

Intusoft Newsletter

Personal Computer Circuit & System Design Tools



Copyright © Intusoft, All Rights Reserved

Issue #48 Nov. 1996

Tel. (310) 833-0710

Fax (310) 833-9658

SPICE made EASY

Intusoft will release an updated version of the SpiceNet schematic entry package on December 16, 1996. The new version of SpiceNet, v8.0, will be available FREE of charge to any interested party via download from Intusoft's web site. It is also available as part of the new ICAP/4Windows version 8.0. Unlike other giveaway versions, SpiceNet is not an evaluation version. It is a fully functional schematic entry package.

Continued on pg. 5

New Design Validator™ Option

Schematic entry for analog and mixed mode simulation is a virtual necessity in order to avoid making netlist entry errors. But when it comes to building a circuit for production, the hapless purchasing agent ...

Continued on pg. 8

In This Issue

- 2 ICAP™ For OrCAD Integration is Here
- 5 New SpiceNet Schematic Entry Posted For FREE
- 8 NewDesignValidator™
- 10 VB Scripting w/SPICE
- 12 Mixed Mode Modeling: Digital Potentiometer
- 14 SPICE CD-Rom
- 15 MD Makes EDN Top 100
- 16 Why is IsSpice Better!
- 18 New SMPS Book

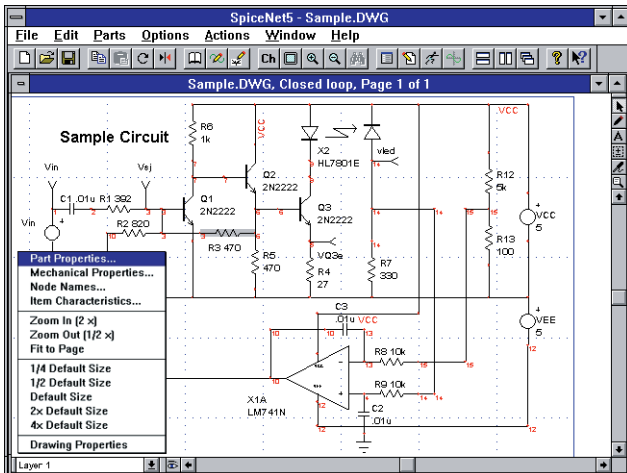


Figure 1, Intusoft's New SpiceNet schematic entry program makes SPICE easier to use than ever before. And best of all, IT'S FREE!

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

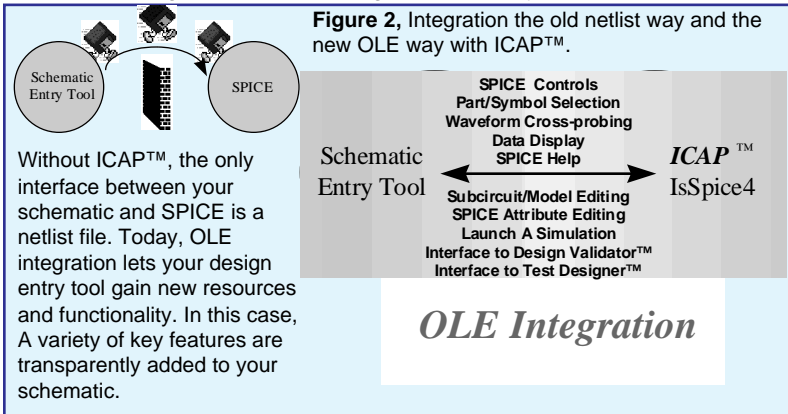
ICAP™ Links with OrCAD, Protel, and Viewlogic

Intusoft is proud to announce that the wait for seamless integration of SPICE 3 based simulation with your schematic capture package is over! Intusoft has introduced 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 a complete set of analog and mixed signal capabilities into the OrCAD Capture™, Protel Schematic³™, and Viewlogic Workview Office™ schematic environments.

The old integration methodology connected simulators to your schematic via a netlist as shown in Figure 2. OLE allows the integration between two programs to take place at a much more sophisticated level even though the programs may come from different vendors. Indeed, many of your favorite programs (Excel, Word) use this paradigm to interconnect different software modules developed by the same vendor.

The benefits of getting all your tools from a single vendor are greatly overstated and largely inaccurate. Because of the nature of the software development process and the expertise required to develop even a single EDA tool, the best tools in each class are not available from a single vendor. Fortunately, in today's software environments, different applications can be just as integrated as a single application.

The **ICAP™** software eliminates the restrictive netlist interface of the past and replaces it with new features that make it easy to perform circuit simulation. For example, SPICE model editing, custom SPICE attribute dialogs for each element, special dialogs for SPICE analysis statements, waveform and operating point cross-probing, and the ability to launch and control the



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

simulator from the schematic (Figure 2) are all directly accessible from within your schematic environment.

Two new and extremely powerful features are the automated parameter sweeping action and the **ICAP™** Analysis Wizard. For example, in Figure 3, an “Alter” dialog is added to Capture. It is accessed through a new menu function (Alter Selected Component) in the Accessories menu. You can choose to sweep any part value or the circuit temperature. A list, linear range, or logarithmic range of values can be swept. When the Run button is pushed, all the simulations run automatically and a curve family is displayed in the IntuScope program (Figure 4). Performing “What-If” studies has never been easier.

