Intusoft Newsletter

Personal Computer Circuit Design Tools

intusoft (310) 833-0710 Fax (310) 833-9658

NEW PRODUCTS AND UPDATES AT **WESCON/91**

Intusoft will be previewing a variety of new circuit design tools and product updates at the WESCON convention in San Francisco CA on Nov. 19. Two new products will be introduced including SPICENET 3.0M, schematic entry for the Macintosh, and FILTERMASTER, a passive filter design program. Product updates will include SPICENET version 3.0 (PC), SPICEMOD 2.0 (PC), and the new RF Device Library. Other products on display will include the new PRESPICE 3.0 update and the new Intusoft publication "A SPICE COOKBOOK".

RF Device Library Gets Updated

The RF Device Library has met with a tremendous amount of success, as well as a demand for more models. The initial set of models afforded many designers the chance to explore RF and microwave frequency circuits with ISSPICE for the first time. The new update, available immediately, includes several major enhancements. Version 2.0 doubles the size of the original library and contains new models for bipolar transistors, FETs, RF beads, and a MMIC. The total library size now stands at 80 models. The models can be used with any Berkeley SPICE compatible simulator on any computer platform including all versions of ISSPICE.

Annual Newsletter Subscription Now Available

With the next *Intusoft Newsletter*, due in February, Intusoft will begin a subscription service for the popular SPICE publication. The subscription will include delivery of the newsletter (4 issues per year) along with one floppy disk per newsletter. The floppy disk will contain the SPICE models, schematics, and other information described in each application note in the newsletter. Readers may sign up for annual subscriptions immediately or at any time in the future.

For Macintosh users, the NEW SPICENET schematic entry program will eliminate the difficulties of typing and editing SPICE netlists. SPICENET allows the user to draw the electrical schematic in the computer. After the appropriate component values are entered, a special menu assists the user in entering the SPICE control statements. SPICENET then automatically compiles a complete SPICE netlist ready for simulation. All the facets of the SPICE netlist are supported including connectivity, control, options, library access, and subcircuits. Node numbers and reference designations are automatically generated. No editing of the netlist is required at all. In fact, using the simple ACTIONS pull-down menu, an ISSPICE simulation can be run directly from the schematic. Symbol support for thousands of devices is included making symbol creation unnecessary. However, a flexible symbol creation option is always on-line in SPICENET allowing the designer to make symbols "on the fly". The new PC version boasts a similar interface including the ability to edit multiple schematics and multiple pages of a single schematic at the same time. (Figure 1). The printing and plotting capabilities of SPICENET have been enhanced to include support for postscript, HPGL/2, and EPS
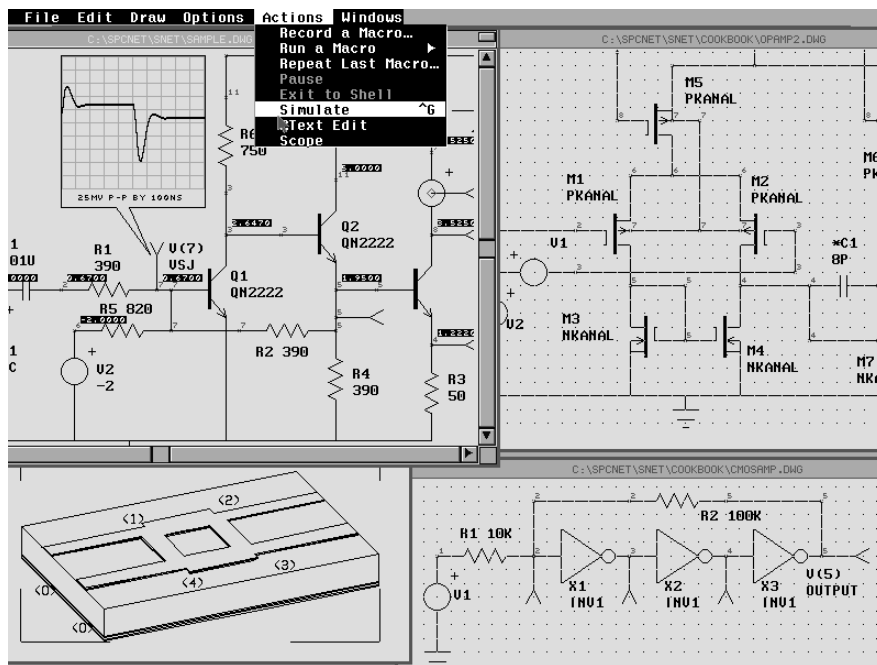

