



(310) 833-0710

Personal Computer
Circuit Design
Tools

NEWSLETTER

Copyright © *intusoft*, October 1988

Intusoft Updates Its Integrated CAE Package

Rounding out this summer's IS_SPICE version 1.41 update, are the updates for our other intusoft CAE products SPICE_NET, Intu_Scope, and PRE_SPICE. In addition, our new book, *SIMULATING WITH SPICE* begins shipping on October 21. Everyone who purchased IS_SPICE version 1.41 or SPICE/386 or an update received a green card entitling them to a free copy. If you still have the card, be sure to mail it in to get your copy of *SIMULATING WITH SPICE*. Additional copies of the book are available.

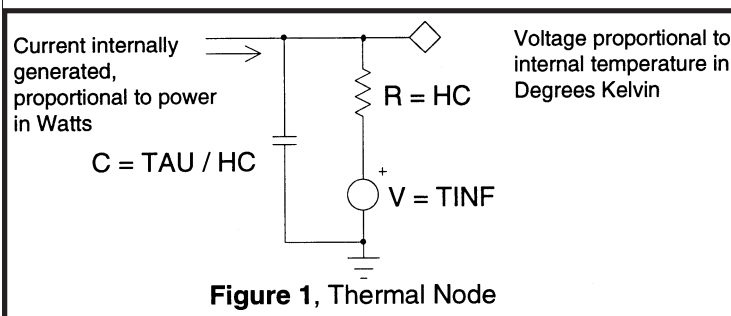
In This Issue

- 2 Thermal Models
- 5 A Light Bulb
- 8 SPICE_NET Update
- 10 PRE_SPICE Update
- 12 Intu_Scope Update

Our Fall updates include PRE_SPICE version 2.1 and Intu_Scope version 2.1 which will begin shipping November 7, 1988. These updates are free if you purchased the respective software packages after August 1, 1988. If you purchased the software before that date, the updates must be purchased. Updates from PRE_SPICE version 1.0 and Intu_Scope version 1.0 to the current version are available. All update orders require the serial number of the original software. The updates are carefully integrated with the SPICE_NET version 2.0 update which will ship on December, 28. The SPICE_NET 2.0 update is free to all who purchased version 1.0 and returned their blue card. Many of you purchased SPICE_NET through your company or from dealers and we have no way of delivering this valuable update without the cards we provide with the software. You must send in the blue update card in order to receive the update. The bingo card on the back of the newsletter can be used to order the PRE_SPICE and Intu_Scope updates.

Thermal Modeling of Semiconductors

In our July 1988 newsletter several models were developed that use temperature as a simulation variable. We concluded the application showing a model for an NPN bipolar junction transistor that included a thermal node. The thermal node introduced in July is shown again in figure 6. Thermal nodes, associated with each component can be interconnected in a two or three dimensional network that represents the heat flow parameters from the device junction through the various interfaces to a heat sink and on toward some definable boundary. Thermal nodes can also stand alone within models such as thermistors or light bulbs where the feedback path is completed within the device model.



It is fairly straight forward to convert the NPN transistor thermal model to a PNP by reversing appropriate polarities and switching from NPN to PNP transistors. Likewise, a diode can be modeled by shorting the NPN collector to base. Unneeded parts can be removed and a diode model substituted for the diode connected transistor. The final subcircuit listings for the PNP transistor and the Diode are shown in table 1.