The Accessories menu also provides access to a new IsSpice4 Simulation Controls dialog (see Figure 7). The dialog makes it easy to specify the analyses you want to run. For beginners, an even easier method is provided; an Analysis Wizard. Shown in Figure 5, the Analysis Wizard leads you through a series of dialogs and prompts you for data along the way. When finished, your answers are converted into the proper SPICE syntax.

ICAP™ includes the **IsSpice4** native mixed mode (analog/digital) circuit simulator, SPICE **Model Libraries** with over 9000 analog and digital parts, the **IntuScope** graphical waveform analyzer, the **SpiceMod** SPICE modeling program, and a set of symbol libraries. These features are all integrated directly into your schematic. No netlist editing and virtually no knowledge of arcane SPICE syntax is needed anymore.

Figure 3, ICAP adds a simple yet powerful quick change feature to Capture. With it, you can sweep any part value, or the circuit temperature, and automatically plot a curve family (Figure 4). An added feature is the display of node voltages and branch current directly on the schematic.

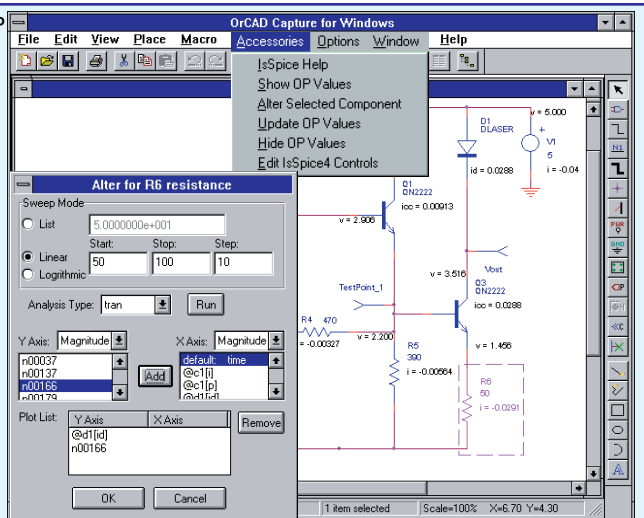
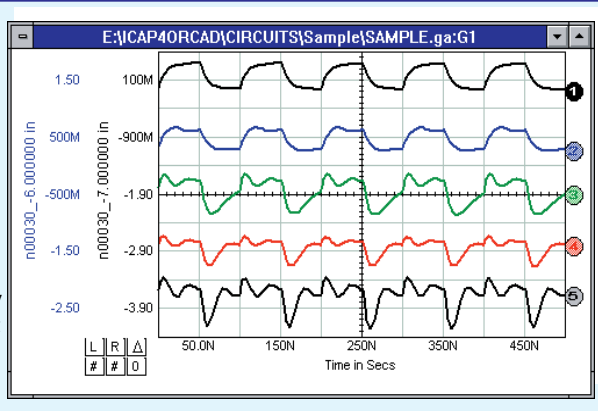


Figure 4, An IntuScope graph generated by the Alter dialog. This curve family was automatically created by sweeping a bias source and monitoring a node voltage. You can sweep virtually any parameter and plot any voltage, current or power dissipation.



Price And Availability

ICAP™ for OrCAD is available immediately. The Protel and Viewlogic version release dates will be announced in the next newsletter. The price of ICAP™, (for OrCAD, Protel, or Viewlogic), 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.

Figure 5, Both ICAP™ and the new SpiceNet packages feature Analysis and Convergence Wizards. These greatly ease difficult aspects of using SPICE. The set of Analysis Wizard dialogs for the transient analysis is shown here.

Intusoft Releases New Schematic Version

Continued
from
page 1

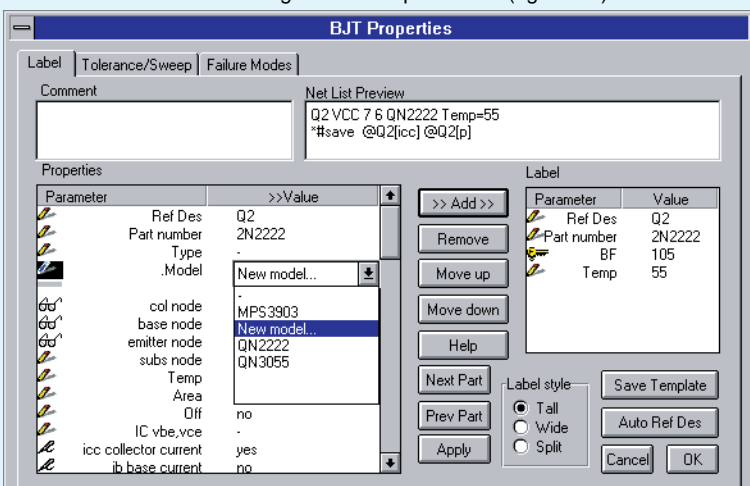
The new SpiceNet program is designed to make SPICE easier to use than ever before, while still providing a wealth of uncompromising features for the power user. Many new features have been added, including several that are not available in any other schematic package.

