Intusoft Newsletter

Personal Computer Circuit & System Design Tools

intusoft

Copyright © Intusoft, All Rights Reserved
Issue #45 Feb. 1996
Tel. (310) 833-0710
Fax (310) 833-9658

The Intusoft Difference

ICAP/4Lite users take note. There is a new year's **Upgrade Sale** just for you. Intusoft is making it just about impossible not to upgrade to the full power of ICAP/4Windows. Right now, until April 15, 1996, you can upgrade ICAP/4Lite to ICAP/4Windows for only **$995, a savings of over $1000** (normal upgrade price is $2000). Some vendors make a big deal when they charge you the price

Continued on pg. 9

In This Issue

2 SALT Connects Hardware & SPICE

2 Intusoft Links with OrCAD™ & Protel™

6 Modeling A TRIAC New SpiceMod Version

9 New IsSpice4 v7.6

10 New Magnetics Design

11 A Real Cap Model, pII

New OLE2 SPICE Links To Hardware

SALT, or SPICE Array Linking Technology is a new and unique SPICE modeling product. It contains two components; a series of models that use OLE (**O**bject

Continued on pg. 2

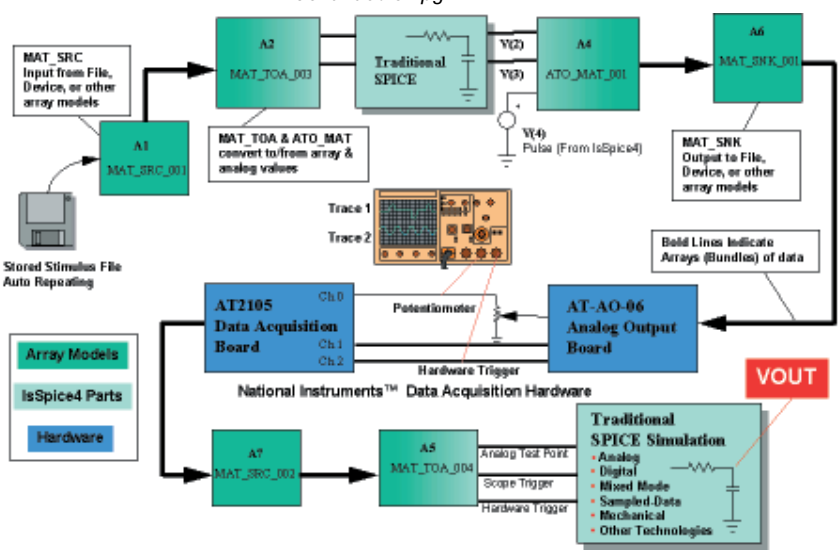


Figure 1, The new **SALT Kit** opens the world of SPICE simulation to hardware. Above, SALT interfaces IsSpice4 to files, arrays, and data acquisition boards.

SPICE ARRAY LINKING TECHNOLOGY

Linking and **E**mbedding) methods to provide various input, output and array processing functions and a methodology for interfacing your own models to IsSpice4. The models you make can be tailored to specialized applications.

At Wescon/95 Intusoft demonstrated a unique "Virtual Test Bench" consisting of IsSpice4 and National Instruments™ data acquisition and output boards. Shown in Figure 1, the demonstration provides a general example of the new file and HARDWARE INTERFACES to SPICE enabled by SALT. This is a unique capability developed by Intusoft and represents a MAJOR breakthrough in hardware test and measurement.

To start the simulation, data is read from a stored file and converted to an analog signal. The format is similar to a PWL waveform, but it may be set up to auto-repeat. After being processed by IsSpice4, the data is sent out of the PC to the test bench using a National Instruments analog output board. The waveform is monitored by a scope and scaled with a 50kohm pot. The altered waveform is then read in by a data acquisition board and further processed by traditional SPICE methods.

As you can see, SALT models can be used to "read in" real world data for use as stimulus or generate test waveforms using a simulation of a circuit too expensive to actually build. The possibilities for connecting SPICE to the real world are endless.

SALT provides a set of ready-to-use models for array processing algorithms (wavelet transforms, FFTs, matrix math), models for file and sound input/output, and the capability to link user-defined models to IsSpice4. Source code for the pre-defined models is included. SALT will be available on March 15, 1995. It is priced at $495 and requires IsSpice4.