Figure 1, The new SPICENET program for the Macintosh and PC features the ability to edit multiple designs at the same time and run an ISSPICE simulation directly from the schematic.

formats, in addition to the standard HP Laser and Epson/IBM compatible printers and HP/Houston Instruments plotters.

SPICENET is completely integrated with all of Intusoft's simulation tools including ISSPICE, PRESPICE, and INTUSCOPE. The new level of integration combined with the powerful ISSPICE simulation and modeling support makes the ICAPS package solid enough for the most challenging designs and easy enough to use for the engineer who must get the job done with minimal learning time. Both SPICENET for the Macintosh and PC work with any version of SPICE that is Berkeley compatible. They will be introduced and demonstrated at WESCON/91. Information about the official release date and prices for SPICENET will be published in the next newsletter.

FILTERMASTER; New Passive Filter Design Program

Intusoft will formally introduce the new FILTERMASTER program at WESCON. This is the first program in Intusoft's line of circuit synthesis products. FILTERMASTER is a powerful, easy to use, and extremely fast design tool for creating passive LC analog filters.

Available filter types include: low-pass, high-pass, bandpass, and bandstop, which can then be synthesized using Butterworth, Chebyshev, Inverse Chebyshev, Elliptic (Cauer) and Bessel (low-pass only) approximations.

After entering the filter specifications, FILTERMASTER automatically synthesizes the filter structure, calculates the component values, and displays the resulting filter on the screen. Editing options, as shown below, include generating dual/reverse circuits, Norton/
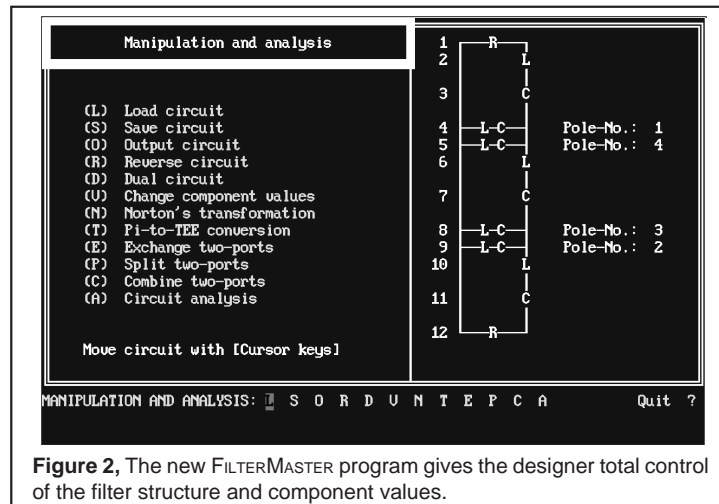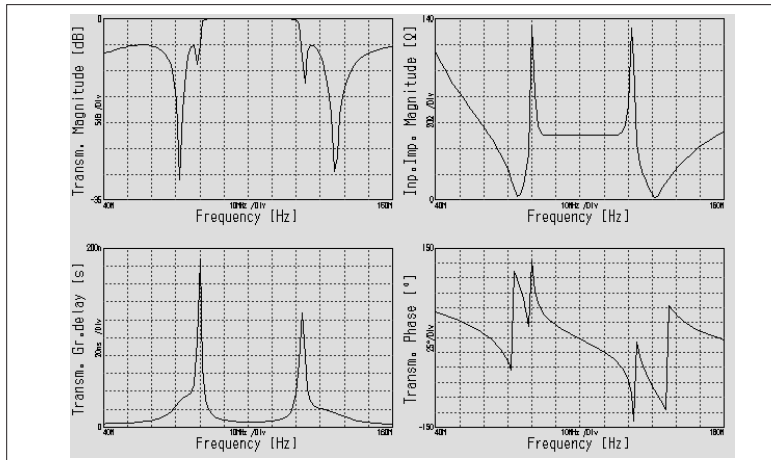

