

Intusoft Newsletter

Personal Computer Circuit & System Design Tools



Copyright © Intusoft, All Rights Reserved

Issue #47 Aug. 1996

Tel. (310) 833-0710

Fax (310) 833-9658

System-Level Simulation

The SPICE engine is the heart of any simulation system. However, the key that unlocks the power of the SPICE engine are the models that accompany the simulator. The results, and the effort required to get them, rely on both components. On the modeling front Intusoft is clearly the leader (see page 16) with more modeling capabilities than all its competitors. Here, we extend our lead with the introduction of several new AHDL models that can empower your simulations.

Spice Array Linking Technology, or **SALT**, is a modeling interface that links huge data sets to the

Continued on pg. 6

In This Issue

2 New Analog Simulation Tools For OrCAD, Protel, and Viewlogic

6 New AHDL Models Extend IsSpice; Repeatable PWLs FFTs, Convolution Filter Your Own Algorithms SOI Mosfet, Switch

10 ICL Powers SPICE To New Heights

14 New H-P Opto Models

15 New Magnetic Designer

ICAP for OrCAD, Protel & Viewlogic

Continued on pg. 2

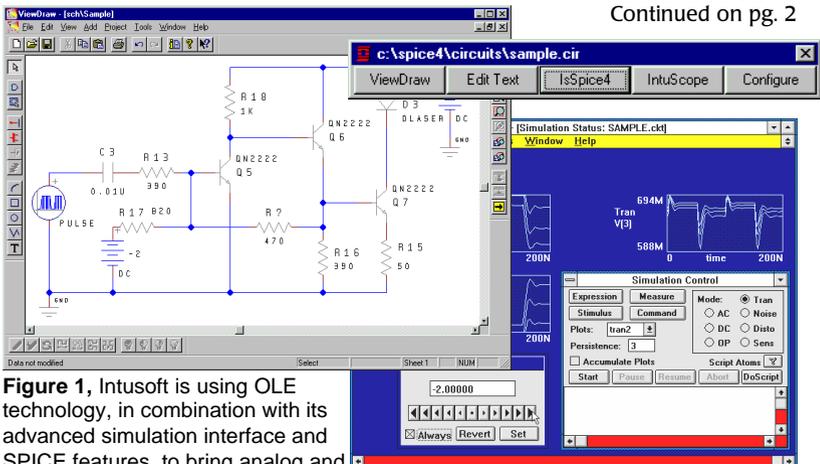


Figure 1, Intusoft is using OLE technology, in combination with its advanced simulation interface and SPICE features, to bring analog and mixed signal simulation to your schematic entry tool. Check Inside For More Details.

Intusoft Links with OrCAD, Protel, and Viewlogic

OLE Technology Used to Integrate Analog Simulation with Schematic Entry

Presently, the most common schematic/SPICE interface is text based. This is clearly insufficient from the integration and interaction points of view. In addition, most schematics do not fully support all of IsSpice4's advanced features. Consequently, analog simulation may be difficult to perform when using a third party schematic tool. What engineer's need is for their analog and mixed signal simulation tools to be as tightly integrated with their schematic entry program as their layout tools. Intusoft is proud to announce that the wait for seamless integration is over!

Intusoft is introducing a new series of products called **ICAP™** for OrCAD®, **ICAP™** for Protel®, and **ICAP™** for Viewlogic®. The new systems use Object Linking and Embedding (OLE) technology to transparently integrate analog and mixed signal capabilities into the OrCAD Capture™, Protel Schematic^{3™}, and Viewlogic Workview Office™ schematic tools. **ICAP™** includes the **IsSpice4** native mixed mode (analog/digital) circuit simulator, **SPICE Model Libraries** with over 8000 analog and digital parts, the **IntuScope** graphical waveform analyzer, the **SpiceMod** SPICE modeling program, a set of symbol libraries, and a new "magical" component, called the **Intusoft Integration Module**. The Integration module attaches a host of features to your schematic; features that make simulation easier to perform than ever. It is this new technology that allows

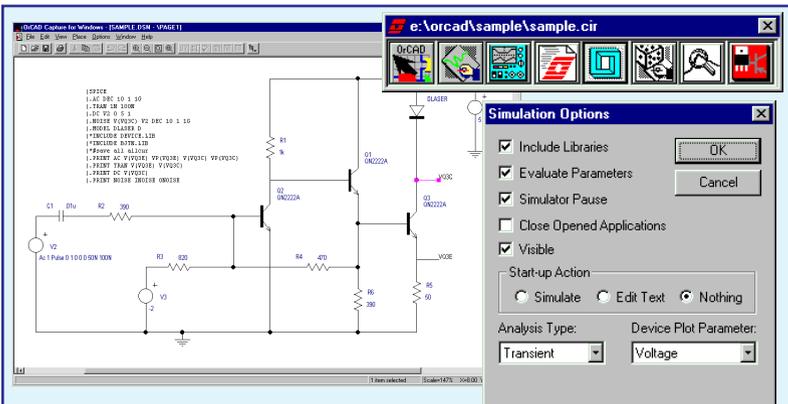


Figure 2, The ICAP™ Toolbar (top right) is customized for each schematic entry package (Capture shown above). It provides schematic users with easy access to model editing, SPICE simulation, cross-probing options, and real time waveform processing. Special menu functions have also been added to ease various simulation tasks (right).

- Accessories
- Options
- Win
- Run IsEd
- Run IntuScope
- Monte Carlo Options
- Run SpiceMod
- Simulation Options
- Optimization Options
- Run IsSpice4

OrCAD, Capture, Protel, Schematic³, Viewlogic, and Workview Office are registered trademarks/trademarks of their respective companies.

you to run SPICE simulations directly from your schematic. No netlist editing and virtually no knowledge of arcane SPICE syntax is needed anymore.