Intusoft Links to OrCAD And Protel

Integration is one of the key issues in the Windows EDA tool market. And rightly so. The overall performance of even the best individual CAE tools can be impaired by poor integration. This is especially true for schematic entry and simulation. Intusoft is addressing the third-party integration issue by becoming the first vendor to **introduce an OLE automation server version of SPICE**. This is significant as it will allow IsSpice4 to be transparently interfaced with other applications much more easily than any other SPICE simulator.

To complement the OLE SPICE, Intusoft has created OrCAD and Protel symbol libraries. Included are symbols for all of Intusoft's 8000+ SPICE models. They can be used with OrCAD SDT 386+™ and Capture™ and Protel's Schematic³™. The symbols are posted on Intusoft's Web site and CompuServe forum. They are available for FREE. While a short tutorial on using IsSpice4 with OrCAD is included here, more complete tutorials for OrCAD and Protel are posted with the symbols.

What OLE Means to You

IsSpice4 exposes its simulation capabilities (i.e. the ability to sweep parameters and analyze a circuit) as OLE automation objects. OLE allows another program, called an OLE Automation Controller, to access the exposed objects and use their properties and methods. The automation interface to IsSpice4 follows a format similar to Microsoft Excel. The main interface to IsSpice4 will be through an API (applications programming interface) using OLE. This will allow other software like schematic entry programs, mathematical engines like MATLAB, and user defined software programs like those written in Visual Basic, to utilize IsSpice4 as a calculation engine.

The IsSpice4 API is more advanced than any available at this time. In addition to waveform cross-probing, OLE and the API will provide complete control and access to the IsSpice4 simulator. API functions include the ability to launch, stop, pause, and specify analyses from another application, plus over 60 other functions (to alter device/model parameter values, get simulation data back DURING the simulation, interactively plot waveforms on the client application, etc.). An

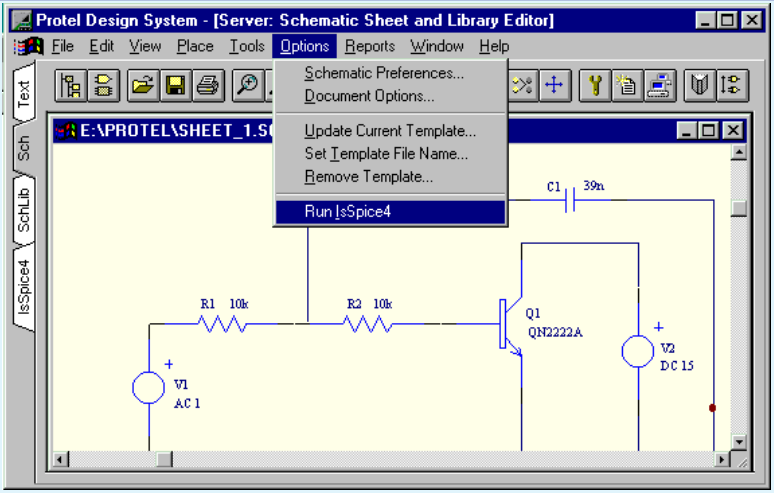


Figure 2, With the OLE interface IsSpice4 can be integrated with third party schematics like Protel's Schematic³.

example of some simple OLE integration between IsSpice4 and Protel is described next.

IsSpice4 and Protel

Protel's Schematic[3] Client-Server interface makes integrating tools from different vendors easy. It does this by allowing free access to new menu commands and by exposing a Basic derived scripting language called Client Basic. Together these features allow a Basic script to be assigned to a menu command as shown in Figure 2. Protel already generates an IsSpice4 netlist. By using the OLE objects exposed by IsSpice4, we were able to further link Protel and IsSpice4 and launch a simulation directly from the schematic.

Using IsSpice4 with OrCAD Capture

A SPICE netlist contains a description of the circuit and control statements that tell the simulator what to do and what to output. The following guide outlines the basic steps required to produce such a netlist for IsSpice4 with OrCAD's Capture program. Similar steps are required by Protel Schematic[3].