These models can then be interconnected as shown in Figure 7 to simulate a class AB complementary amplifier mounted to a heat sink. The thermal nodes are then connected to a heat sink model. The heat sink consists of a three dimensional finite element representation using resistors to represent conduction loss and capacitors for modeling specific heat. Additional detail can be added by considering the thermal interface at the die attach and case to heat sink mounting. For this example, these were all lumped with the heat sink and a single junction to heat sink thermal resistance was used. The time constant used in the transistor model was that of the semiconductor die. The TINF and HC terms in the diode and transistor models are

Table 1, Thermal models for a PNP transistor and Diode

```

.SUBCKT TPBJT 7 16 10 4
V1 7 1
V2 6 2
H1 3 0 V1 1
RH1 3 0 1MEG
H2 5 0 V2 1
RH2 5 0 1MEG
EX2 13 0 POLY(2) 7 10 3 0 0 0 0 1
RX2 13 0 1MEG
EX3 8 0 POLY(2) 5 0 16 10 0 0 0 1
RX3 8 0 1MEG
EX4 9 0 POLY(2) 13 0 8 0 0 1 1
RX4 9 0 1MEG
G1 0 4 9 0 1
R1 4 11 {HC}
C1 4 0 {TAU/HC}
V3 11 0 {TINF}
*ICBO SET TO DOUBLE EVERY 10 Deg. C
*Vbe decreases 2mv per Deg. C
EEXP 40 0 4 0 {6.93E-2*.0258642}
RX5 40 0 1MEG
X5 40 12 EXPP
G2 16 7 12 0 {ICBO*9.35}
E1 6 16 0 4.62M
Q1 1 2 10 PBJT
.MODEL PBJT PNP(CJC={2.2*COB} TF={.16/FT} TR={1.7*TS}
+ CJE={7*COB} RC={.5/IMAX} VAF=100 IKF={.7*IMAX} IS={2E-15*IMAX})
.ENDS
*****

.SUBCKT TDIODE 16 10 4
V2 6 1
H2 5 0 V2 1
RH2 5 0 1MEG
EX3 9 0 POLY(2) 5 0 16 10 0 0 0 1
RX3 9 0 1MEG
G1 0 4 9 0 1
R1 4 11 {HC}
C1 4 0 {TAU/HC}
V3 11 0 {TINF}
*ICO SET TO DOUBLE EVERY 10 Deg. C
*Vbe decreases 2mv per Deg. C
EEXP 40 0 4 0 {6.93E-2*.0258642}
RX5 40 0 1MEG
X5 40 12 EXPP
G2 10 16 12 0 {ICO*9.35}
E1 16 6 0 4.62M
D1 1 10 DIODEX
.MODEL DIODEX D(CJO={30P*IMAX} IS={5N*IMAX} TT={1.44*TRR}
+ BV={1.5*VMAX} IBV=1UA N=2 RS={.075/IMAX})
.ENDS
*****

.SUBCKT EXPP 3 2
R 2 0 1MEG
V 3 4
RD 4 0 1MEG
D 4 0 DIODE
.MODEL DIODE D(IS=1E-10)
I 0 4 -1E-10
H 2 0 V 1
.ENDS
*****

