ICAP/4

Complete Family of Simulation Tools for Design Entry, Verification and Test

*World's Most Powerful Virtual Circuit Design Lab*

ICAP/4 is a powerful analog and mixed-signal SPICE design-automation product suite that's scalable in price and features. Its ease of use and unique interactive design features replace slow and cumbersome methods of traditional SPICE or building circuit prototypes. ICAP/4's design entry system (SpiceNet) is full featured, including patented circuit layering and configuration capability. The 4th-generation "IsSpice4" kernel is built from significantly enhanced SPICE 3F.5 and XSPICE, with outstanding convergence on tough design topologies. With SpiceNet and IsSpice4, any type of simulation can be performed on a variety of design, such as transient analysis on a top level SMPS topology, worst case analysis on a critical portion of a GPS receiver, and pole-zero or 2-port analysis to measure stability of a high-gain feedback loop. Add in a host of fast "what-if" debug features and you've got unprecedented exploration into every aspect of analog or mixed-mode designs. Finally, ICAP/4's suite of verification, fault and test applications satisfy any degree of design compliance before committing to a production board or IC.

SpiceNet - Design Entry Made Easy

All ICAP/4 tools use "SpiceNet" as the schematic entry and design management system. SpiceNet is seamless to all simulation and verification functions. It maps circuit interconnects and components to the IsSpice4 simulation kernel, including type of simulation run, waveform viewing and valuable simulation messaging.
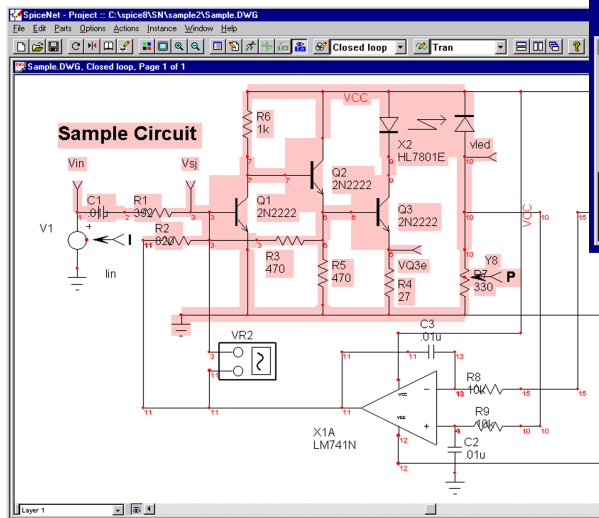
The figures below show different schematic layers combined into a single circuit using SpiceNet's fast inter-layer wiring. One-button layer highlighting provides fast visibility of any circuit section. Design configurations are also quickly brought to view and any changes are automatically dispersed across all configurations.
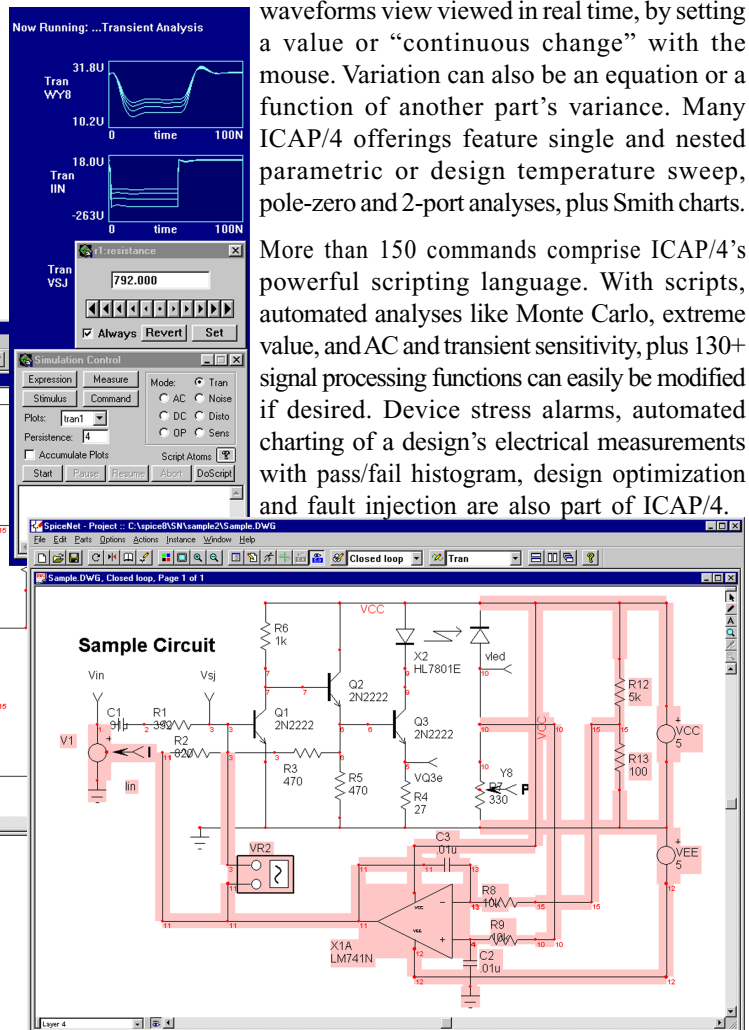
SpiceNet is easy; it's all menu, icon and dialog-box driven. Component placement is fast from an easy-to-use parts browser of more than 23,800 parts, including direct model editing from the dialog box. SpiceNet's configurable tool bar and menus provide fast access to a multitude of functions like schematic configurations, different simulation setups, window viewing preference, view of Operating Point data on schematic, group copy/paste of a design section with rubberbanding and much more. Easily place text, waveforms and custom artwork creations such as lines, arcs, circles, ellipses, and arrows directly on the schematic.
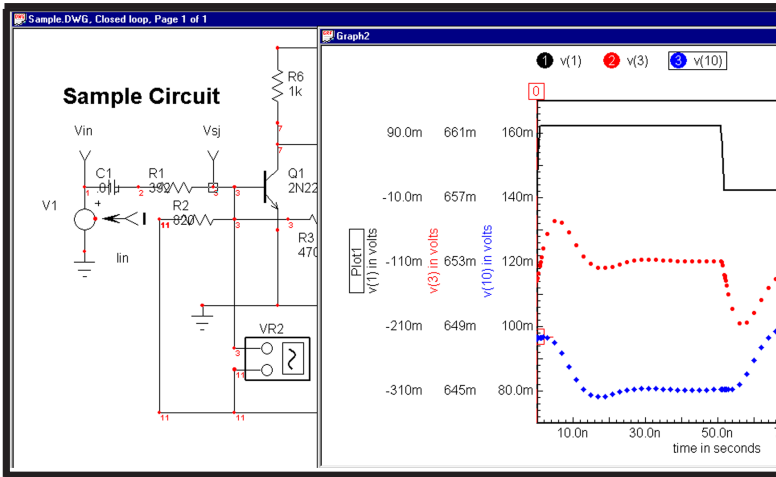
IsSpice4 - Proven Simulation Technology