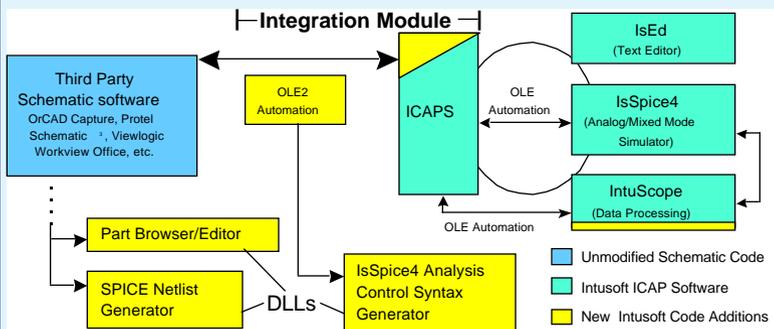
How It Works

Many of you may know that Intusoft has had its own schematic program for SPICE, SpiceNet, for over 9 years. In that time we have learned what schematic features are needed to make simulation easier. The Integration Module adds these features to your schematic tool. For example, SPICE model editing, custom SPICE attribute dialogs for each element, special dialogs for SPICE analysis statements, Monte Carlo analysis, waveform and operating point cross-probing from both the schematic and PCB layout, and the ability to launch and control the simulator from the schematic (Table 1) are provided.

In the case of Schematic³, **ICAP™** uses Protel's EDA Client/Server technology to take advantage of many of Schematic³'s features. **ICAP™** for Protel® contains the first simulation server ever available for Protel's EDA Client/Server architecture.

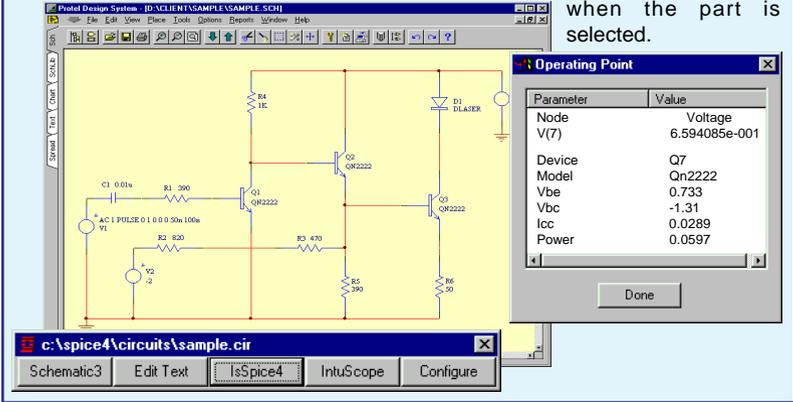
The OLE interface paradigm allows Intusoft to add capabilities independently of the schematic software. This means that you can keep up with the latest SPICE technology without needing to update your schematic entry programs. The result is that **ICAP™** makes Capture, Schematic³, and Workview Office vastly easier to use for analog/mixed signal simulation, and much more tightly coupled to SPICE.

Figure 3, The **ICAP™** system consists of the **IsSpice4** simulator, the **IntuScope** graphical data analysis tool, **SpiceMod**, **SPICE model libraries**, and the **Intusoft Integration Module**. The Integration Module is composed of the ICAPS (Interactive Circuit Analysis Program Selector) toolbar and an OLE module that hooks ICAPS into one of several third party schematics.



Several DLLs that enhance the schematic's ability to access SPICE attributes and other advanced simulation features are hooked into the schematic transparently using OLE technology.

Figure 4, You can select different kinds of information to cross-probe. Here, in Protel's Schematic³, a dialog displays each part's operating point information when the part is selected.



IsSpice4 is run by either creating a SPICE netlist or by selecting the "Run IsSpice4" menu function, depending on your schematic. When a simulation is run, the **ICAP™** Toolbar appears and IsSpice4 displays the simulated waveforms in real time (Figure 1). The communication is outlined in Figure 3.

The Toolbar gives you quick access to all simulation tasks such as netlist editing, simulation options, and data processing. The simulation options dialog, shown in Figure 2, controls the types of analysis data (AC, DC, Operating Point or Transient) and the types of waveforms (voltage current, or power dissipation) that you can cross-probe. If you click on a node or part in the schematic, you can see the operating point information (Figure

Figure 5, The IntuSoft Integration Module allows you to click on any node or device in your schematic. The corresponding voltages, device currents, or device power dissipations will be immediately displayed in IntuScope.