Figure 2, The new FILTERMASTER program gives the designer total control of the filter structure and component values.

Figure 3, FILTERMASTER can quickly analyze a filter providing instant feedback on the transfer characteristics, reflections, impedance, and pulse/step response.

Impedance transformations, Pi-to-Tee conversions, two-port manipulations, and direct manipulation of individual component values.

When the filter is finalized, FILTERMASTER can then analyze and graph the filter transmission characteristics including amplitude, phase, and group delay (Figure 3). Other display options include reflection factors, input/output impedances, and the time domain response using impulse or step stimulus.

Filter Program Links Directly To Schematic

FILTERMASTER includes a special interface to SPICENET and ISSPICE allowing transfer of the synthesized filter directly onto your schematic. A simulation of a complete system, including the filter, can then be performed. FILTERMASTER can create an ISSPICE netlist in a subcircuit or stand-alone format. It also allows insertion of Monte Carlo tolerances on individual or classes of components which can then be used for statistical yield analysis. Unlike other filter design programs, the coupling of ISSPICE and FILTERMASTER allows a total characterization of the filter's performance, as well as the behavior of the filter in the circuit.

FILTERMASTER for the PC will be available on December 9. Hardware requirements include DOS 3.x and up, 640K RAM, and CGA, EGA, VGA or Hercules graphics. A mouse is optional. Support for Epson/IBM dot matrix printers, HP laser printers, and HPGL compatible plotters is included. FILTERMASTER may be run in conjunction with Intusoft's other CAE tools or stand-alone. Demonstration disks are available by contacting Intusoft or using the attached bingo card.

SPICEMOD 2.0 Adds IGBT and SCR Modeling

Intusoft will exhibit version 2.0 of the SPICEMOD, SPICE modeling spreadsheet, program at the upcoming WESCON convention. The new update features the ability to make SPICE models for IGBTs as well as SCRs.

SPICEMOD is an easy to use program that instantly creates accurate SPICE models from manufacturer's data sheet parameters. The new version can model diodes, zeners, bipolar transistors, Darlington transistors, Power transistors, JFETS, MOSFETS, and Power MOSFETs in addition to the newly added IGBT and SCR devices. Availability and the official release date will be announced in the next newsletter.

PRESPICE Update and "COOKBOOK" Now Shipping

The new PRESPICE version 3.0 update is now shipping. The update contains over 1000 new SPICE models, as well as, major revisions to the ISSPICE text editor, IsEd. Along with the currently available vendor supplied op-amp models and the new RF Device Library, the update truly rounds out Intusoft's SPICE model offering which is now at over 1600 models. The models can be used with any SPICE simulator and may be purchased or updated separately from the ISSPICE simulator. All of the models are stored in ASCII files which can be viewed and edited with any text editor. The specific contents of the PRESPICE model libraries, are available by contacting Intusoft.

The new Intusoft SPICE reference publication, "A SPICE COOKBOOK" is also shipping. It contains over 100 examples of practical circuits from different disciplines and how they were simulated with SPICE. It provides an excellent reference source for novice, as well as advanced users of SPICE. "A SPICE COOKBOOK" is available and includes all of the SPICENET schematics, ISSPICE netlists, and SPICE models from the book on floppy disk.

