The Virtual Circuit Design Lab

intusoft

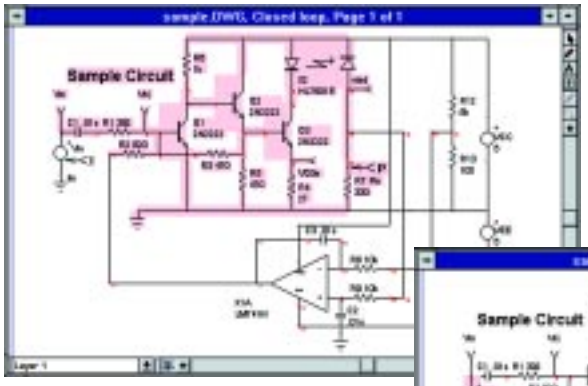
ICAP/4

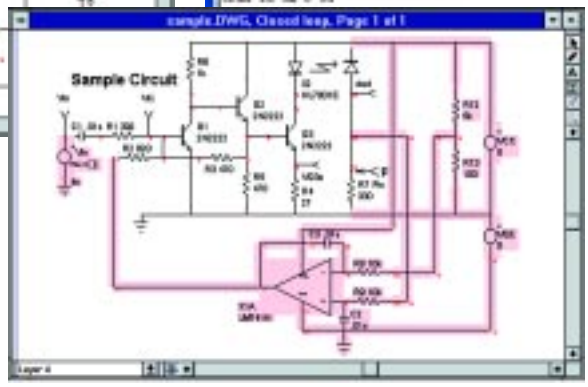
Schematic Capture...Mixed-Signal Simulation.....Waveform Analysis...Model Libraries

ICAP/4 is a powerful analog and mixed-signal circuit simulation software package. Ease-of-use combined with interactive design features allow you to experience simulation just the way you do when working with real hardware in the laboratory. Instead of building a breadboard, you can draw a schematic and ICAP/4 will create a simulation. Using special features in the schematic, you can interconnect the circuit to make different configurations, then for each configuration, you can run different simulations. For example, in one case you can perform a worst case transient analysis and in another you can perform an AC control system stability evaluation. All of this is tailored to making it easy for you, the designer, to explore every aspect of your design and to expose your design to simulated environments before committing the design to real hardware fabrication.

SpiceNet

The ICAP/4 package uses the schematic, SpiceNet, to manage all of your simulation data. It tells the IsSpice simulator about the component interconnections (netlists), the kind of simulation to run (TRAN, Worst Case) and the data that the simulator must produce (Trise, Gain margin). The schematic saves the simulation directives, test configurations and measurements in its own database. The figure below illustrates the way different layers in the schematic are combined into a single circuit for which performance can be measured using the IsSpice simulator.





Multiple Schematic Layers
SpiceNet lets you put schematic objects (test circuitry, artwork, etc.) on different layers. You can configure the combination of layers you want to simulate, print, or send them to PCB Layout.

SpiceNet places most IsSpice parts with a single keystroke; it even advances the starting position for the next symbol so that many interconnections are made without using a "wiring" tool. A user-configurable tool bar gives you single-click access to any menu item, including launching a simulation and viewing operating point data on the schematic. You can place text or artwork anywhere on the schematic. Waveform thumbnails can be imported from IsSpice using the probe tool and single-clicking on a node or part. You can even embed or link another document or image anywhere on the schematic. You can create symbols with the built-in symbol editor which includes Bezier curve and bitmap imports.

IsSpice4 Technology

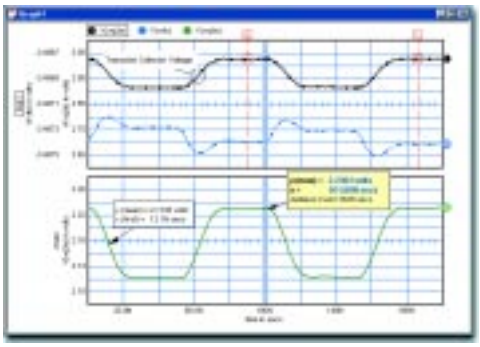
IsSpice4, the first commercial version of UC Berkeley SPICE 3F to use Georgia Tech's XSPICE, is still the only truly interactive Spice-based simulator. In the interactive mode, you can vary parameters and instantly view the results as a new simulation is run (see figure at left). Combining fast workstations with Intusoft's high performance models, you can make a design interactively and improve its quality by zeroing in on key performance parameters.

