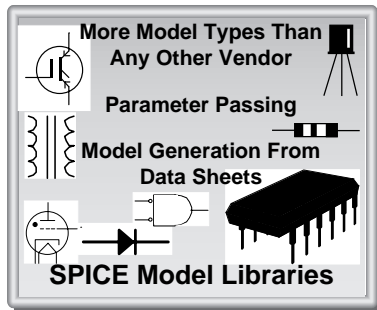
Advanced Analyses (All ICAP packages)

- Monte Carlo Analysis: Statistical yield analysis of circuit performance
 - Randomly vary circuit parameters to test performance
- Circuit Optimization: Automatically find a parameter value to maximize a user defined objective function
- Any circuit performance criteria may be measured including curve families

SPICE Model Libraries

Description: ICAP/4Windows and Macintosh include an extensive array of over **4000 SPICE models**.

Benefits: The SPICE Device Libraries contain a wide variety of models including diodes, zeners, BJTs, Darlingtons, op-amps, comparators, transformers, non-linear magnetics, JFETs, SCRs, IGBTs, Triacs, power MOSFETs, PWMs, SC filters, analog behavioral models, digital logic gates, switches, opto-isolators,



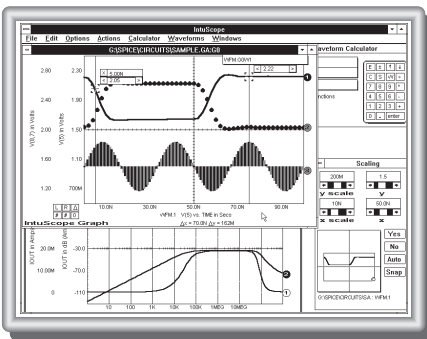
transmission line models, crystals, vacuum tube models and more. Over 100 "Generic Template" models that convert data sheet parameters into SPICE parameters are also included. Models are stored in ASCII text files that can be viewed and edited. A complete list of models is available.

Note: The vendor supplied IC libraries (over 1150 models) and the RF Device Library (over 300 models) are available separately.

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

SPICEMOD: SPICE Model Generation

Description: SPICEMOD is a DOS based program that creates SPICE models from data sheet values. It makes models for Diodes, Zeners, BJTs, Darlington BJTs, JFETs, MOSFETs, Power MOSFETs/BJTs, IGBTs, and SCRs. Although not part of the ICAPS package, SPICEMOD is a powerful addition to the models in PRESPICE.

Benefits: SPICEMOD is particularly useful in the circuit design phase because it allows engineers to create SPICE models based on electrical specifications before an actual device is selected. It is also a must for engineers that may have to generate their own SPICE models at a moments notice.



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

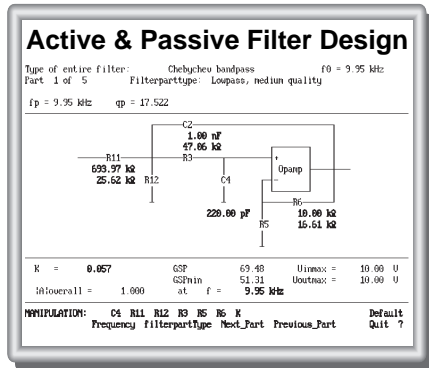
NEW "SPICE APPLICATIONS HANDBOOK, 2nd Edition" - Collections of past Intusoft Newsletters, 6/86 - 2/94 (34 in all!).

"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

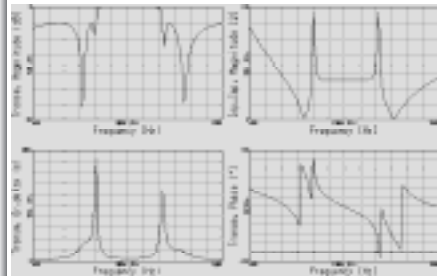
Active/Passive Filter Design

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).