New Intusoft Phone And Fax Numbers

As of November 2, 1991 the area code for telephone numbers in much of Los Angeles county will change to 310. This is due to an increase in phone, fax, and cellular traffic. The new Intusoft phone/FAX numbers as of 11/2/91 will be:

Telephone (310) 833-0710
FAX (310) 833-9658

New Models Simulate RF Circuits

Its no news to those who simulate that the accuracy of SPICE is directly related to the accuracy of the models. What may be news is that simulation of high frequency circuits well into the gigahertz range is now possible due to Intusoft's RF Device Library.

To illustrate the improved simulation accuracy, the RF models were used to analyze a 500 MHz oscillator (Figure 1). The oscillator generates a relatively high power output with a very distorted output waveform (typical at power levels over the 1-2mW range). The simulation goals were to study the start-up characteristics, oscillating frequency and amplitude, and the resulting harmonic distortion. Two simulations were run. The first using the standard Gummel-Poon BJT model and simple inductor chokes and a second using an improved 2N5109 model along with a new Intusoft RF bead model from the RF Device Library.
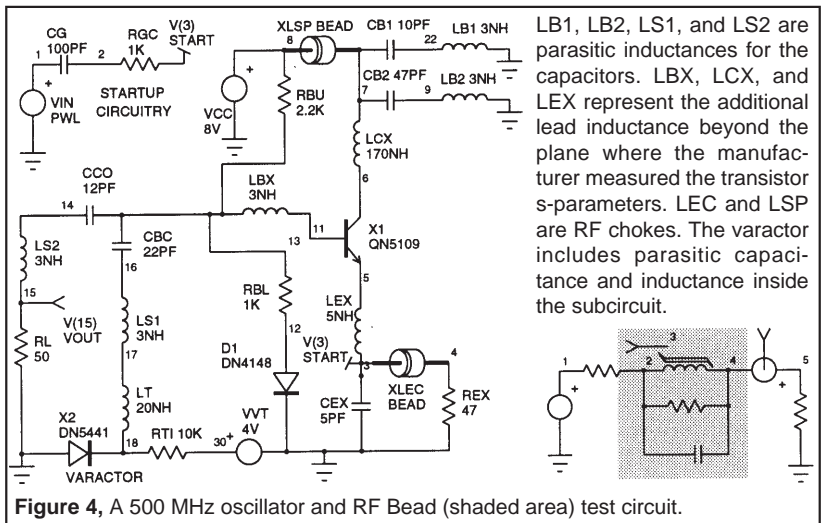
Above approximately 100-200 MHz, the built-in SPICE BJT model, based on the Gummel Poon model, fails to accurately predict the real device performance. The BJT must be remodeled as a subcircuit (Table 1) in order to accurately model the package and bond wire parasitics which are of greater significance at higher frequencies. The improved model includes the package parasitics and matches the s-parameters up to 2 GHz. The RF library was created by Analog & RF Models, specialists in the creation of RF models, (602-575-5323, FAX 602-297-5160) for Intusoft.

Modeling An RF Bead

Although SPICE does contain a model for a nonlinear inductor it does not have a built-in bead model. Fortunately, one can be created using the Intusoft nonlinear magnetics model and passive elements. The new model, as shown in Figures 4 & 5, accurately simulates the impedance vs. frequency response and the change in impedance vs. temperature. The change in impedance with DC bias is also modeled due to the addition of the core model instead of a simple inductor. The bead model gives a very low DC impedance while providing a large impedance at higher frequencies. The model used here is for a ferrite bead made of a medium permeability nickel zinc material from Fair-Rite, P#2743009112. Other types of beads including beads on leads, wound beads and surface mount beads can be found in the RF library.
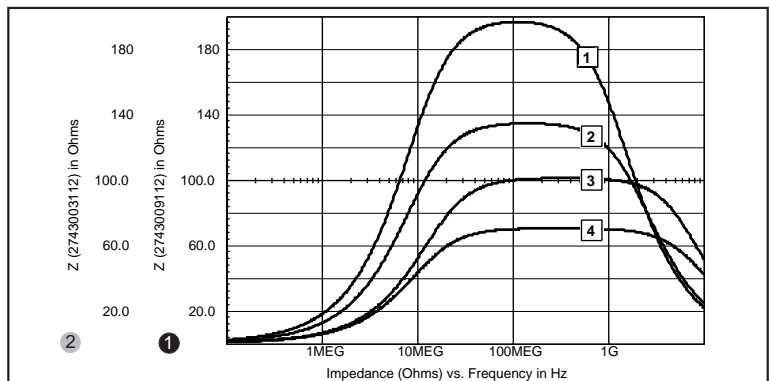
Starting An Oscillator