```

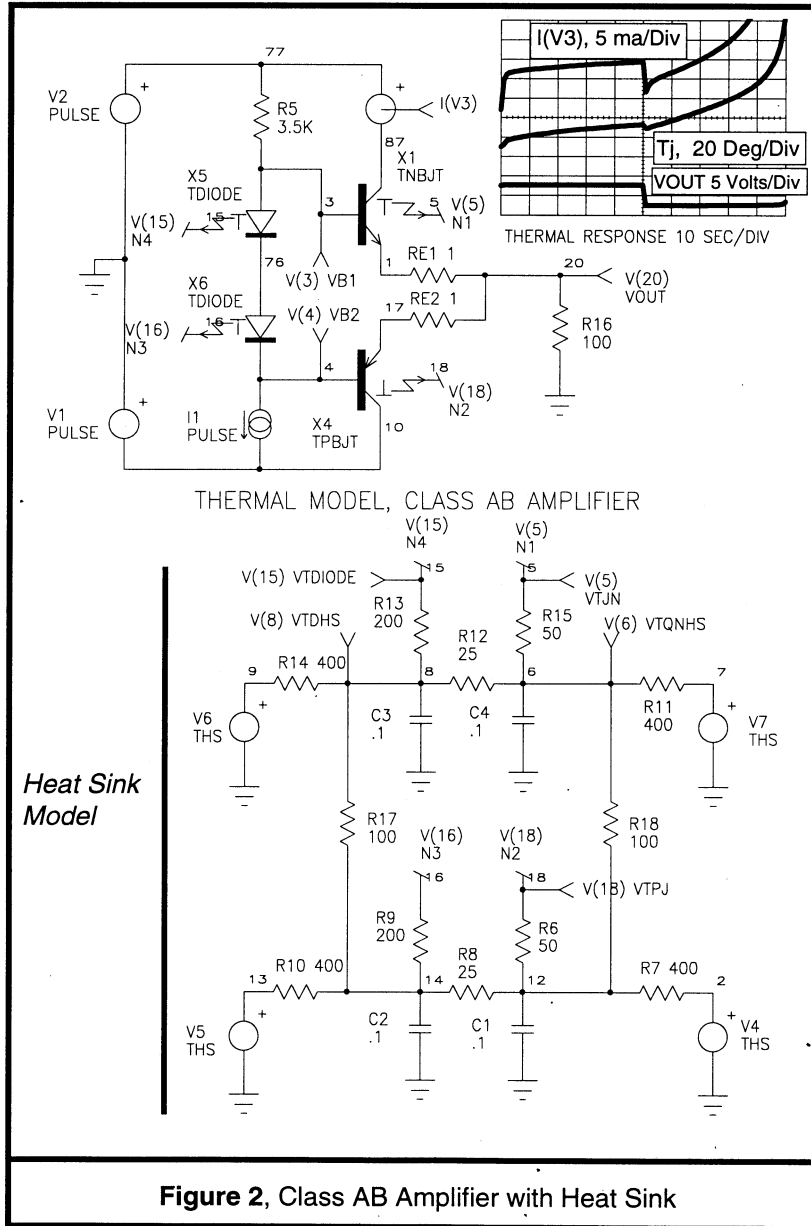
TPBJT Node Connections
Collector, Base, Emitter, Thermal

TDIODE Node Connections
Anode, Cathode, Thermal

EXPP Node Connections
In, Out

Thermal Modeling of Semiconductors

negligible because they represent the radiation and convection paths from the die to the outside world. The ratio of τ_{th}/hC represents specific heat and therefore its computation must be reasonable. Transistor and heat sink parameters were set up to represent TO5 geometry transistors, with a PC card used for the heat sink.



The diodes in this circuit compensate for V_{be} changes with temperature in order to prevent thermal runaway. The thermal stability of this circuit depends on the heat sink characteristics as well as the values of emitter resistors (RE1 and RE2); hence the need for a dynamic thermal model. The results shown in Figure 2 represent some arbitrary design parameters, typical of a first cut design. The power supplies were turned on at the start of the simulation. After 50 seconds, a pulse caused the output to go 5 volts negative. After the first 50 seconds, the temperature and bias current were approaching steady state. Shortly after the event at 50 seconds, the overall heat sink temperature rose past a critical point and thermal runaway began. Temperature was rising rapidly at 100 seconds; indicating eminent circuit failure. Stability of this circuit is a trade off between circuit performance (low emitter resistor value), heat sink size, and good thermal coupling to the temperature compensation diodes. Adding the thermal models to SPICE allows the circuit behavior to be explored using simulation. The level of detail in a simulation exceeds that for hand analysis and of course saves the burned fingers encountered when testing for thermal runaway in the laboratory.

■ A Tungsten Lamp Model

Tungsten filaments operate at temperatures in excess of 3000 Deg. Kelvin. The positive temperature coefficient results in an inrush of current an order of magnitude above steady state. Solid state controllers must be designed to withstand this severe change in the electrical environment; hence a need for an electrical simulation. The modeling of a Tungsten Lamp for SPICE requires a model for the temperature variation of the resistance of Tungsten. Incorporating thermal feedback will then differentiate transient from steady state operation and allow accurate prediction of control circuit performance. We used handbook data to get the resistance versus temperature data tabulated below.