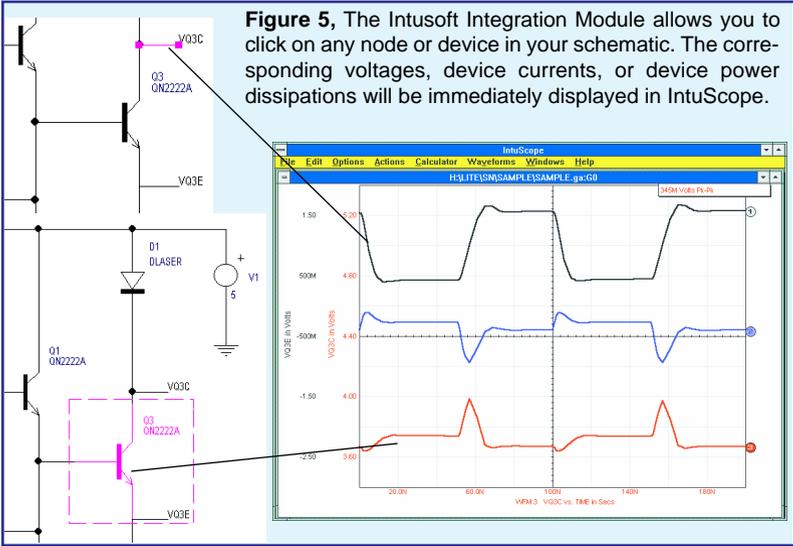


Table 1, ICAP Benefits for OrCAD, Protel, and Viewlogic Users

Feature	Benefit/Description
Launch a simulation directly from the schematic	Easily run a simulation (with a single menu function) without learning a new program interface
Cross-probe simulation results from schematic or layout	Quickly display the correct circuit waveforms; Display voltages, currents, power, and operating point info by clicking on a node or schematic symbol
Symbol Library Support	Symbols are provided for all SPICE models
Support for ALL SPICE Analysis statements and part attributes	No need to learn SPICE syntax; Special Help dialogs for All SPICE analyses and elements are provided
Real Time Waveform Display	IsSpice4 displays instant feedback as the simulation runs. You don't have to wait for the simulation to finish in order to see the results
2 simple steps to change a value and resimulate	Seamless integration makes it easy to see the results of a value change
Complete SPICE netlist generation	No need to edit the SPICE netlist, Schematic and simulation always agree automatically
Parameter Passing	Generic models/subcircuits are supported
Monte Carlo support	Tolerance/Yield and Worst Case analysis are supported
Support for AHDL Models and user defined C code subroutines	Special attribute dialogs for user generated models allow ASIC, Board and System level designers to create custom SPICE models

4) or the waveform (Figure 5) pertaining to the selected item. This process is called cross-probing, and it makes analyzing your design effortless. The schematic-IsSpice4 combination is truly **interactive**. At any time, you can analyze simulation data, cross-probe waveforms, or edit your schematic, even while the simulation is running.

With the introduction of these **ICAP™** products, Intusoft finally brings integrated analog and mixed mode simulation to your schematic environment at a price you can afford, and with the ease of use you have come to expect from Intusoft.

Price And Availability

The price of any **ICAP™** system, no matter which schematic the user owns, is \$2995. Each **ICAP™** package is available in two versions. The **RF Designer's version** has a special SPICE library for RF parts, while the **Power Supply Designer's version** has a special SPICE library for power electronic parts. **This is the only difference.** Owners of Intusoft's current (version 7.6 or greater) ICAP/4Windows software can add any **ICAP™** package for \$495 each. The **ICAP™** for OrCAD package will be available on October 18, 1996. Other **ICAP™** packages will be available shortly thereafter.

New AHDL Models Extend SPICE's Reach

IsSpice4 simulator. The interface is composed of two parts; the SALT kernel, Array.DLL, and a set of OLE automation servers (Figure 6). The SALT kernel interfaces and synchronizes data arrays from external software or hardware. The OLE automation servers represent predefined SPICE models which dynamically link to IsSpice4 at run time. Intusoft provides several servers along with source code. These servers can be used as provided or, with the CMSDK, you can develop servers which are tailored to your specialized applications.

As the name implies, the type of data used by SALT is an array. The array is a matrix of time or frequency dependent data. It can be used to represent a sound, such as a .WAV file, large sets of analog simulator data stored by IsSpice4, abstract data taken from laboratory test hardware, or even links to other software. The possibilities are virtually endless.

SALT is ideal for the simulation of tracking and control problems which are common to robotics, guidance and navigation, image tracking, toys, and household appliances. These problems span technologies from active and passive electronic components to signal and image processing and heuristic control algorithms. Linking the basic arrays and your own SALT servers to the IsSpice4 simulation engine allows you to perform a comprehensive **system level simulation** of these problems. In some cases, the simulation can operate in real time and be used to collect data and test algorithm performance without first building prototype control and signal processing hardware. The power of SALT is in its ability to extend SPICE by allowing you to easily add models that perform tasks specific to your job. To get you going, however, SALT includes a number of predefined models, a few of which we review here.

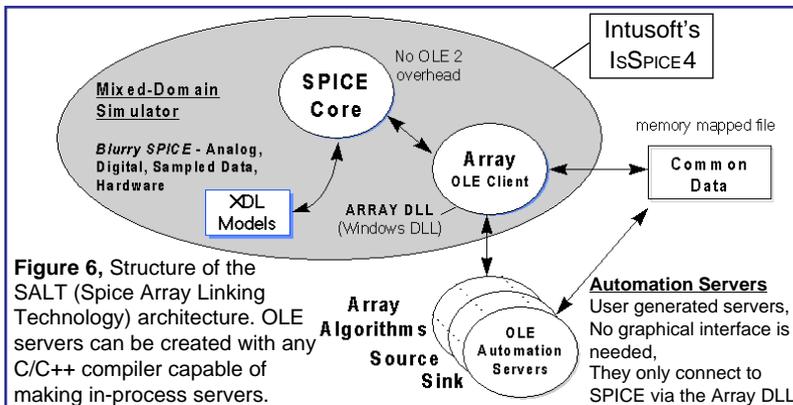
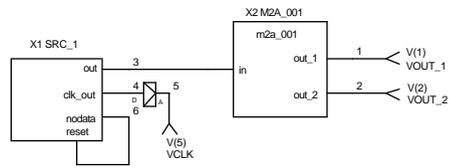


Figure 6, Structure of the SALT (Spice Array Linking Technology) architecture. OLE servers can be created with any C/C++ compiler capable of making in-process servers.

Figure 7, New IsSpice4 model for a repeating PWL stimulus generator. A random noise option is also available.



```
.Model Data_src Mat_Src
+ Op="Source"
+ Argname= "Matrix_File"
+ Dim=[128 3] Period=99N
+ Input=Data.Dta
```

Argname	Function
sound_file	Read .WAV files
matrix_file	read PWL data file
random	output random values
atod	hardware interface (A/D)

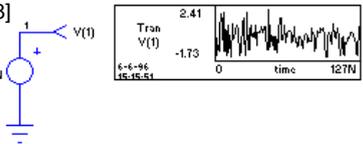
Random Noise, Repeating PWLs, External File Access

SALT provides both Source and Sink models. As the name implies these models retrieve or store data, respectively, from/to a file, a device, or another program. Referring to Figure 7, the source model, SRC_1, has several parameters. "SRC" calls the desired automation server, while "Argname" determines the function the model will perform. Each function has a set of additional parameters that modify the function. For example, wavelet compression is available for the sound_file function.