Over 100 Interactive Commands Language (ICL) commands and functions are available to help you run theses interactive simulations and extract important design information. ICL scripts can also be passed into the simulator and run automatically. Simulation templates were invented by Intusoft to extend the ICL in order to run a family of Worst Case analyses on top of any standard IsSpice simulation — making Intusoft's ICAP/4 package the only simulator to run Worst Case analysis for a transient simulation.

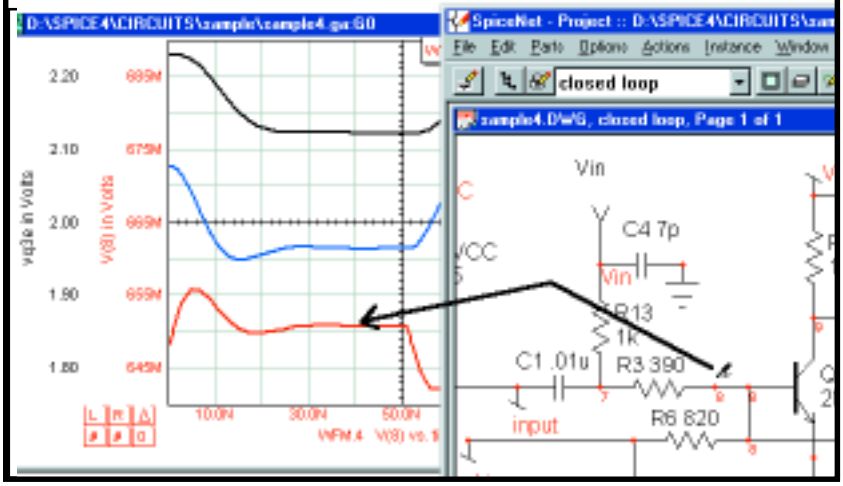
Intusoft P.O. Box 710 San Pedro, CA 90733-0710 Phone: (310) 833-0710 Fax: (310) 833-9658

IntuScope

IntuScope is an interactive graphical waveform processing program especially designed to display and analyze IsSpice output data. IntuScope can display waveforms from any Berkeley SPICE-compatible program, as well as user generated data files. It interfaces with IsSpice and SpiceNet to cross probe simulation data.



Cross Probing from SpiceNet: Waveforms are selected graphically, using the probe tool to view node voltage, component current or power.



IntuScope is more than just a SPICE post processor. It is a very powerful data processing system. It displays data as waveforms and contains a comprehensive set of waveform processing functions and operations. These functions include:

- Rise time
- Gain and phase margin
- Fourier transforms
- Group delay

And most recently IntuScope includes ICL compatibility, using the same script language as IsSpice.

Model Libraries

We have the best model library in the known world! Our total library size is now over **14,000** parts including vendor-supplied models that you may have tried to find on the web. With hundreds of part types, organized using our Part Browser, you can shop for the part that does the job for you.

And if you can't find the model you need in our library, we provide you with SpiceMod in our deluxe option package, a program that gives you the power to create an unlimited number of SPICE models for thousands of semiconductors. With SpiceMod, all you need is the manufacturer's data sheet and a few minutes of your time to make a model!

Why Buy ICAP/4?

Easy-To-Use

ICAP/4 provides on-line help with comprehensive SPICE reference materials and Intusoft's unique Multimedia Movies that guide you through the simulation process.

Affordable

With our wide range of products, ICAP/4 is affordable on any budget.

Unequaled SPICE Power

Schematic Layering: SpiceNet provides you with the unique ability to "layer" your design, taking the frustration out of your design work.

Simulation Control: ICAP/4 provides you with the ability to stop, start and pause your simulation at will.

Eliminates Product Hazards: Introduce faults and check for hazardous conditions using stress alarms.

ICAP/4 Family of Products

- **ICAP/4 Students** - Our educational version of ICAP/4. Versatility at a low, low, price!
- **ICAP/4Rx** - Reduced complexity. The power to handle tough designs at a reasonable price. Also available with RF Deluxe and Power Deluxe options.
- **ICAP/4 Windows** - The ultimate in power, price, and performance. RF and Power Deluxe options are also available.
- **ICAP/4 Professional** - The premiere system, IC, and board-level simulation and analysis toolset.
- **Power Supply Designer** - Integrates Intusoft's schematic technology with the proven IsSpice4 analog and mixed-signal simulator and the powerful Magnetics Designer transformer and inductor synthesis tool.
- **Test Designer** - Our high-end product for failure analysis, product acceptance Test Program Set development (TPS) and fault isolation test design.

1 year maintenance program included with all ICAP/4 products

See us on the web at www.intusoft.com