Reference Designation Control

IsSpice4 requires that all parts have unique reference designators (ref-des). A ref-des in IsSpice4 is equivalent to the Part Reference in Capture. By default, Capture appends a "?" to the Part Reference to signify that it is a newly placed part. After

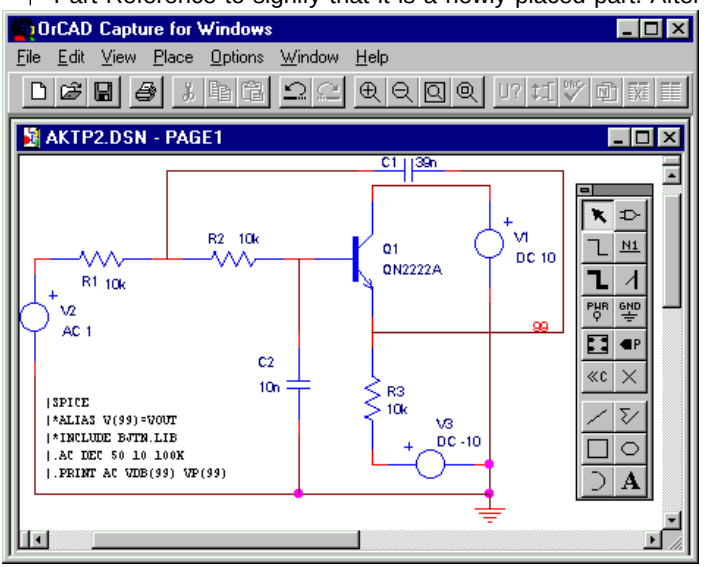


Figure 3, OrCAD Capture can be used to create an IsSpice4 netlist for simulation.

entering your schematic, you can use the “Update Part References” function from the Tools menu to make each part’s ref-des unique. Make sure the “Update Entire Design” and “Incremental reference update” options are checked.

SPICE Control Statements

Control statements (i.e. .TRAN, .PRINT, .OPTIONS) are entered using the Capture Text command located under the Place menu. In the dialog that is provided, enter “|SPICE” as the first line. You can then enter other control statements on separate lines, preceded by a pipe “|” character. As an example, the following control statements on the left are generated by the entries on the right (Figure 3).

∗INCLUDE BJTN.LIB	|SPICE
.AC DEC 50 10 100K	|∗INCLUDE BJTN.LIB
.PRINT AC VDB(99) VP(99)	|.AC DEC 50 10 100K
	|.PRINT AC VDB(99) VP(99)

Model/Symbol Resolution and Output

SPICE simulators require that the input netlist include all of the .MODEL and .SUBCKT statements before a simulation can proceed. To facilitate this, IsSpice4 automatically searches library files containing the models and appends the requested information to your netlist. To do this, the SPICE netlist must have the proper “∗INCLUDE” statements to point to the libraries to search. Intusoft provides a set of symbol libraries whose name corresponds to the SPICE model library where the model can be found. For example, a symbol from BJTN.OLB requires a ∗INCLUDE BJTN.LIB statement in the netlist.

Generating Output is the trickiest part of the process. Capture produces a completely new SPICE netlist each time the “Create Netlist” function is selected. This means that node numbers may be different each time. To alleviate this problem, you can lock a net name using the “Net Alias” command under the Place menu. By entering a number for the net alias, you can lock the name to any desired number. A .PRINT statement can then be used without it ever needing to be updated. Note: to avoid this situation, you can issue the “SAVE ALL” statement to save all of the circuit data, but this can use much more memory.

Creating The Netlist

An IsSpice4 netlist is generated using the “Create Netlist” function under the Tools menu. This function is only active if the schematic has been saved and the Design Page is active. The “Create Netlist” dialog is used to configure the netlist entries. The part value field determines the items that are placed after the node numbers. Generally this will be {Value} {PARAMS}. The PARAMS field is used to implement the parameter passing

Continued on pg. 13

Modeling A Triac

By Slim Petrie, AP Laboratories

A triac is a bi-lateral switch that can be triggered into conduction regardless of its polarity. It is modeled by using two NPN/PNP transistor pairs connected back-to-back as shown on the left of Figure 4. The base of each transistor is connected to the collector of the other. This produces positive feedback, resulting in the required switching action. At MT2, the emitter metallization overlaps the base of the NPN transistor. This forms a lateral resistor in the base (P) region and is shown as RH in the model. RH determines the holding current of the triac. A similar resistor exists at MT1. These resistors hold the triac off unless triggered by the gate or the holding current from a previous “on” condition.

Although this configuration has the basic triggering function of the triac, it must be enhanced to emulate other important parameters such as off state leakage, breakover voltage and current, and voltage and current characteristics in all four modes of operation. Figure 4 shows the full triac subcircuit. Resistors and zener diodes are used to simulate the breakdown voltages and leakage currents. Dependent sources are used to emulate the various triggering modes and to allow a wider range of trigger data to be entered.