The most significant change is to the SPICE attribute dialogs. A custom "properties" dialog is included for every element. Figure 6 shows the BJT properties dialog. Through it you can manipulate every aspect of the SPICE transistor model including instance and model parameters. **No knowledge of SPICE syntax is necessary.** You can even edit SPICE subcircuit netlists and model parameters directly in the schematic. SpiceNet is smart enough to create a new model instance on-the-fly if you change any parameters.

A new SPICE analysis setup feature is provided (Figure 7). It contains a series of fill-in-the-blank dialogs for each IsSpice4 analysis. Again, the need to remember complex SPICE syntax is eliminated. You can even set up and choose from different sets of analysis statements.

SpiceNet contains improved support for mixed mode circuits, part tolerances, and Monte Carlo analysis. There is also a new feature for defining part failure modes. A custom part sweeping

Figure 6, The new BJT Properties dialog provides greatly expanded access to device attributes. You can now edit models (left side) and subcircuits directly from the schematic and configure custom part labels (right side).



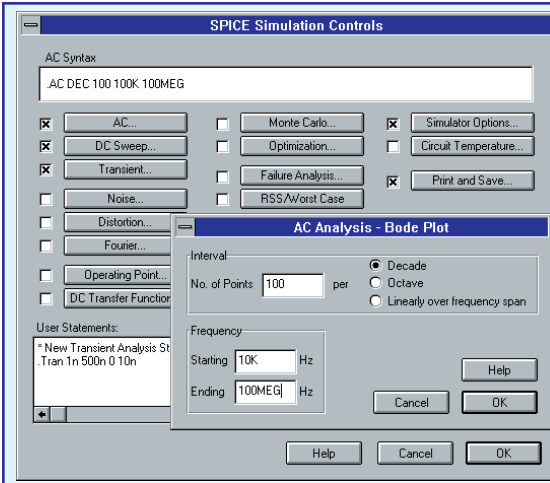


Figure 7, Entry of SPICE analysis statements is greatly eased by the new Simulation Setup dialogs.

They provide easy access to all of IsSpice4's powerful analysis capabilities.

There is no need to learn any SPICE syntax.

dialog completely automates the generation of curve families (Figure 3). The difficulties associated with SPICE signal generator syntax are handled in the schematic with new stimulus dialogs.

While SpiceNet is directed primarily toward driving analog and mixed signal simulation, it provides virtually all common schematic editing functions. For example, SpiceNet supports full concurrent multipage, multidesign editing. You can even split the screen to obtain multiple views of the schematic. The revised symbol editor is now interactive and contains many unique features. For instance, you can import artwork via the clipboard and utilize Windows meta-files and bitmap images in symbols to create superior looking schematics. Embedded OLE objects, like MS Word equations and documents and Excel spreadsheets can be placed directly into SpiceNet.

Another powerful new feature is the ability to handle mixed-technology or complex electrical designs via a multilayer structure. Schematic objects can be mixed on various layers to create various simulation scenarios. Users can specify which layers to view, edit, print simulate and combine into separate "Configurations" (Figure 8). This allows different levels of complexity and connectivity to be easily manipulated.

Several new features are included to empower ASIC and System designers. For ASIC designers, there is support for parameter passing, equation based generic modeling, and symbol substitution. For system designers, there is support for custom C code subroutines including OLE dialog generation and symbol generation from within user defined DLLs, plus complete support for equation based blocks.

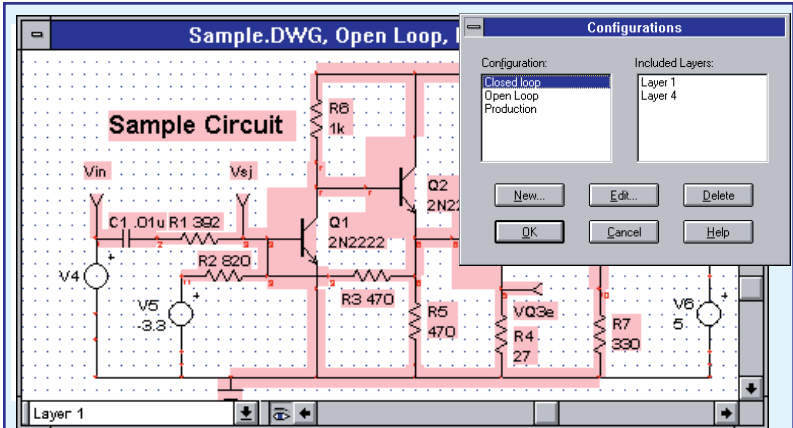


Figure 8, SpiceNet supports multiple configurations consisting of different layers. Each layer can contain different elements of the design. You can simulate any or all configurations, and SpiceNet will do the work of building the correct netlist. Shown above is the Configuration setup dialog and the highlighted circuitry from a single layer. The non-highlighted parts are on a different layer.