The matrix_file entry instructs the Source server to read its input from a file named by the "Input" parameter. The source reads a periods worth of data into the array matrix established by the "dim" model parameter. After every complete period the Clk_out signal goes high. At the end of the data file, the nodata signal goes high. This is fed into the reset signal which instructs the source to repeat the contents of the file. Hence we have a repetitive input signal from a finite set of piecewise linear data. The input file is a text file using a simple structure that is superior to the standard SPICE PWL format. The second model, M2A (matrix to analog), converts the array data to a form that can be used by traditional SPICE elements. The process for storing analog data to a disk file proceeds in a similar, but reverse direction (analog to matrix model -> Sink model -> file).

The subcircuit below demonstrates a random noise generator. The "Random" argname instructs the Source server to generate random values. The passed "seed" parameter is used to initialize the random pattern for each simulation. Using different

```
.Subckt RANGEN Hi Lo
.Model Mat_Toa_003 Mat_Toa (Dim=[128] Loc=[1 1 ])
.Model Mat_SRC_001 MAT_SRC(OP="Source"
+ Argname="Random Seed={ Seed }" Dim=[128]
+ Period=99N Compress = 0)
A1 99 Clkout2 Reset Reset Mat_Src_001 X2
A2 99 [2] Mat_Toa_003
E1 hi lo 2 0 { Mag }
.Ends
```



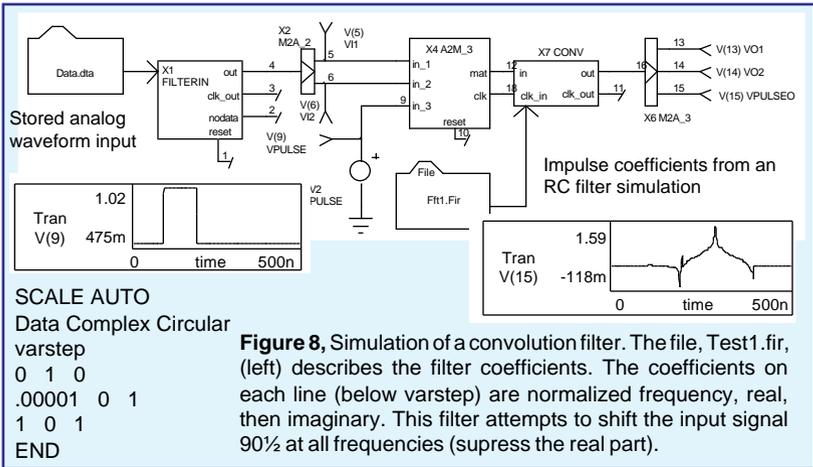


Figure 8, Simulation of a convolution filter. The file, Test1.fir, (left) describes the filter coefficients. The coefficients on each line (below varstep) are normalized frequency, real, then imaginary. This filter attempts to shift the input signal 90½ at all frequencies (suppress the real part).

“seed” values for different sources will insure that the sources are uncorrelated, but give repeatable data for each simulation.

Convolution filter and FFTs

Data in the array format, whether read in by the Source server or converted by the analog to matrix model, can be processed by a separate array model. The array model supports 2 servers; Fourier and Matrix Math. The Fourier server performs FFT, IFFT, and convolution functions. The next example, in Figure 8, demonstrates the latter (frequency domain multiplication). The model for the convolution element, X7 is:

```
.Model CONV_001 array(dim=[128 3] op="fourier"
+ argname="convolution" paramfile="Test1.Fir")
```

The “paramfile”, contains a text description of the filter coefficients for the convolution model. The “Circular” keyword on the “Data” line instructs the convolution algorithm to perform circular convolution. Time domain data, when transformed to frequency, has unique properties at frequencies greater than 1/2 the sampling frequency. The resulting frequency spectrum, above this Nyquist frequency is the complex conjugate of the lower frequency spectrum, due to the time series having no imaginary terms. When using this spectrum in a filter, it produces what is called “circular” convolution which causes the filter to appear periodic. For aperiodic filtering, it is necessary to make the second half of the frequency spectrum zero. This effectively doubles the length of the filter. If you are applying a filter to a periodic wave form use the “Circular” keyword. Note that these convolution filters are only approximations of continuous filters. The impulse response must approach zero at the end of the filter period. Filters that don’t meet this criteria will tend to become circular.

For more information on the SALT model parameters and file formats, please see the SALT documentation posted on the Intusoft Web Site.

New Unified SOI MOSFET Model

Fully-depleted SOI MOS devices offer significant advantages over conventional bulk MOSFETs, for example, smaller parasitic capacitances, better performance in harsh environments, and absence of latchup in SOI CMOS circuits. In low-voltage low-power, or high-speed digital applications, the SOI CMOS technology seems especially promising. For these reasons the usage of fully depleted SOI MOSFETs is increasing. In particular, high-performance analog applications have been recently demonstrated. Accurate CAD-compatible models for these devices are therefore necessary.

A new C_{∞} -continuous fully-depleted SOI MOSFET model, suitable for simulation, is available in IsSpice4. This model was developed by B. Iniguez (from UIB, Universitat de les Illes Balears, Spain), and D. Flandre, L. F. Ferreira and B. Gentinne from UCL (Universite Catholique de Louvain, Belgium) [1]. The model is written in XDL (XSPICE Description Language), an AHDL supported by a number of SPICE vendors.