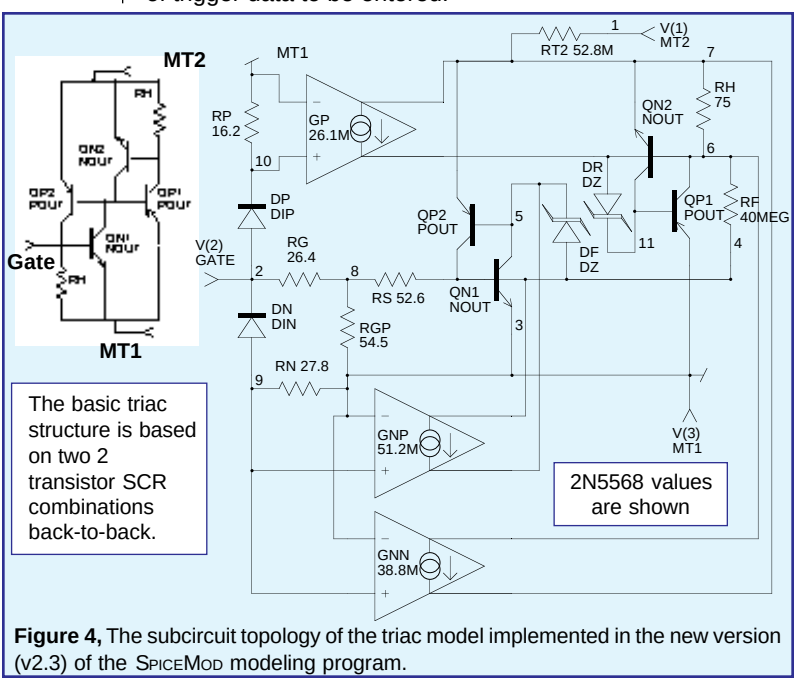


Figure 4, The subcircuit topology of the triac model implemented in the new version (v2.3) of the SpiceMod modeling program.

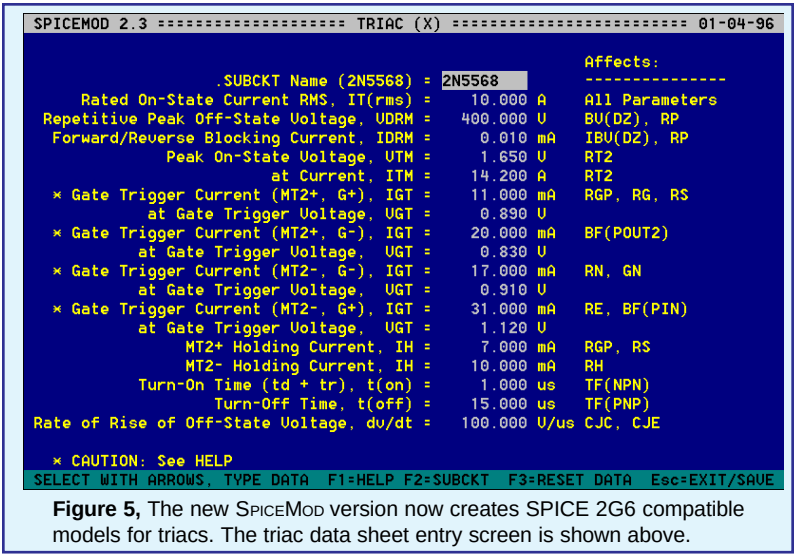


Figure 5, The new SpiceMod version now creates SPICE 2G6 compatible models for triacs. The triac data sheet entry screen is shown above.

If adjusting the model parameters for the triac subcircuit seems a little daunting, you can use SpiceMod. SpiceMod is a software program that quickly converts data book parameters into SPICE model parameters. Entering only the device type and maximum voltage and current ratings will produce a realistic model as all other parameters are scaled from them. Naturally, the more data you enter, the more exact the model will be. Figure 5 shows the triac entry screen from SpiceMod.

Data books may show the same values for all four gate trigger current and voltage modes. However, the values are different in all four modes. In a real triac, when the gate is driven with the opposite polarity from the MT2 terminal, the gate must supply enough current to drive the shunting resistance to twice the gate voltage, thus the trigger current cannot be the same for both polarities, regardless of what the data book limits say. G+ with MT2- always requires more gate drive than other modes. In this case, it is only necessary to enter the first mode gate voltage and current and let the program estimate the others for the most realistic model. SpiceMod guards against entering grossly unrealistic data and will attempt to produce the closest realistic model in these situations.