Resistance (Normalized)	Temperature (°K)	Resistance (Normalized)	Temperature (°K)
.05	300	.92	2102
.25	403	.96	2493
.42	547	.96	2493
.55	725	1.00	2900
.64	950	1.00	2900
.72	1201	1.00	2900
.80	1470	1.04	3324
.87	1764	1.04	3324
.92	2102	1.08	3765
		1.08	3765

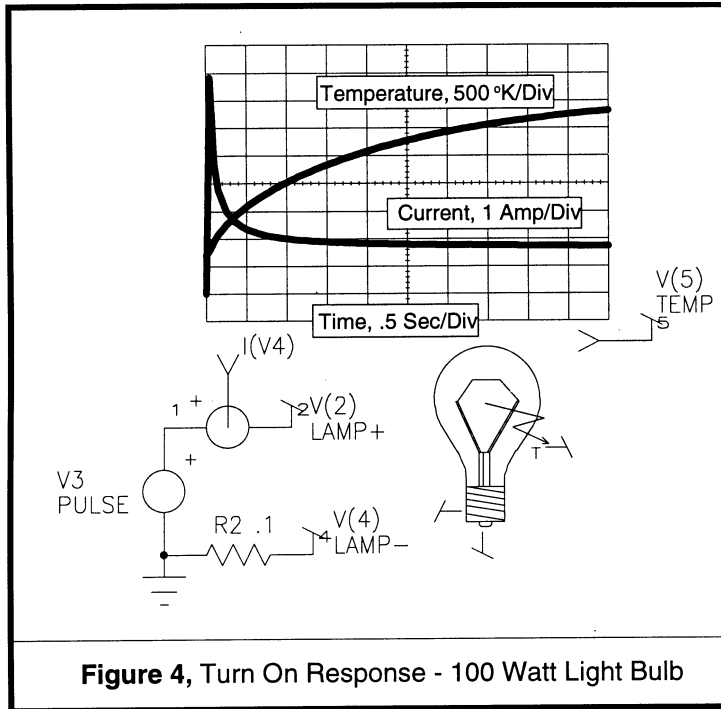


Figure 4, Turn On Response - 100 Watt Light Bulb

Table 3, LAMP Subcircuit

```

.SUBCKT LAMP 2 3 8
*          HI  LOW  THERMAL
V4 2 5
RV4 3 5 1MEG
E9 5 3 POLY(2) 4 0 10 0 0 0 0 0 {V*V/P}
RE5 5 3 1MEG
G6 0 8 POLY(2) 4 0 5 3 0 0 0 0 1
R2 8 7 {2600/P}
V3 7 0 300
C1 8 0 {1.57M*P}
EPOLY 10 0 8 0 -4.000190E-1, 1.789738E-3, -9.088125E-7,
+ 2.048935E-10, -1.642730E-14,
REPOLY 10 0 1MEG
H1 4 0 V4 1
RH1 4 0 1MEG
.ENDS
*****

```

LAMP Node Connections
Hi, Low, Thermal

Properties of Tungsten:

Density: 19.3 gr/cm³
Thermal Conductivity: 1.99 Watts/cm °C
Specific Heat: .034 Cal/gr °C

SPICE_NET Version 2.0 Update Review

Our most exciting updates are to the SPICE_NET program. Not only is SPICE_NET fun to use, its fun to update. It's really difficult to stop enhancing the program and let the users benefit from all the new goodies. We added a tremendous amount of power and features to SPICE_NET, with most of the ideas coming from our users. We want to thank you for all your excellent suggestions. We hope you find most of them in the new update. In any case, please keep the suggestions coming. We will be sending the update out beginning November 28. Make sure you've sent us the blue update card. This is a free update!