The specialized support for IsSpice4 and its numerous SPICE extensions, plus its interactive display features, are what separates SpiceNet from other schematic programs.

Why Is Intusoft Distributing SpiceNet For Free?

The Internet marketing model has shown that giving away good products can have positive results. For the EDA world, this is a significant departure from the try-before-buy limited demo marketing approach. It is Intusoft's aim to provide the best-in-class simulation tools no matter what schematic tool you are using. We hope to get more people to buy our simulation tools by seeing how easy it is to run SPICE. Distributing SpiceNet helps us do that. Once convinced of the benefits of simulation, you can either purchase IsSpice4 and use SpiceNet or utilize ICAP™ to link IsSpice4 to another schematic entry system.

What's FREE And What Do You Pay For?

SpiceNet is included in new ICAP/4Windows version 8.0. An upgrade to v8.0 from any previous 7.5x version is \$495 and includes the first 8.x update. ICAP/4Windows 8.0 is available after December 16, 1996. The ICAP/4 update includes SpiceNet v8.0, a new ICAPS menu shell interface, enhanced parameter sweeping and cross-probing, enhanced SPICE model libraries, new Library Manager utility, improved Monte Carlo analysis, new Batch Mode multi-simulation interface, a basic analog and mixed signal fault analysis capability, and interfaces to the new Design Validator™ and Test Designer™ options. If you have purchased ICAP/4Windows after September 15, 1996 the upgrade to version 8.0 is free.

Design Validator™, New Technology Supporting Validation and Test Set Design

Continued from page 1

...can't buy the parasitic parts that were included in the simulation and the technician can't locate the SPICE pulse generator for the device under test. For each major test configuration, the design engineer needs to generate a new schematic. Design changes must then be copied back into each version of the schematic to validate the design integrity.

With SpiceNet 8.0 Intusoft provides powerful new features that not only solve these problems, but also enable the circuit designer to have **validation test suites** and provide the framework for **automatic test set software design**.

SpiceNet divides the schematic into layers and configurations (Figure 8). Each configuration is composed of a combination of layers. For example, the core circuitry could be on one layer (called "main circuitry") and the SPICE generators and test circuitry on a second layer (called "spice stuff"). A third layer could contain the PWB interface (called "PWB interface"). Then we could have 2 configurations (called "production" and "Spice Test") as shown in the following table:

Layers -->	main circuitry	spice stuff	PWB interface
Configurations			
Production	X		X
Spice Test	X	X	

Moreover, you could have any number of layers and configurations to test various aspects of your design; for example, you could add feedback circuitry to check out closed loop performance or data processing circuitry for mixing or demodulation.

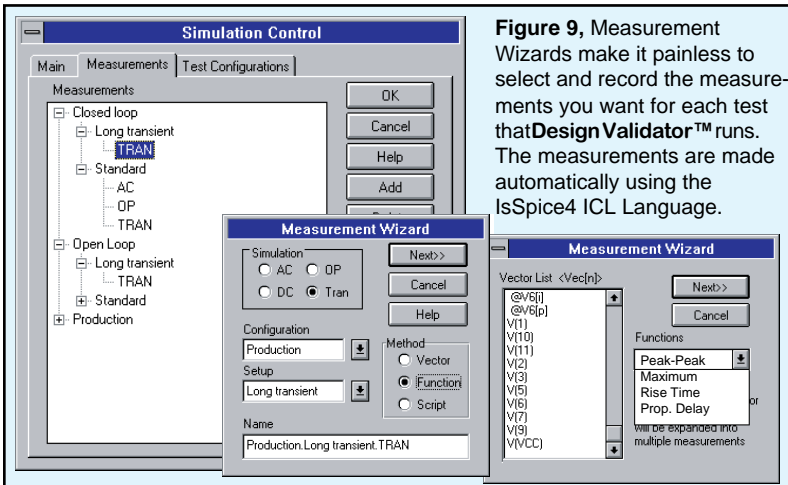


Figure 9, Measurement Wizards make it painless to select and record the measurements you want for each test that **Design Validator™** runs. The measurements are made automatically using the IsSpice4 ICL Language.

Design Validator™ allows each configuration to be combined with a set of IsSpice4 analysis settings. The IsSpice4 settings direct the type of simulation (AC, Transient). The combination of analyses and a configuration is referred to as a “Test”. Each test is assigned one or more measurements (Figure 9). **Design Validator™** returns the measurement results to the schematic after running the tests, where the results are organized into reports. The reports contain the measurement limits which are used to grade the circuit performance. The table below illustrates the pairings of analyses and measurements with a configuration in order to define a test.