The SPICE 2G compatible netlist for the triac is shown in Table 1 (next page). Along with the “Affects” column in Figure 5 you can get an idea of the relationship between which data sheet parameters correspond to which SPICE model parameters.

Triacs have two stable states. Therefore, it may be necessary to tell SPICE which state to use, especially when you want to

start a simulation with the triac in the off state. To do this you can issue the “OFF” keyword on the subcircuit transistor lines.

Figure 6 shows the low current region of a Motorola 2N5568 triac with no input applied to the gate. This curve was generated by holding the current to the gate at zero and sweeping the MT2 current from -50mA to +50mA. The slope of the curve at the zero axis is determined by the IDRM specification. The maximum voltage swings in the off state are determined by the VDRM specification. The trigger points (+6 and -10mA) are determined by the holding current.

Simplified behavioral triac models are included with IsSpice4. Their superior simulation speed and idealized response can make them useful for investigating triac control circuitry. However, with power devices such as the triac, you will normally need models that exhibit 2nd order effects as well. Without them you can’t run realistic simulations. Most other SPICE vendors do not offer such sophisticated power semiconductor models. Fortunately, Intusoft does.

New SpiceMod Version Adds Triacs

SpiceMod v2.3 models diodes, Bjts, Jfets, Mosfets, SCRs, IGBTs, Power Mos, Power BJTs, and Darlington BJTs and now models zeners and triacs. SpiceMod is compatible with all SPICE simulators and is available immediately for $300. Upgrades are $50. Thanks to the author and developer of SpiceMod, Slim Petrie, 7 W. Lillian Ave., Arlington Heights, IL 60004.

```
.SUBCKT 2N5568   1  2  3
*    TERMINALS: MT2 G MT1 Mot. 400V 10A
QN1  5 4 3  NOUT          ; Output Transistors
QN2 11 6 7  NOUT          ; “OFF” keyword for Q devices
QP1 6 11 3  POUT
QP2  4 5 7  POUT
DF  4  5  DZ              ; Forward breakdown diode
DR  6 11  DZ              ; Reverse breakdown diode
RF  4 6  40MEG            ; Forward leakage current (controls IDRM).
RT2 1 7  52.8M            ; Controls “on” resistance
RH  7 6  75               ; Controls reverse holding current
RGP 8 3  54.5             ; Controls forward holding current and trigger current
RG  2 8  26.4             ; with RGP controls VGT
RS  8 4  52.6             ; with RGP controls forward holding current
DN  9 2  DIN              ; Diode to isolate G- triggering modes
RN  9 3  27.8             ; Controls current in G- trigger modes
GNN  6 7 9 3 38.8M        ; Controls G-, MT2- trigger voltages
GNP  4 5 9 3 51.2M        ; Controls G-, MT2+ trigger voltages
DP 2 10  DIP              ; Diode to isolate G+, MT2- trigger mode
RP 10 3  16.2             ; Controls current in G+, MT2- trigger mode
GP  7 6 10 3 26.1M        ; Controls G+, MT2- trigger voltage
.MODEL DIN  D (IS=53.5F) ; Conducts in G- modes
.MODEL DIP  D (IS=53.5F N=1.19) ; Higher drop diode conducts only in G+, MT2- mode
.MODEL DZ   D (IS=53.5F N=1.5 IBV=10U BV=400)
.MODEL POUT PNP (IS=53.5F BF=5 CJE=235P TF=25.5U)
.MODEL NOUT NPN (IS=53.5F BF=20 CJE=235P CJC=46.9P TF=1.7U)
.ENDS
```

Table 1, SPICE 2G Triac model generated by the SpiceMod modeling program from manufacturer’s data sheet parameters.