The model is based on the charge-sheet model unified expressions of the inversion charge densities, which are used to calculate all large and small-signal parameters (including terminal charges and capacitances) using the QS (quasi-static) approximation. Since all expressions have an infinite order of continuity, the model provides smooth transitions between the different regimes of operation. Small-geometry effects have been included in the model using physical and continuous equations. The model has been proven to accurately model current and capacitance measurements for devices with channel lengths down to $1\mu\text{m}$ and frequencies surpassing 1GHz. The clear meaning of most model parameter values allows good estimations using direct techniques. Afterwards, their values can be improved using a global optimization technique.

The new model is of great utility for analog design, where continuous and accurate derivatives of the drain current, charges and capacitances are required. In particular, the model is appropriate for low-power low-voltage analog circuits, where transistors are biased in the moderate inversion regime, and especially for circuits where charge conservation is critical (like switched-capacitor and current circuits). Since the model is also very efficient for ultra low-voltage digital circuits in which subthreshold operation may no longer be negligible, its application range also extends to mixed analog-digital circuits.

More Info
on the New
SOI Mosfet
will be
available in
the next
Newsletter.

[1] B. Iniguez, L. F. Ferreira, B. Gentinne and D. Flandre. "A Physically Based C_{∞} -Continuous Fully-Depleted SOI Mosfet Model for Analog Applications", IEEE Trans on Elect. Dev., vol 43, #4, April 1996.

New AHDL Switch Model

Shown below is the majority of the AHDL code for a new analog switch model. The model provides a voltage controlled impedance with a smooth (continuous derivatives) transition region between the on and off states. While the same switch can be modeled with a behavioral model, the AHDL version runs about 3 times faster and requires fewer simulation timepoints.

Users can develop their own compiled AHDL models with the CMSDK package. You can even run IsSpice4 in a debug mode so that you can interactively debug your model code with Microsoft Visual C++ using your actual test circuits. To find out about developing your own models check the Intusoft Web Site.

The new switch model and source code are posted on the Intusoft Web Site and the *Intusoft Newsletter* floppy for subscribers. To activate the new model, just copy the Switch.DLL file into your IsSpice4 directory. The power to create and compile a C code based model and add it so easily to IsSpice4 is a big advantage for Intusoft customers.

```
void cm_vswitch(ARGS)
{
  Variable Declarations
  Input/Model Parameter Assignment
  switch (ANALYSIS){
    case MIF_DC:
    case MIF_TRAN:
      if (von > voff){
        if (vin < voff){
          iout = vout/roff;      OUTPUT(out) = iout;
          PARTIAL(out,out) = 1.0/roff;}
        else if (vin > von){
          iout = vout/ron;
          OUTPUT(out) = iout;
          PARTIAL(out,out) = 1.0/ron;}
        else{
          vdfif = von-voff;
          vsum = von+voff;
          aux1 = log(sqrt(ron*roff));
          aux2 = log(ron/roff);
          aux3 = vin-vsum*0.5;
          aux4 = (3.0*aux2*aux3)/(2.0*vdfif);
          aux5 = (2.0*aux2*pow(aux3,3.0))/pow(vdfif,3);
          geq = 1.0/exp(aux1+aux4-aux5);
          iout = vout*geq;
          OUTPUT(out) = iout;
          PARTIAL(out,out) = geq;}
        } else /*Voff <= Von Case */
          Similar to Von > Voff section
        }
      break;
    case MIF_AC:
      AC Analysis Processing
  }
}
```

Voltage-Controlled Switch Syntax
A1 Out+ Out- In+ In- ModName
.Model Modname Vswitch
+ Ron=1 Roff=1Meg
+ Von=1 Voff=0

Figure 9, AHDL code for the new analog switch model. The DLL file containing the model is posted on the Intusoft Web site.

The AHDL is based mainly on C with extensions that eliminate the need for you to understand the internal working of SPICE in order to develop a model.

Once a model is developed it can be shared with other IsSPICE users. You don't need the CMSDK to use the model, just to create it.

IsSPICE4 Scripting Gives You More Power

By Scott Frankel, Analytical Engineering

One of the most powerful features of Intusoft's SPICE is the Interactive Command Language (ICL). Unfortunately, it is also one of the most unused features. This article will attempt to illustrate the potential capabilities of ICL by demonstrating its ability to make complex simulations easy.

ICL is basically a macro or scripting language. You can use ICL to perform complex tasks, or test procedures, using a series of commands rather than by pulling down a menu or by double clicking a part and changing its value. The ICL contains over 60 commands that allow you to:

- Sweep any circuit variable (resistance, voltage, temperature, model parameters, etc.)
- Watch for any operating condition or combination of circuit variables and make decisions on what test path to choose based on the results
- Set Simulation Breakpoints on any circuit condition
- Run any SPICE analysis, in any order, as many times as desired
- Record data from anywhere in the circuit (curve families, power dissip., complex measurements, etc.)

ICL scripts can be run in one of two ways; either batch style or interactively. If the netlist is placed in the input .CIR file, IsSpice4 will auto-run the entire script when the simulator is started. If the script is placed in the Simulation Control Window inside IsSpice4, then the script can be run interactively, line by line. The latter method is useful for script debugging. In both cases, the simulation is not limited to only 1 AC, DC, or transient analysis. Any number of analyses can be run. Two example scripts are shown below.