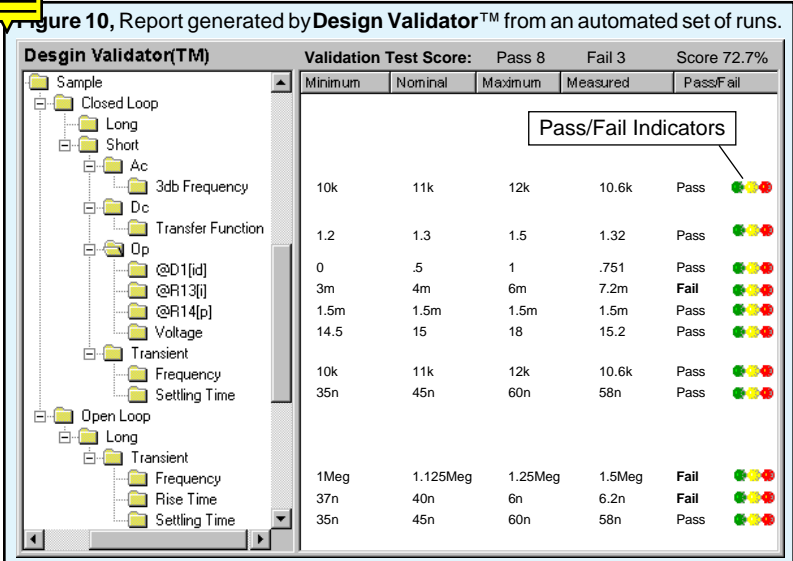
	<i>Configuration</i>	<i>Analysis 1</i>	<i>Analysis 2</i>
<i>Measurements</i>	Spice Test	DC operating point	Transient
DC Test	X	X	
Rise time	X		X

Simply put, **Design Validator™** keeps track of multiple circuit configurations, analysis setups and measurements. It then **runs the entire suite of tests automatically**. Finally, **Design Validator™** creates a report that summarizes the pass-fail grades and lists the results in a report which is similar to Figure 10. This powerful capability extends SPICE well beyond its current single simulation oriented boundaries. The price of the **Design Validator™** option is \$995. It requires ICAP/4 version 8.0. It will be available January 20, 1996.

In our next newsletter, we will show you the new IsSpice4 measurement capabilities and introduce our new Test Designer™ software which extends the **Design Validator™** concepts deep into the Test Software Development arena.



Figure 10, Report generated by **Design Validator™** from an automated set of runs.



Visual Basic Scripting With IsSPICE4

OLE provides a common framework through which programs from one or more vendors can communicate, and a common foundation in which programmers can work. In its simplest form, OLE is a technology that allows one program, an OLE Controller, to take advantage of services provided by another program, an OLE Server. The computing features and functions exposed by the OLE Server are called OLE objects. An OLE interface essentially allows two programs to share functionality without having to know about one another.

IsSpice4 supports two types of OLE enhancements. The first is that **IsSpice4 is an OLE Server**. This means that any OLE Controller can “drive” the simulator. For example, a math program, like MATLAB, could use IsSpice4 as a simulation engine and ask it to run an analysis and pass back the results so that they can be used in a subsequent MATLAB calculation.

Secondly, IsSpice4 exposes an Interactive Command Language (ICL) as an accessible set of OLE objects. ICL is a scripting language that contains a wide variety of simulation related functions such as analysis commands and functions that change part values.

A simple “script”, or macro, to run 2 simulations would be:

```
view tran v(3) ; display V(3) as the simulation runs
tran 1n 200n ; run a transient analysis for 200ns
print v(3) ; send the data for V(3) to the OLE Controller
alter @r1[resistance]=@r1[resistance]*1.10 ; increment R1 10%
tran 2n 300n ; run another longer transient analysis for 300ns
print v(3) ; send the data for V(3) to the OLE Controller
```

The ICL is a publicly available protocol enhanced by Intusoft to interface external applications to the IsSpice4 simulator. It exposes all of the interactive functionality of IsSpice4. With OLE, the ICL interface is distilled into a form that makes it easy to access all of the capabilities of SPICE. Additional OLE interfaces to control the IntuScope post processor and to implement schematic cross-probing features can also be obtained from the IsSpice4 OLE interface.

In order to demonstrate IsSpice4’s OLE interface, we will use a Visual Basic (VB) Script (Figure 11) to start IsSpice4 and remotely run Transient and Fourier analyses. In this case, the script is run from the Microsoft Excel spreadsheet program. The simulation uses values taken from the spreadsheet. The IsSpice4 results are returned to the spreadsheet.

For i = 2 To 4 Step 1

' The format is Object.property value for example SPICE.DoScript '

' DoScript is a method for running an ICL command. '

' The ScriptOutput property gets the IsSpice4 output window contents '

SPICE.DoScript "setplot tran" ' sets the current analysis type to Tran '

strAlter = "alter @Rf[resistance]= " + Worksheets("Data").Range("a" + CStr(i)).Value

' Change the value of resistor RF based on the worksheet using the range Column A Row i where i is a variable '

SPICE.DoScript strAlter

SPICE.DoScript "run start"

SPICE.DoScript "sendplot v(3)"

SPICE.DoScript "fourier 10k v(3)"

strOutput = SPICE.ScriptOutput

pos1 = InStr(strOutput, "THD:") ' Parses output from IsSpice4 '

pos2 = InStr(strOutput, "%")

strTHD = Mid(strOutput, (pos1 + 4), pos2 - (pos1 + 4))