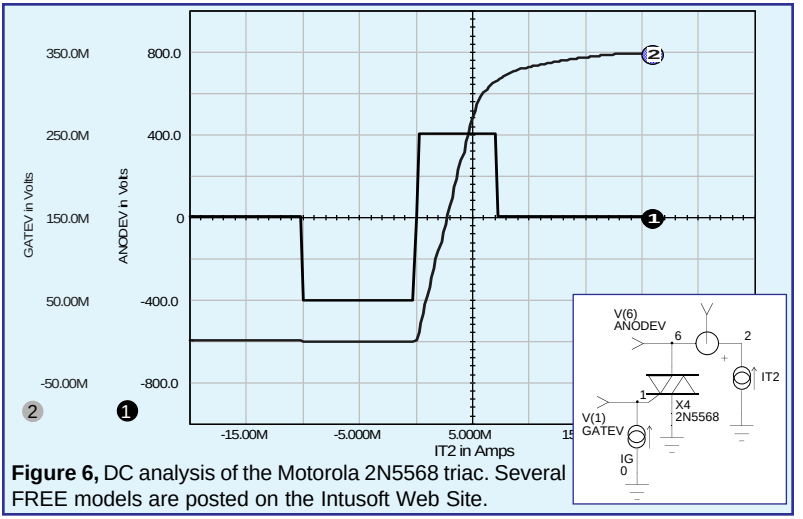


Figure 6, DC analysis of the Motorola 2N5568 triac. Several FREE models are posted on the Intusoft Web Site.

Big Savings On ICAP/4Lite Upgrades

Continued from pg. 1

difference for an upgrade. At Intusoft, its just standard policy. With Intusoft your investment in software lasts forever! Unlike our competitors, upgrades are ALWAYS the price difference or LESS with no time limits. And we're ready to back up our words. In this case, its **50% less**. You must have purchased your ICAP/4Lite system as of Feb. 1, 1996. Some of the features you can look forward to: waveform cross-probing, full native mixed mode simulation, over 6500 more SPICE models, powerful post processing, and more. Take advantage of this unbeatable offer before time runs out!

New IsSpice 7.6 To Be Released

The new version of IsSpice4 will be released on February 29, 1996. The simulator has been enhanced with the following features:

- **New behavioral modeling constructs**
- **Lossy line with frequency dependence (skin effect/dielectric loss)**
- **R, L, C, B expressions can use frequency, time and temperature**
- **B element expressions a function of device currents in the time and frequency domain**
- **More analog and digital models (now at over 8000)**
- **Enhancements to the Interactive Command Scripting language**

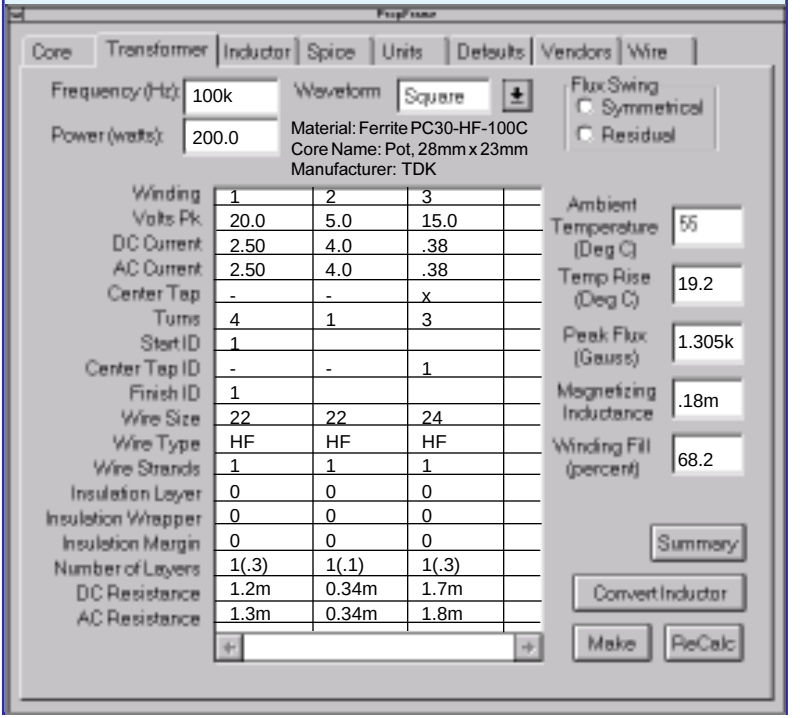
Upgrades from version 7.51 will be $200. Contact Intusoft for information on pricing for other versions. The basic price of the ICAP/4Windows package will remain the same.

New Magnetics Design Software

Intusoft will shortly be releasing a NEW nonlinear magnetics design program. **Magnetics Designer** for Windows will debut in March at APEC 1996 (San Jose). The original **Magnetics Designer** software was acquired by Intusoft from **Analytic Artistry**™. Intusoft is porting the package to Windows and enhancing it with a variety of new and unique features.

Magnetics Designer synthesizes and analyzes various types of transformers, inductors, and chokes from either electrical or geometrical specifications. **Magnetics Designer** will produce a "winding sheet" that you can send out to a transformer manufacturer. (You will even be able to email the data directly from the program to your local manufacturer!!) And most notably, **Magnetics Designer** will also produce a SPICE model, directly compatible with IsSpice4, that is ready to simulate! Core manufacturers from around the world will be represented in the program. Complete information on the release date and price of **Magnetics Designer** will be available in the March *Intusoft Newsletter*.