Simulation of oscillators present a variety of challenges, not the least of which is getting the oscillator to oscillate. When IsSpice

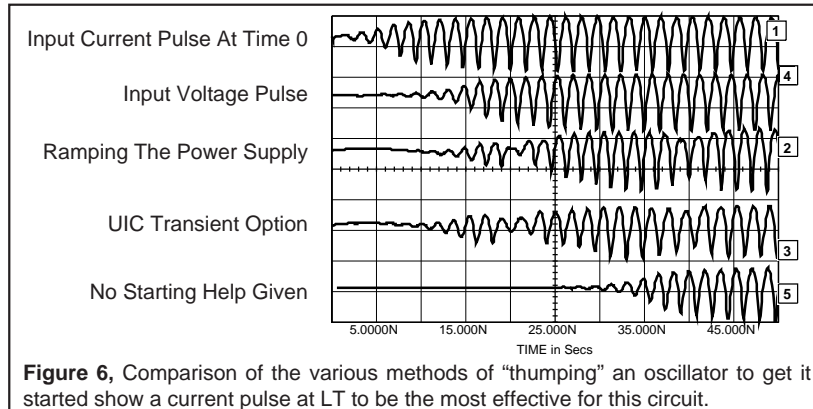**Figure 4,** A 500 MHz oscillator and RF Bead (shaded area) test circuit.

performs an AC or Transient analysis it first performs a DC analysis in order to establish the starting initial operating point for the circuit. If a stable operating point is found, which is the goal of the DC analysis, the oscillator may not oscillate during the transient (time domain) analysis unless some random disturbance is encountered.

There are a number of ways to start an oscillator. The method chosen here was to introduce a voltage pulse into the circuit, specifically, at the emitter of the transistor (`VIN 1 0 PWL...`). Another possible method is to insert a current pulse somewhere in the resonant portion of the circuit, for example at LT (`I1 18 17 PULSE .01 0`). If the DC analysis does not converge, a sign that the circuit is unstable and may want to oscillate without any help, the UIC keyword can be issued in the .TRAN statement. For example, `.TRAN .1NS 50NS UIC`. This will cause the simulation

Figure 5, The Intusoft RF Bead model simulates the proper impedance vs. frequency characteristics. The graph displays the response for several devices.

Starting An Oscillator *(cont'd)*

Input Current Pulse At Time 0
Input Voltage Pulse
Ramping The Power Supply
UIC Transient Option
No Starting Help Given

5.0000N 15.000N 25.000N 35.000N 45.000N
TIME in Secs

Figure 6, Comparison of the various methods of "thumping" an oscillator to get it started show a current pulse at LT to be the most effective for this circuit.

to proceed directly to the Transient analysis bypassing the DC analysis. The .IC and "IC=" parameters can then be used to set initial transient conditions and unbalance the oscillator. One problem with using UIC is that no DC operating point will be produced inhibiting study of the circuit bias. The last method is similar to the first and involves ramping of the power supply (`VCC 8 0 PULSE 0 8V 0 5NS`). This method may not work well, however, due to the bypass capacitors. In general, when ramping a source, make sure to give the ramp a realistic slope in order to avoid "timestep too small" errors. The first two methods are the most often recommended as the other methods may not work or may introduce transient start-up residues (Figure 6).