Worksheets("Results").Range("b" + CStr(i)).Value = strTHD

' Assign value obtained from IsSpice4 to Column B row i '

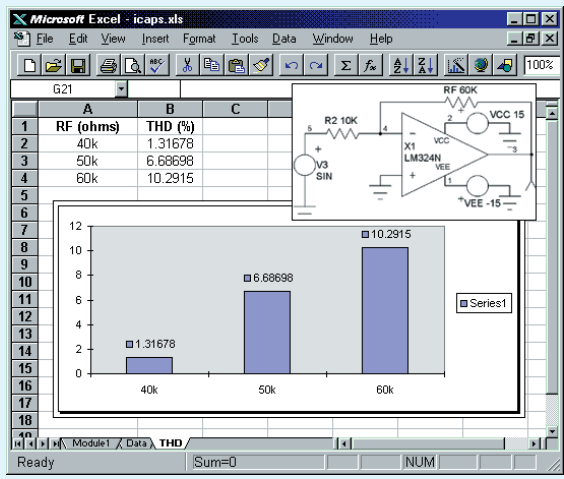
Next i

Figure 11, Visual Basic script fragment that drives IsSpice4 from Excel. More information and the full script are available from Intusoft.

The SPICE.DoScript command sends ICL functions to IsSpice4. Several ICL statements can be concatenated into one DoScript or sent individually. In the above VB script, 3 transient simulations are run. For each analysis the value of the resistor (RF) is set based on the Excel spreadsheet (column A, rows 2:4) values using the strAlter= line. The total harmonic distortion results of a Fourier analysis are returned in SPICE.ScriptOutput and displayed in column B, rows 2:4. Many other more complex applications are possible.

IsSpice4 is the ONLY SPICE simulator to offer OLE and Visual Basic interfaces. It is just another way in which Intusoft distinguishes itself from other EDA tool vendors.

Figure 12, A Visual Basic script, shown in Figure 11, is used to interface an Excel spreadsheet with IsSpice4. The spreadsheet shows the value of resistor RF and the resulting harmonic distortion of the amplifier circuit (inset). The VB script passes the RF values, runs the simulation, and returns the results. A graph of the THD is also shown.



Mixed Mode Modeling: Digital Pots

Intusoft is the ONLY SPICE vendor that makes SPICE models FREE of charge for its registered users. Many of the models shown in the newsletter were motivated by such requests. It is an extremely valuable service that really distinguishes Intusoft from the competition. Some of the requested models are simple, while others require a great deal of research.

Recently, Intusoft's technical support was asked to model the Xicor X931x digital potentiometer line (Figure 13). The X931x, and similar X9C10x line, are solid-state nonvolatile potentiometers that are ideal for digitally controlled resistance trimming. They use digital circuitry to control a series of electronic switches that connect to a resistor array. It turned out to be an interesting modeling task, especially in light of the model's mixed mode nature.

Model Workings

The resistor ladder is emulated by 2 voltage controlled resistors, R_{top} and R_{bot} (Figures 13 & 14). Their values are controlled by a voltage, V_{pos} , equal to the output of the $Bdtoa$ element in the $Analog_State$ subcircuit. R_{tot} is the total resistance and $TAPS$ is the total number of taps. The $1U$ is used to keep the resistance from being set to 0 when the wiper hits its limits. SPICE does not allow resistors to have a zero value; instead it sets them to $1k\Omega$.

A state machine (Astate line in $Analog_State$) model, unique to Intusoft's $IsSpice4$ program, is used to model the digital logic of the memory, up/down counter, and the one-of-100 decoder. The state machine transitions to the previous or next state depending on the status of $V_{up/dn}$ when V_{inc} transitions low.