The Transformer Design Screen in the new Magnetics Designer program



Modeling Corner

The Intusoft

In this edition of the modeling corner, we present models from Kemet Electronics Corp. and Intusoft Tech support. The *Intusoft Newsletter* subscription disk contains new pressure sensor models and RF BJT models from Motorola, and over 200 new triac models from several different vendors. Part 2 of our capacitor modeling article, continued from the November 1995 issue, is shown next.

Modeling A Tantulum Capacitor

In part 1, we covered simulation of ceramic capacitors. A model for tantulum capacitors is discussed here. Unlike the ceramic model, the tantulum requires a capacitance that varies with frequency. The capacitance of a ceramic device may decay by 1% per decade of frequency. This is insignificant in modeling the performance and is disregarded. However, the tantalum can decay 20% or more per decade. We could utilize a frequency dependent capacitance expression if one could be found, but the capacitance is most easily characterized with an RC ladder network (Figure 7).

The ladder has consistent resistive elements between increasing capacitive elements as the depth increases in the ladder. Use of actual elements, as opposed to an expression using the AC analysis variable FREQ, has the added benefit of working in the time domain. The RS element is temperature and frequency dependent, while the RC elements are simply temperature dependent. The 5 capacitive elements are sufficient for duplicating the actual performance in terms of capacitance roll-off, impedance and ESR with frequency.

This electrical presentation of the tantalum capacitor fits with the understanding of the physical aspects of the capacitor; the capacitance decays with increasing frequency as less of the signal penetrates the depth of the anode bulk. The capacitance available to the circuit at higher frequencies is all located near the outer surface of the anode.

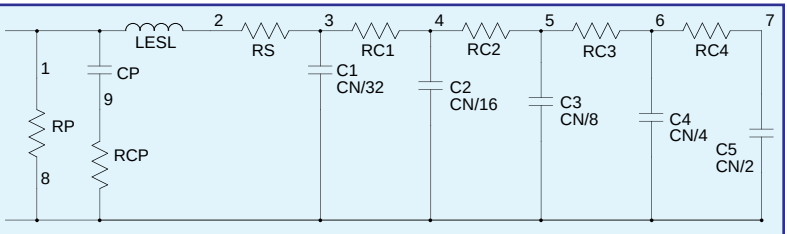


Figure 7, A tantulum capacitor model. The model is made up of elements that are frequency (RS), and temperature (RC) dependent.

```
.SUBCKT T491 1 8 {Esl=1n R=.06 Fx=3 Cnom=1u Temp=25} ; Defaul Parameter Values
*Cn=Nominal Cap/ (1/2+1/4+1/8+1/16+1/32) = Cnom*1.032258065
*Rt = R * 4^((25-TEMP)/100), = Rc (@25½C)
RP 1 8 {1000/Cnom} ; 1000ý-farad/Cap (in farads)
CP 1 9 {100 * Cnom*1.03226} ; 100*Cn
RCP 8 9 {R * 1K} ; typical parallel resonant RC, ³ 1000 * Rnom
LESL 1 2 {Esl} ; typical effective series inductance
RS 2 3 R={R * 4^((25-TEMP)/100)} + {R * 4^((25-TEMP)/100)} *
+ (10^({Fx} - log(Freq)) + 10^(log(Freq)-7))}
* Esr = Rt + Rt * (10^(Fx - log(Freq)) + 10^(log(Freq)-7))
C1 3 8 {Cnom*1.03226/32} ; Cn/32
RC1 3 4 {R * 4^((25-TEMP)/100)}
C2 4 8 {Cnom*1.03226/16} ; Cn/16
RC2 4 5 {R * 4^((25-TEMP)/100)}
C3 5 8 {Cnom*1.03226/8} ; Cn/8
RC3 5 6 {R * 4^((25-TEMP)/100)}
C4 6 8 {Cnom*1.03226/4} ; Cn/4
RC4 6 7 {R * 4^((25-TEMP)/100)}
C5 7 8 {Cnom*1.03226/2} ; Cn/2
.ENDS
```

Figure 8, Generic IsSpice4 netlist for the realistic tantulum capacitor model. The ESR is made frequency dependent using a new feature in IsSpice4 that allows frequency dependent resistors to be modeled.