Multiple Pages, Plotting and Printing

SPICE_NET now supports multiple pages. The continuation symbol allows nodes to connect across pages or on the same page by using names for nodes. Wires are no longer necessary to connect distant nodes. Sections of circuitry can be cut and pasted between different pages or schematics. There are now 5 page sizes, from A through E size and two orientations, landscape and portrait, for each page. The outline of the paper limits are shown on the screen, but margins will be up to you and your output device. We have completed the Houston Instruments plotter driver and plan to include HP Laser Series II and HP DeskJet drivers. Node numbers will be printable on all output formats when the proper options switch is turned on.

Symbols, Subcircuits, and Libraries

Dozens of new symbols have been added corresponding to the new SPICE models added to PRE_SPICE. Symbols now have the ability to have hidden pins and interconnections. The associated subcircuit calls and netlist code generated is still compatible with SPICE 2G6. This will allow subcircuit nodes, such as commonly used power connections VCC or VEE to be hidden.

Subcircuits, ready for insertion into a model library, can now be created automatically. SPICE_NET will convert everything on a page into a subcircuit. Continuation symbols are used to designate the node list. The netlist is then saved in a separate file. SPICE_NET remembers the correct sequence for the node list and will tell you when and where to place the proper pins when the corresponding symbol is created. The thermal model for the tungsten lamp found earlier in this newsletter is an example of how SPICE_NET creates symbols and subcircuits quickly and easily.

Our symbol library was getting too large for a single file, so we now use the DOS tree structure for a file hierarchy. Subdirectories are listed in parentheses at the top of the subcircuit selection menu,

followed by the symbols. When a new directory is selected, its contents are shown. The directories are made by you and can contain any symbols in any combination, though we recommend making directories with names corresponding to our library names.

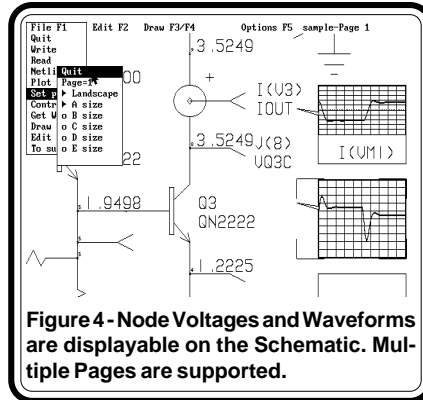
SPICE_NET supports the environment variable "ISLIB" found in PRE_SPICE, which allows model libraries to be placed anywhere on the hard disk. Each symbol now has a library name attached which will now cause a *INCLUDE command to be automatically generated in the control menu. You can edit the include item, for example change from NONLIN.LIB to LIN.LIB, if you want to use a different library than originally designated by the component.

Viewing DC Operating Point Information

Node voltages from the SPICE simulation output file can now be viewed. This capability provides a tremendous help in debugging your circuit. Voltages are automatically placed next to each node in a window. Look at Figure 4 below to see the results.

User Interface, VGA Graphics and More Colors

SPICE_NET now supports VGA, and both EGA and VGA will use up to 16 colors selected from a 64 color palette. Components, artwork, wires, nodes, menus, frames, waveforms, grids and the cursor can each have different colors. Double clicking on a symbol will now bring up its part label menu. The right mouse button is used to activate the fit screen view or to zoom in. Several function key macros have been



added. The function keys F6 through F10 are used to allow zooming in or out or the changing of pages. The Del key will delete selected objects. The Ins key allows selected objects to be copied. Test points will automatically be given names corresponding to their SPICE identification, I(Vxxx) or V(xxx). Other names may also be assigned to test points (alias feature). We hope that between these two features, the printing of node numbers will become unnecessary. All built-in symbols can have multiple artwork definitions or case frames using a prefix number. Various *intusoft* and SPICE related syntax help has also been added to the control menu, label menu and the message window.

intusoft Program Updates

Bugs Corrections

Many of the changes discussed corrected deficiencies in the programs. A number of bugs were also fixed such as the copy command carrying the prior symbol's attributes over to copied parts. The cursor will now be left in its old position after keystroke macros are performed. The slow ruler redraw has been sped up and the re-netting function was changed slightly and is now more robust.