Parameter Sweeping - ICL Script Example #1

```
view tran v(3)           Display waveform in real time
foreach param -2 -4 -6  Parameter values for dummy variable "param"
alter @V7[DC]=$param     Change the DC value of V7 to equal param
tran 1n 200n             Run a transient analysis
print v(3)               Save the data for V(3)
end                       Do next parameter (if any)
```

Repeating Loop W/Breakpoints - ICL Script Example #2

```
save all allcur allpow   Save data
view tran v(3)           Real time display
function rms(vec) sqrt(mean(vec*vec)) Create a function
dowhile rms(v(3)) > 400m Do script while V(3) > .4
stop when @q5[p]<.25 when V(8)>1.6 Break if OR condition is true
tran 1n 100n             Run a transient analysis
alter @q5[temp]=@q5[temp]+10 Increment the temperature
print @q5[temp] rms(v(3)) Save data
end                       End loop
```

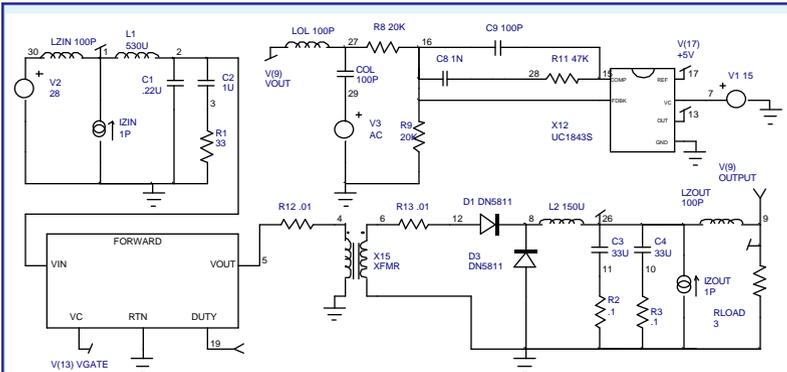


Figure 10, Schematic of a buck converter. The simulation is driven by the ICL script in Figure 11.

The circuit in Figure 10 is a forward buck derived converter. You will notice that there are several inductors, capacitors, and AC current and voltage sources in the schematic that do not affect the operation of the circuit. We will use these components to measure the frequency response, and input and output impedance with a single simulation. The ICL script is shown in Figure

```
*INCLUDE ICL.TST ; Get script (.Control -> *.PRINT lines) out of the ICL.TST file
.CONTROL ; Start of the Included ICL script
SAVE ALL ; Save all node voltages, necessary to print out waveforms later
*The next two lines effectively open the control loop so we can check the loop response
ALTER @LOL[INDUCTANCE]=100 ; Set LOL = 100 Henries
ALTER @COL[CAPACITANCE]=100 ; Set COL = 100 Farads
ALIAS GAIN VDB(9) ; Measure the gain in DB at node 9
ALIAS PHASE VP(9) ; Measure the phase at node 9
AC DEC 20 10 1000K ; Perform an AC analysis
PRINT GAIN PHASE ; Save the Gain and Phase waveforms to the output file
* The next section of code allows the circuit to measure input impedance
ALTER @LOL[INDUCTANCE]=10P ; Set LOL = 10pH in order to close the loop
ALTER @COL[CAPACITANCE]=10P ; Set COL = 10pF in order to close the loop
ALTER @LZIN[INDUCTANCE]=100 ; Set LZIN = 100H in order to inject an AC signal
* into the input in order to measure input impedance
ALTER @IZIN[ACMAG]=1 ; Inject an AC current of 1 (magnitude)
ALTER @Izout[ACMAG]=0 ; Turn off AC current source on the output of the converter
ALIAS ZIN VDB(1) ; Measure the gain in DB at node 1
AC DEC 20 10 100K ; Perform another AC analysis
PRINT ZIN ; Saves the input impedance waveform to the output file
*The next section of code allow the circuit to measure output impedance
ALTER @LZIN=1P ; Set LZIN = 1PH
ALTER @IZIN[ACMAG]=0 ; Change input AC magnitude to zero
ALTER @Izout[ACMAG]=1 ; Change output AC magnitude to 1A
ALIAS ZOUT VDB(26) ; Measure the gain in DB at node 26
AC DEC 20 10 100K ; Perform another AC analysis
PRINT ZOUT ; Save the output impedance waveform to the output file
.ENDC ; End the ICL control block
*.PRINT AC GAIN PHASE ; Optional, Generate IntuScope Waveform List
*.PRINT AC ZOUT
*.PRINT AC ZIN
```

Figure 11, An ICL script (with explanations) used by IsSPICE4 to run a series of unattended AC simulations on the forward converter.