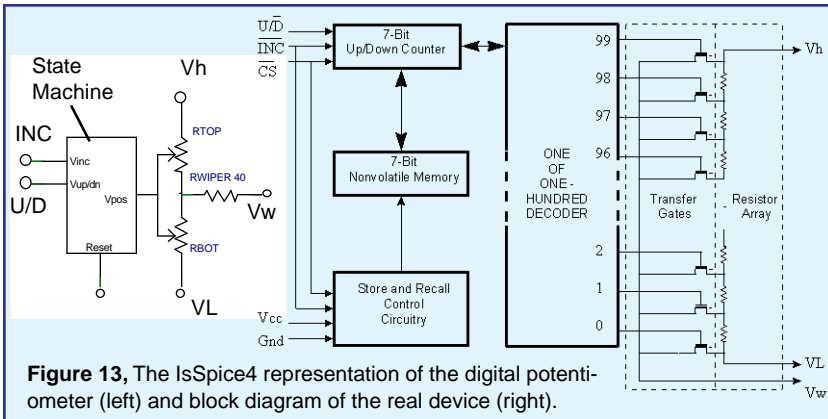


Figure 13, The $IsSpice4$ representation of the digital potentiometer (left) and block diagram of the real device (right).

```

.SUBCKT X9312Z 1 3 2 Vupdn Vinc Reset {Rtot=1K TAPS=100}
*Connections Top Wiper Bottom Vupdn Vinc Reset
XA Vinc Vupdn VPos Reset Analog_State
RTop 1 30 R= {Rtot} * ((TAPS-1) - V(Vpos)) / {TAPS-1} +1U
RBot 30 2 R= {Rtot} * (V(Vpos) / {TAPS-1}) + 1U
RWiper 30 3 40 ; Comment out and set node 3 to 30 to remove wiper resistance
.ENDS
*****
.SUBCKT Analog_State 8 9 Vout 92
*Connections Vinc Vupdn VPos Reset
*DEFINE /Vinit=0 ; Define can be moved to the main circuit for easier access
Bdtoa Vout 0 V=V(1)+V(2)*2+V(3)*4+V(4)*8+V(5)*16+ V(6)*32+V(7)*64
AState [90] ~80 91 [10 20 30 40 50 60 70] Counter
*Connections Vupdn Vinc Reset d0 d1 d2 d3 d4 d5 d6
.Model Counter d_state(clk_delay=1u reset_delay=100n
+ state_file=bcnt7.txt reset_state= /Vinit)

Ad2a [10 20 30 40 50 60 70] [1 2 3 4 5 6 7] D2A
.Model D2A dac_bridge(out_low = 0 out_high = 1 out_undef = .5
+ t_rise = 1n t_fall = 1n)
Aa2d [8 9 92] [80 90 91] A2D
.Model A2D adc_bridge(in_low = 0.1 in_high = 0.9
+ rise_delay=1n fall_delay=1n)
.ENDS

```

Figure 14, Mixed mode model for a digital potentiometer using a state machine.

Its output is used to keep track of the wiper position. The 7 bits of the state machine are weighted and summed in the Bdtoa behavioral element as the voltage Vout (=Vpos).

The first and last states (0 and 99) are set up so that the output can not move below 0 or above 99. A ~ in front of a node number inverts the digital signal. The AtoD and DtoA bridge elements connect the analog and digital signals. They convert between continuous time voltages and discrete digital states.

The only functions not modeled are the ability to move the wiper several positions (shorting out several resistor taps) before the Wiper is finally set, the full Chip Select operation, and the power on/off operation. The model changes on each clock pulse. To force the model to change state at another interval (T), set `clk_delay=T` in the `d_state` model in `Analog_State`. The `*DE-`

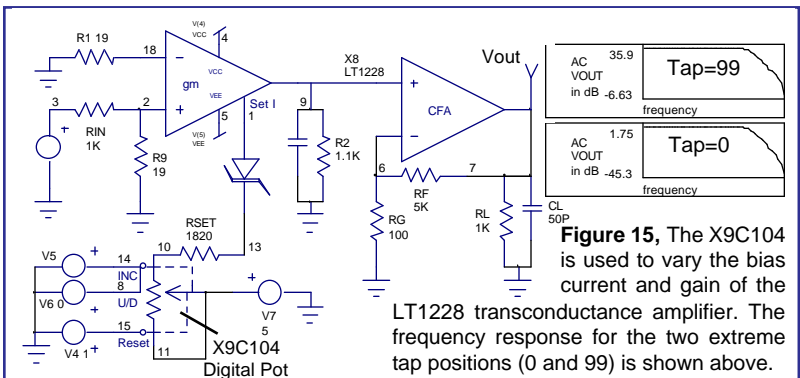


Figure 15, The X9C104 is used to vary the bias current and gain of the LT1228 transconductance amplifier. The frequency response for the two extreme tap positions (0 and 99) is shown above.

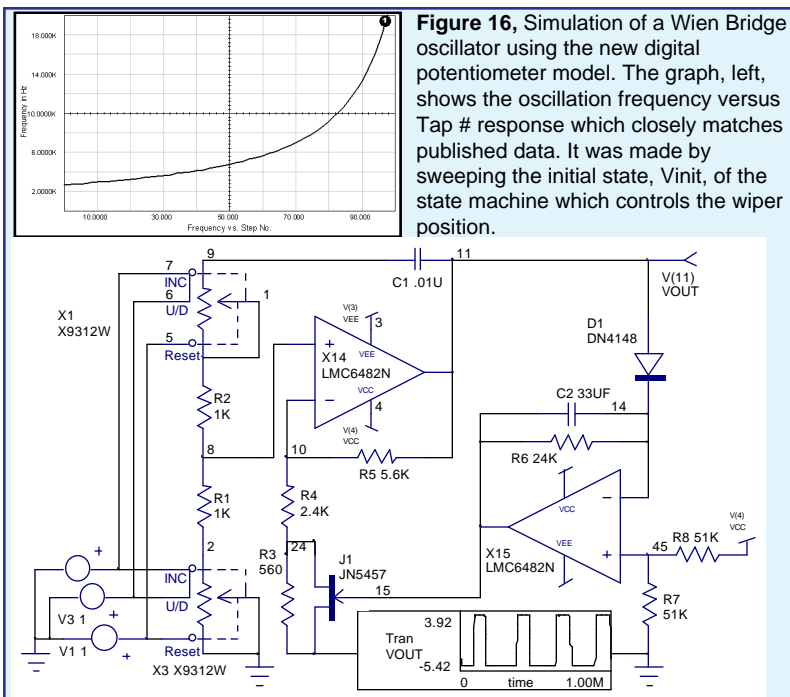


Figure 16, Simulation of a Wien Bridge oscillator using the new digital potentiometer model. The graph, left, shows the oscillation frequency versus Tap # response which closely matches published data. It was made by sweeping the initial state, Vinit, of the state machine which controls the wiper position.