PRE_SPICE Version 2.1 Update Review

The ICAPS interactive program selector now supports mouse control and SPICE circuit netlists, intermediate netlists and output files can now be printed from the menu.

New Model Libraries

The following SPICE libraries and models have been added to PRE_SPICE. This is the first in a series of major updates to our SPICE model libraries. Additional updates to our SPICE model libraries will include greater numbers of discrete parts (OpAmps, Diodes, Transistors, and Power Mosfets), as well as more complex macro models (Relays, MOV's, TTL gates, Analog Switches, Pulse Width Modulators, Comparators, Voltage Regulators, and Vacuum Tubes).

New SPICE Models Available with PRE_SPICE

THERMAL.LIB (Thermal Models for Temperature Simulation)

TNBJT	NPN transistor with temp. variable
TPBJT	PNP transistor with temp variable
TDIODE	Diode with temp. variable
THRMSTR	Thermistor with temp. variable
LAMP	Tungsten Lamp with temp. variable
TCELL	General purpose thermal cell

DEVICE2.LIB (Generic and Discrete Component Models)

RPOT	Voltage Controlled Resistor
EXPP	e ^x Function
DIVIDE	Divider
HTAN	Hyperbolic Tangent Function
XTAL	Crystal, for oscillators and filters
Crystals	10Mhz, 3.579545Mhz
Diodes	1N4002, 1N5400, 1N5718, 1N914
HL7801E	Laser Diode with threshold thermal + monitor diode
Zeners	1N746-1N759, 1N4370-4372, 1N957-1N992

SIGNAL.LIB	(Generic Waveform Generators)
SIN	Sin Wave Generator
PULSE	Pulse Wave Generator
SAW	Sawtooth Wave Generator
VCO	Voltage Controlled Oscillator
AM	AM Source
FM	FM Source
RANDOM.LIB	(Random Noise Signal Sources)
RAN1	Random Voltage Generator
RAN2	Random Voltage Generator
RAN3	Random Voltage Generator
RAN4	Random Voltage Generator
RAN5	Random Voltage Generator
POWER.LIB	(Power Electronics Related Models)
GEN3	Three Phase Power Source

Total library entries now include 72 high level generic models capable of simulating thousands of components. These generic components provide immediate and comprehensive conversions from data sheet parameters to usable SPICE parameters and models. The user is free to view, alter and copy any model in the intusoft library. Documentation and examples of each generic template model are provided. Other discrete SPICE models of commonly used components are installed in libraries and ready for immediate use.

New Features

As discussed before, a DOS environment variable path name, ISLIB = path, was added, allowing model libraries to be stored and searched anywhere on the hard disk.

A *DEFINE statement was added allowing long expressions to be defined and substituted throughout the file. The command will perform an A=B substitution. Each time expression "A" is encountered in the file, expression "B" will be substituted. It simplifies the typing of parameter lists for subcircuits or MOS transistors. For example, if the statement,

```
*DEFINE OP10=OP10 {VOS=10U IOS=5UA FT=10MEG}
```

were placed in the netlist, each occurrence of "OP10" would be replaced with "OP10 {VOS= }". This not only simplifies the data entry, but gives a more meaningful purpose to the SPICE_NET schematic name/value field. The *DEFINE is used in the upcoming SPICE_NET update to assign node numbers to symbolic names. This technique allows selected schematic interconnections, such as power lines, to be hidden.

The functions which may be evaluated in subcircuits and have parameters passed to them can now contain the following functions:

intusoft Program Updates

log, exp(e^x), sin, cos, tan, asin, acos, and atan. Subcircuits that use parameter passing may also have default parameters. This eliminates the need to pass any or all parameters. The default parameters are placed at the end of the subcircuit definition line.

The size limitation on Monte Carlo output files has been doubled and the maximum number of measurement functions has been increased to 64.

Bug Corrections

The IS_ED editor will no longer choke when long file sections are copied or deleted. The Monte.exe program no longer truncates small numbers to zero. Param now accepts subcircuit names with 7 characters and will provide improved extended syntax error messages. The include program will now find transistors which use the 4th substrate node. Several model syntax errors have been fixed.