Results In Part II, Next Newsletter

In part II of this article we will give the results of the simulation of the two versions of the oscillator. We will also explore the different characteristics resulting from use of the transistor model with parasitics and the RF bead. The RF oscillator circuit, rat race circuit (shown opposite page), and several transistor/bead models are available on floppy disk for $20.
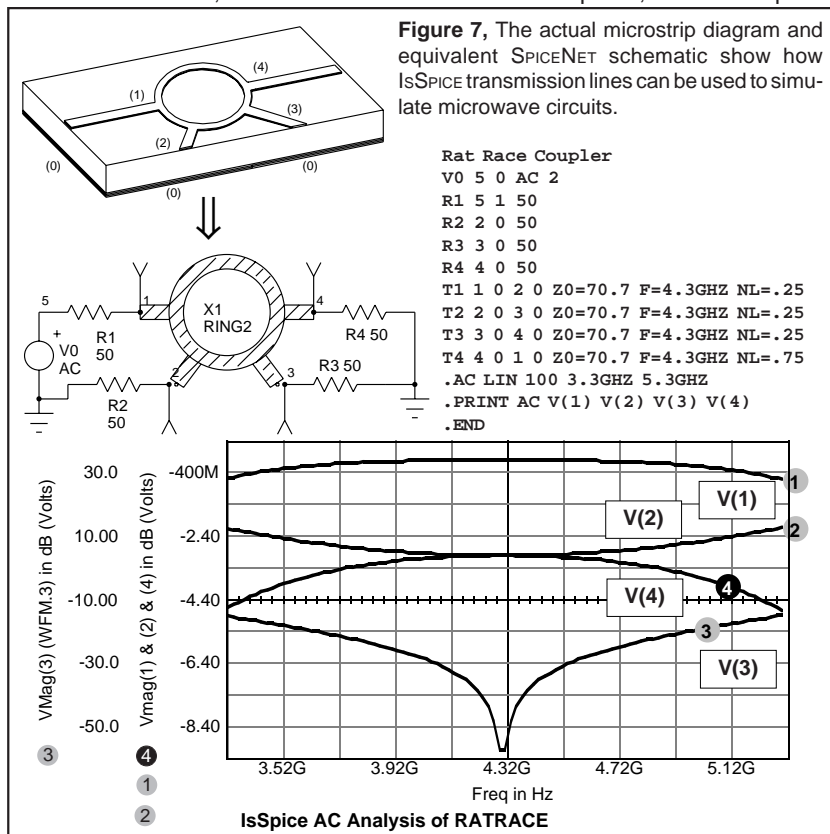
Table 1, IsSpice Subcircuit for an RF Transistor & Bead

```
.SUBCKT QN5109 1 2 3
LC 1 4 0.875E-9
LB 2 6 1.590E-9
LE 5 3 2.650E-9
CC 4 3 1.410E-12
CB 4 6 0.561E-12
Q1 4 6 5 QR33
.MODEL QR33 NPN ( BF=44 VAF=160 VAR=16.0 RC=0.69
+RB=1.57 RE=2.75 IKF=0.28E+00 ISE=0.36E-13 TF=0.111E-09
+TR=0.80E-08 ITF=0.82E-01 VTF=0.66E+01 CJC=2.758E-12
+CJE=1.822E-12 XTI=3.0 NE=1.5 ISC=0.12E-13 EG=1.11
+XTB=1.5 BR=1.14 VJC=0.75 VJE=0.75 IS=0.40E-14
+MJC=0.33 MJE=0.33 XTF=4.0 IKR=0.28E+00 KF=0.1E-14
+NC=1.7 FC=0.50 RBM=1.1 IRB=0.40E-01 XCJC=0.5 )
.ENDS
```

```
.SUBCKT  BEAD 1  2
R4 1 2 220 TC=-.00333
C2 1 2 .9PF
RX 3 2 1E12
CB 3 2 7.432N
F1 1 2 VM1 1
G2 2 3 1 2 1
E1 4 2 3 2 1
VM1 4 5
RB 5 2 341.5
RS 5 6 3.7904
VP 7 2 300
D1 6 7 DCLAMP
VN 2 8 300
D2 8 6 DCLAMP
```