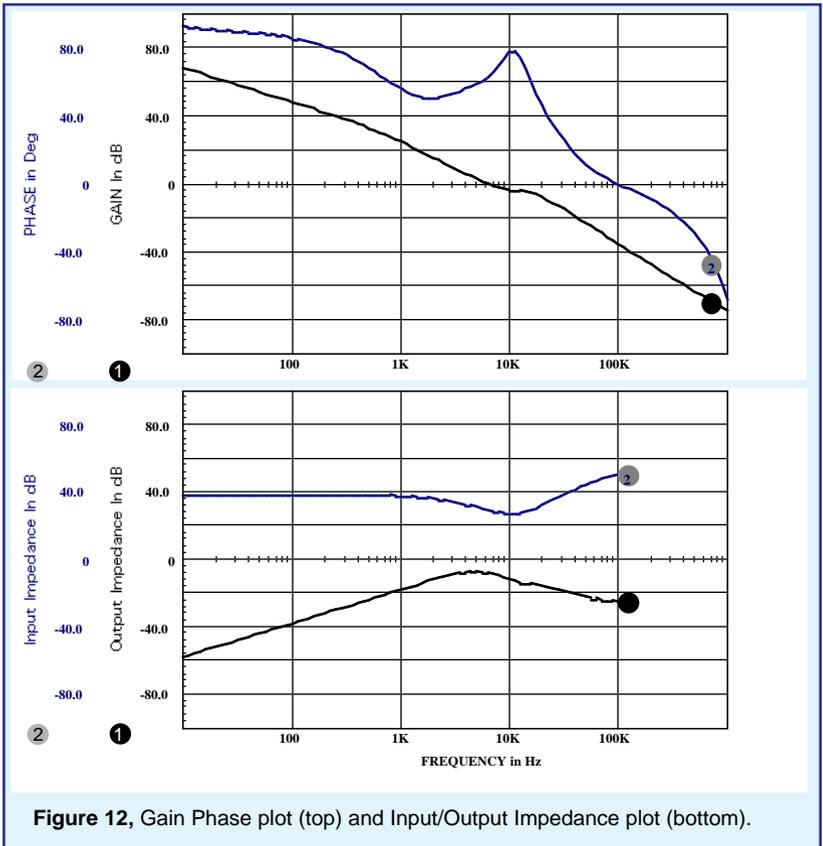


Figure 12, Gain Phase plot (top) and Input/Output Impedance plot (bottom).

11. The script, called by the*INCLUDE statement, is appended to the SPICE netlist and run when IsSpice4 is started. In this batch style mode, the .Control and .Endc statements are required. The results of the ICL script are shown in Figure 12.

Remember, without ICL, these results would have taken three separate SPICE runs with three separate schematics and three separate netlists. The simple demonstration presented here is only a hint of the abilities of ICL. This capability could be used to run a Monte Carlo analysis or an Optimizer routine and return numerous parameters using only one IsSpice4 simulation file! Clearly the potential benefits of ICL suggest more investigation into this powerful tool. Read the chapter on ICL in your IsSpice4 manual and start saving time today.

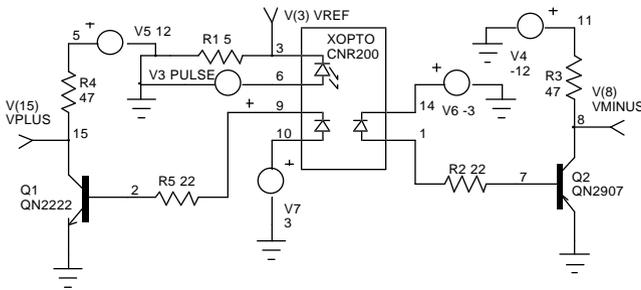
Scott Frankel graduated from Arizona State University with a BSEE degree. He currently works at Analytical Engineering Incorporated (602) 890-7197. AEI offers engineering design, simulation, and circuit modeling services.

H-P Gives Intusoft New Opto Models

Three new opto-isolator models are available on the *Intusoft Newsletter* floppy disk for subscribers. Included are the HCPL-7820 high-Common-Mode Rejection (CMR) analog isolation amplifier, the HCNR200 High-linearity temperature insensitive isolation amplifier, and HCPL-4506 High-CMR opto-isolator for Intelligent Power Module (IPM) Applications.

The example in Figure 13 uses the CNR200 Opto-Isolator to control two complementary switching transistor circuits. The photodiode within the opto-isolator is driven by a pulse (V3 6 0 PULSE 0 110 1M .01M .01M 5M 11M). We will use the ICL script feature of IsSpice4 to drive the simulation.

After starting IsSpice4 we pasted the temperature sweep ICL script (shown in Figure 14) into the Simulation Control Dialog that pops up. Clicking the DoScript button runs the script. The



```
.SUBCKT CNR200 1 2 3 4 5 6 ; Temperature Insensitive isolation amplifier
*Connections: LED Cathode Anode PD1 Cathode Anode PD2 Anode PD2 Cathode
QLED1 8 2 7 QCPL .5 ;LED circuit below
QLED2 9 2 7 QCPL .5
VLED 7 1 DC .685
VPD1 1 8 DC -2
VPD2 1 9 DC -2 ; Input photodiode circuit below
DPD1 4 3 DPHOTO
FPD1 3 4 POLY(1) VPD1 0 1 ; 32.5 -450K 3.1G
CPD1 4 3 8P ; Output photodiode below
DPD2 5 6 DPHOTO
FPD2 6 5 POLY(1) VPD2 0 1 ; 32.5 -450K 3.1G
CPD2 5 6 8P
.MODEL DPHOTO D(IS=4.5E-12 RS=150 N=1.3
+ XTI=4 EG=1.11 CJO=14P M=0.9 VJ=1.9)
.MODEL QCPL NPN(IS=2.214E-19 BF=10m
+ NF=1.010 IKF=11.00m ISE=1.167P
+ NE=1.737 RB=3.469 VAF=100 TF=1.77U
+ CJE=80P); LED/Optical-coupling transistor model
.ENDS
```

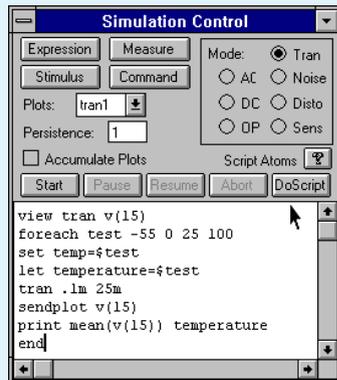


Figure 13, Opto-Isolator test circuit using the H-P CNR200 temperature insensitive isolation amplifier. The ICL script is run interactively from the IsSpice4 Simulation Control dialog.

Figure 14, Temperature Sweep ICL Script

view tran v(15)	Real time display
foreach test -55 0 25 100	Parameter values
set temp=\$test	Change the circuit temperature
let temperature=\$test	Create dummy variable to hold temp value
tran .1m 25m	Run a transient analysis
sendplot V(15)	Create a curve family in IntuScope
print mean(v(15)) temperature	Save data and do next param
end	Do next parameter (if any)

script uses the sendplot command to send waveforms to IntuScope in order to create a curve family. As IsSpice4 runs the simulation, the calculated waveform data is sent to IntuScope. Therefore, make sure that IntuScope is running when you click on the DoScript button in the IsSpice Simulation Control Dialog.