Intu_Scope Version 2.1 Update Review

New Intu_Scope features include support for Houston Instrument Plotters, 4 through 9 cycle log grids, and alias name substitution.

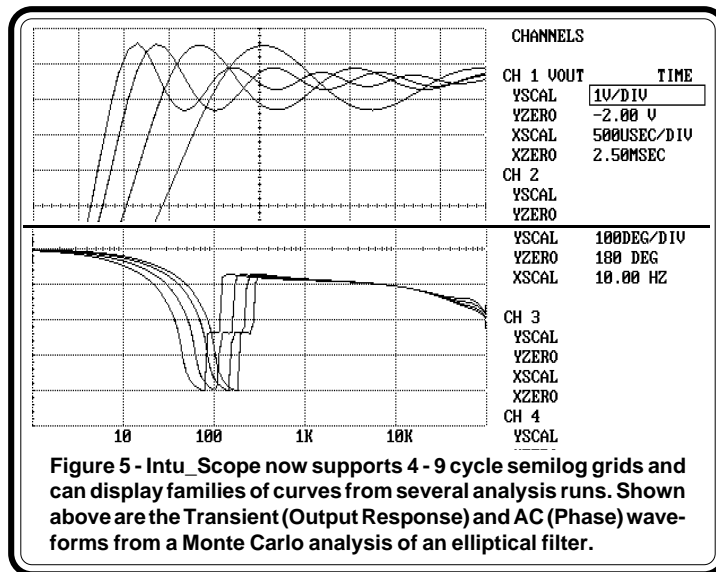
The alias feature allows Intu_Scope to use and display symbolic node and current names in the place of node numbers. Very long node fields can now be redefined with a name allowing Intu_Scope to display any output designation. For example, the voltage difference designation V(12000,39000) could be called VOUT using the *ALIAS option in the SPICE_NET label menu or in your SPICE netlist. Vout would then appear in the node selection field in Intu_Scope.

"N" cycle log grids have been added to the standard 1 and 3 cycle grids. The new grids vary from 4 through 9 cycles with the default for AC analysis now set to 5 cycles. Figure 5 shows a 5 cycle format.

Along with the Houston Instruments plotter support is the capability to rotate the plot by 90 Degrees in order to plot both portrait and landscape formats.

The ICAPS program now passes the current filename into the Intu_Scope program except when parameter sweeping, circuit optimization and Monte Carlo analysis are being performed.

Data analysis for the Monte Carlo, parameter sweeping, and circuit optimization analyses has been augmented with several new capabilities. Intu_Scope can now save whole waveforms during each Monte Carlo, parameter sweep, or circuit optimization run. This gives the user the ability to save a family of curves which depict the behavior of the circuit over the whole analysis range. Another feature of this wave saving capability allows the user to create a ceiling or floor



(envelope) representation of the family of curves. With our existing histogram and cumulative probability grids, this is the most complete Monte Carlo statistical yield and evaluation package for PC's.

Several printout bugs were fixed and the dot matrix character output is now generated using internal vectors to make graphic characters. This eliminates character pitch dependencies for various plotter manufacturers.

A file editing capability was added to the PRE_SPICE package allowing keystroke (.PGM files) to be edited. This allows complex macros to easily be written and saved for later use.

Colors selections (EGA and VGA) for waveforms were increased from 3 to 7 colors and now cycle automatically as different waveforms are selected. VGA resolution is now available.

Bug Corrections

When the max or min functions are selected, the function key controlled cursor movements will now go to those selections, fixing the pass-by problem. Several underflow problems were detected and corrected. Horizontal and vertical overscan were reduced and vertical limiting introduced in order to fix overflows that were observed on certain expanded waveforms. Data points are now plotted as a larger five point "dot" if the data is sparse. The rejection of 256 to 512 point inverse FFT was corrected. Finally, the number of node references per analysis type was increased to 64.