.MODEL DCLAMP D
+CJO=8.2286P VJ=25
.ENDS

A Microwave Frequency Microstrip Circuit

In recent years, microwave circuits have been built using microstrip because it is less expensive than using waveguides. Simulation of these circuits is possible using the built-in IsSpice transmission line. The rat race coupler simulation, shown below, is one of a number of coupler simulations described in detail in the new Intusoft publication "A Spice Cookbook" by Karl Müller. Here, the transmission lines are arranged circularly. This contrasts a hybrid coupler which is arranged in a rectangular structure. This is an advantage as the electric field pattern remains undistorted. Reference lines, which determine the electric length of the transmission lines, take their course right in the center of the conductors and, therefore, allow an almost exact prediction of the circuit's behavior. With rectangular bends, more or less distinct discontinuities appear depending on the slimness and bending angle of the conductors. A signal entering at port 1 will split between ports 1 and 4, with port 4's signal being 180 degrees out of phase with port 2's signal. This property is often exploited in balanced mixers and image rejection mixer designs. As shown below, almost total cancellation occurs at port 3, the isolated port.

Figure 7, The actual microstrip diagram and equivalent SpiceNet schematic show how IsSpice transmission lines can be used to simulate microwave circuits.

```
Rat Race Coupler
V0 5 0 AC 2
R1 5 1 50
R2 2 0 50
R3 3 0 50
R4 4 0 50
T1 1 0 2 0 Z0=70.7 F=4.3GHZ NL=.25
T2 2 0 3 0 Z0=70.7 F=4.3GHZ NL=.25
T3 3 0 4 0 Z0=70.7 F=4.3GHZ NL=.25
T4 4 0 1 0 Z0=70.7 F=4.3GHZ NL=.75
.AC LIN 100 3.3GHZ 5.3GHZ
.PRINT AC V(1) V(2) V(3) V(4)
.END
```

IsSpice AC Analysis of RATRACE

RF Device Library Update (continued from page 23-1)

A number of major improvements have been made to the bipolar transistor models in the new RF library including:

- **more accurate HFE rolloff at all collector currents**
- **more accurate junction leakage**
- **better tracking of HFE and the s-parameters with temperature**
- **better tracking of the s-parameters with bias**

A custom subcircuit approach is utilized that accounts for all package parasitics and matches the published s - parameter magnitude and phase data. These models will allow any SPICE program to simulate linear and nonlinear RF circuits using AC, DC, and Transient analyses.

The specialized library includes IsSPICE netlists, measured and simulated device s-parameters, and schematic symbols for the SPICENET schematic entry program. Test circuits that will allow a transistor's s - parameters to be plotted from an IsSPICE simulations are also included. The models and circuits are available on floppy disk in PC (5-1/4" and 3-1/2", ASCII) and Macintosh (TEXT) formats. A current list of the models is available by contacting Intusoft.

Newsletter Subscription (continued from page 23-1)

Annual subscriptions will be $40 (domestic) and $55 (Foreign). Multiple year rates are also available.

The new subscription offer will allow current and interested readers to be assured of quick delivery, as well as of being on the mailing list. Normally, a copy of the newsletter along with its accompanying floppy disk is available for $20 per issue. The subscription program represents a substantial cost savings and a tremendous convenience for all those who value its quality technical information. So if you do not get your own copy or would like to make sure that you don't miss an issue, here is your chance.

Some of the expected topics to be covered in the newsletter include: transformer modeling, IGBT/SCR modeling, mixed mode simulation, pin diode modeling, filter design and more. Newsletter subscribers will also be able to purchase back issues for only $10 each as long as supplies last. You do not have to own any of Intusoft's CAE tools to purchase the newsletter subscription.