We used the ICL sweep script to generate four curves for V(15). The peak amplitude of the waveform for the -55 degrees C case was slightly less than the peak level at the other temperatures.

Thanks to Drew Plant and H-P for the models. 350 W. Trimble Rd., MS 90TN San Jose, CA 95131, drew_plant@hp-sanjose om1.om.hp.com

New Magnetics Design Software

Intusoft has released a new software program for transformer and inductor design. The program, called **Magnetics Designer** synthesizes and analyzes various types of transformers, inductors, and chokes. It is optimized to design layer wound magnetics using cores that have rectangular or circular winding areas. Typical designs may include ferrite pot cores, EI, EE, EP, cut C, and RM cores, toroids, and EI laminated cores. Magnetics Designer is capable of designing single phase multi-winding transformers and inductors operating from 10Hz to well above 1MHz. Typical applications include 60 Hz line transformers, switching regulator transformers, and output filter inductors. A database with thousands of cores and a wide variety of materials is included with the program. You can manipulate over 20 parameters such as the wire size and type, number of strands, gap length, insulation thickness and more. Magnetics Designer will predict characteristics like peak flux density, AC flux density, DC and high frequency AC resistance, core and copper losses, leakage inductance, winding capacitance, weight, efficiency, temperature rise, and window fill percentage. Magnetics Designer also produces a SPICE model of your transformer or inductor.

The price is \$1500. More information, including an application note and a demonstration version of Magnetics Designer, is available on the Intusoft Web Site <http://www.intusoft.com>.

All Prices shown are U.S. Domestic ONLY. Prices in Foreign countries may vary.

WHAT MAKES IsSPICE SO POWERFUL

IsSPICE4 provides a quantum leap in performance over other analog and mixed mode simulators. It is the first commercially available version of SPICE based on Berkeley SPICE 3F.5 and Georgia Institute of Technology's XSPICE.

IsSPICE4 allows you to explore circuit performance by interactively running different analyses and sweeping any circuit variable. With the ability to simulate electrical, sampled-data, mechanical, physical, thermal, and other systems, **IsSPICE4 is the ONLY true native mixed mode SPICE 3 based simulator.** The advanced features of IsSPICE4 allow all types of applications to be simulated: switch mode power supplies, mixed signal ASICs, RF communication systems, interconnects, control systems, and mixed mechanical/physical systems.

Intusoft has spent hundreds of man-hours improving SPICE. And although IsSPICE4 is based on SPICE 3, Intusoft has greatly enhanced the program over and above the public domain version and added more features, an interactive interface, superior analysis and model support, and improved convergence algorithms - all for a price no one can match. IsSPICE4 is simply the best and most affordable SPICE program on the market today. Just take a look at some of its features:

State-of-the-Art Operation

- Native mixed mode simulation - IsSPICE4 includes an event driven simulator that supports mixed analog, digital and DSP circuits.
- Interactive Operation - IsSPICE4 operates interactively, and frees you from the restrictive batch style of older SPICE simulators.

Built-in Models

- Elements: Resistors, Capacitors, Inductors, Coupled Inductors, Transmission Lines, Diodes, BJTs, JFETs, MOSFETs (Level 1-8), GaAs Mesfets, Switches, and Boolean logic expressions.
- Digital and AHDL Models: Digital logic gates, Flip-Flops, Latches, State Machine, Frequency Divider, RAM, Sampled-Data Filters, Nonlinear VCOs, Laplace Equations
- Behavioral Modeling: In-line Equations, Table models, If-Then-Else

Advanced Models

- HDL Models and C Subroutines; Create models based on a powerful non-proprietary HDL using C
- Develop your own OLE2 interfaces and user defined algorithms
- Support for non-electrical applications and top-down system design
- Powerful Behavioral Modeling
- Three types of mixed mode modeling
- Lossy (distributed) transmission lines with frequency dependent losses
- MOS: BSIM1, BSIM2, and New BSIM3 version3

Analysis Support

- AC, DC, transient, noise, Fourier, distortion, DC/AC sensitivity, Pole-Zero analyses, and Temperature variations on individual elements
- Monte Carlo Analysis, Circuit Optimization/Performance Analysis

Additional Interactive, AHDL & Mixed Mode Features

- Real-time Display of voltages, currents and power dissipation
- Simulation Scripts: a robust scripting language that allows simulation breakpoints and loops of different analyses to be run as a test procedure.
- Interactively run analyses without having to edit the netlist or restart the simulator, Add, delete, or rescale waveforms on the real-time display
- Digital Simulation: 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

Convergence and Speed Improvements

- Automatic Gmin stepping algorithm
- Automatic Source stepping algorithm
- New Pseudo-Transient algorithm
- Improved Predictor-Corrector, Latency, and Bypass algorithms
- Improved program defaults
- Alternate UIC algorithm
- Special Circuit Debugging Options
- Full Gear Integration option

Compatibility

- Works with ALL popular schematic entry programs

Did You Know Intusoft was the First?

The following is a list of capabilities that Intusoft introduced to the analog simulation world.

- **SPICE 2 Models for:** IGBTs, fuses, lasers, vacuum tubes, generic template models, dual gate Mosfets, SC filters, neural networks, digital gates, RF beads, IBIS buffers, saturable cores, and PWMs (using the state space approach)
- **Products/Features:** 32-bit version of SPICE for DOS, integrated schematic entry dedicated to SPICE, SPICE 2 compatible model generation software, support for all Macintosh platforms, parameter passing

A SPICE For Everyone!

Affordability is the hallmark of Intusoft's products. With our ICAP/4Lite, ICAP/4Windows, and ICAP/4Macintosh software all affordably priced, there's an IsSPICE that's just right for you.