FINE /Vinit=0 in the Analog_State subcircuit defines the initial state of the potentiometer as well as the potentiometer state when Reset goes low.

Two example circuits are shown. In Figure 15, the digital potentiometer is used to vary the bias current, and hence the gain, of a transconductance amplifier. In Figure 16, a variable frequency Wien bridge oscillator is investigated.

New SPICE CD-ROM Available



Intusoft has released a CD-ROM with a variety of useful information for **ALL SPICE users**. The FREE CD contains SPICE models, applications notes on how to model different electronic devices, technical articles on how to simulate various types of designs, and over a dozen issues of the popular



Evaluation versions of Intusoft's analog and mixed signal design tools, magnetics design tools, filter synthesis tools, and SPICE modeling tools are included along with a beginner's tutorial guide to **Getting Started With SPICE**.

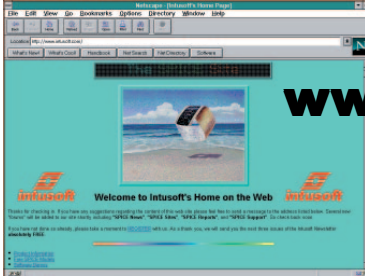


A wealth of valuable technical information is included, such as our paper on "Solving SPICE Convergence Problems". The CD is perfect for engineers looking for guidance on using any SPICE program, not just Intusoft's offerings.

Europe Gets Intusoft Mirror Site

A new mirror site for Intusoft's Home page has been set up by the official Intusoft dealer in the U.K., Technology Sources.

Those European based engineers who are interested in Intusoft's wealth of SPICE related information and product evaluation versions can download them more easily from this site.



www.softsim.com

New European
SPICE Site on the Web

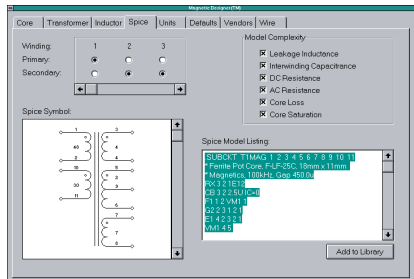
Magnetics Designer Makes EDN Top 100

Magnetics Designer was named by EDN Magazine as one of **1996's Top 100 Hot Products**. The survey is published in the December 5 1996 issue.

Magnetics Designer synthesizes and analyzes various types of transformers, inductors, and chokes. Typical designs may include ferrite pot, EE, EP, cut C, planar and RM cores, toroids, and EI laminated cores. Typical

applications include 60 Hz line transformers, switching regulator transformers, and output filter inductors. The program produces reports and a SPICE model along with the finished design. A database that you can add to and modify with thousands of cores is included with the program.

Magnetics Design Made Easy!



Updated Core Database Available For FREE

An updated Magnetics Designer database of core geometries and materials is available from the Intusoft Web Site (www.intusoft.com). Ferrite Toroids from Magnetics and EP cores from Philips have been added. This update is part of Intusoft's Free technical support services. Core database updates will be made on a regular basis. If you would like to see a particular set of cores added, please contact Intusoft tech support. You can also contact your local core supplier. Intusoft is happy to provide your core supplier with all the materials needed to make database additions.

WHAT MAKES IsSPICE4 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 nonproprietary HDL using C
- Develop your own OLE interfaces and user defined algorithms
- Support for nonelectrical applications and top-down system design
- Powerful Behavioral Modeling
- Three types of digital/mixed mode modeling
- Lossy (distributed) transmission lines with frequency dependent losses
- MOS: BSIM1, BSIM2, and New BSIM3 version3, SOI MOSFET

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

- True OLE Integration with 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.

New SMPS Book Available

Intusoft is pleased to announce the availability of an important NEW reference book. The book, entitled “**SMPS Simulation With SPICE 3**”, is published by McGraw-Hill. It is authored by Steve Sandler, a well known authority in the power electronics and simulation arena. The book is a **MUST HAVE for every power circuit designer** who is using or thinking of using SPICE. It includes information on modeling, simulation, and circuit design techniques. Topics include:

EMI Filter Design: Using SPICE to calculate harmonic content

Buck-Flyback Topology Converters: Average vs. Transient Modeling for Voltage/Current Mode control

Linear Regulators: Control Loop Stability

DC-AC Conversions: State machine modeling, Sine ROMs

Improving Simulation Speed: SPICE Tips for Power Supply Designers

A chapter from renowned magnetics expert Rudy Severns on **Modeling Magnetics** (inductors, transformers, nonlinear core models, and reluctance modeling) is also included. The book is priced at \$55 and is available from Intusoft or your local Intusoft dealer after December 16, 1996.