With the tantalum capacitor, there is no noted effect of DC biases while temperature affects the ESR and capacitance. The relationship for the ESR is similar to that of the ceramic capacitor. The variation in the frequency response is related to the size and materials of the anode.

The parallel resistor, RP, represents the leakage or insulation resistance. RCP and CP create a secondary, or parallel, resonance that represents a capacitance that shunts the RLC elements. CP is dependent on the body and chip size. A generic tantulum capacitor subcircuit netlist is shown in Figure 8. A specific model derived from the generic version is shown in Figure 9. Figure 10 shows the impedance and ESR responses from 100Hz to 100kHz.

The required capacitor parameter values may be obtained from your capacitor manufacturer. Kemet Electronics Corp. (P.O. Box 5928, Greenville, SC 29606, (803) 963-6300) was responsible for the information supplied to Intusoft (Thanks to J.

```
T491D476M010 - Esl=2.5nH R=.06 Fx=3 Cnom=22uF
.SUBCKT T491D476M010  1 8  ; 22µF tantulum capacitor model
RP 1 8 45.455MEG
CP 1 9 2.2710M
RCP 8 9 60.000
LESL 1 2 2.5000N
RS 2 3 R = 60M + 60M * (10^(3  - LOG(FREQ)) + 10^(LOG(FREQ)-7))}
C1 3 8 709.68N
RC 3 4 60.000M
C2 4 8 1.4194U
RC 4 5 60.000M
C3 5 8 2.8387U
RC 5 6 60.000M
C4 6 8 5.6774U
RC 6 7 60.000M
C5 7 8 11.355U
.ENDS
```

Figure 9, IsSpice4 capacitor model for a 22µF tantulum capacitor T491D226M020.

Prymak @ Kemet). Reference 1 and 2 provide further information on the tantulum model.

Models were not provided in the references due to SPICE syntax hurdles. The new IsSpice4 B element alleviates these roadblocks. However, some of the B element syntax extensions used to model the frequency dependence are ONLY available in the newIsSpice4 7.6 release (see page 9). Several models for the T495/T491 families of tantulum capacitors are available on the Intusoft Web site with the complete offering on the newsletter subscribers disk.

[1] "SPICE Modeling of Capacitors", John Prymak, CARTS 95: 15th Capacitor and Resistor Technology Sym., March 1995
[2] Capacitor SPICE Models, Kemet Electronics Corp. Greenville, SC,1994

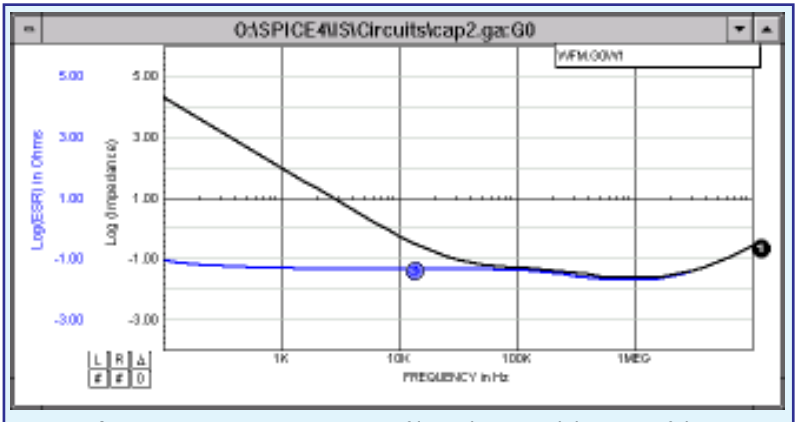


Figure 10, Frequency response of impedance and the ESR of the T491D226M020 22µF tantulum capacitor model.

Using IsSpice4 With OrCAD

Continued from pg. 5

feature of IsSpice which is used with generic models. Both the "Include unconnected pins" and "Use net names" options should be checked. Unconnected pins are connectivity errors and will be flagged by IsSpice4. You may enter any filename with a .CIR extension and check the "View Output" option. Select OK when complete. If there are no errors, the netlist will be created. The View Output option will cause the Intusoft ICAPS program to be started automatically and the .CIR file opened. The circuit can then be simulated.

Most schematic capture programs feature a netlist interface to IsSpice4. With the new OLE features added, Intusoft is in a perfect position to further integrate IsSpice4 with other popular tools. An evaluation kit containing the OLE server version of SPICE and its API is available free of charge to those interested in incorporating IsSpice4 into their product lines.