IsSpice4 was the first commercial version of U.C. Berkeley SPICE 3F.5 to be integrated with XSPICE. Today, the powerful kernel features more speed and unique convergence algorithms than any other competitor. Also provided is a convergence wizard for helping automate convergence on difficult designs. Component parameters can be varied interactively without schematic intervention, with waveforms view viewed in real time, by setting a value or "continuous change" with the mouse. Variation can also be an equation or a function of another part's variance. Many ICAP/4 offerings feature single and nested parametric or design temperature sweep, pole-zero and 2-port analyses, plus Smith charts.

More than 150 commands comprise ICAP/4's powerful scripting language. With scripts, automated analyses like Monte Carlo, extreme value, and AC and transient sensitivity, plus 130+ signal processing functions can easily be modified if desired. Device stress alarms, automated charting of a design's electrical measurements with pass/fail histogram, design optimization and fault injection are also part of ICAP/4.

Multiple Schematic Layers

SpiceNet enables insertion of objects (e.g., high-reliability parts, artwork, different stimuli, wave probes) on desired schematic layers. You can also quickly wire any combination of layers together as a unique design configuration for test, documentation, mil standard, export to PCB, etc.

Sample Circuit

IntuScope - Industry's Most Powerful Waveform Viewer

"IntuScope" is an interactive waveform viewing tool. It interfaces directly to SpiceNet schematics for direct cross-probing of signal waveforms. This includes node voltages, plus component current and power. IntuScope provides more features than you would expect from an advanced waveform viewer. Updating of existing signal plots can be done three ways: update plots, append waveforms, or create new plot. Along with cross-probing, this minimizes an engineer's time during repetitive debug cycles. IntuScope's 130+ math and signal-processing calculator functions, provide things like waveform filters, signal conversions and waveform synthesis. Examples include: EMI spectrum (class A–D), group delay, FFT, filter functions, Hanning taper and eye diagrams. Multiple graphs, cursors and numerical data are fully configurable. IntuScope also furnishes multiple plot types and statistical data from Monte Carlo runs. Waveform views are provided for different simulation types at the click of the mouse, and in selectable units of measure. You can also plot one independent variable against another (i.e., Ic verses temp.) . All data and graphs can be saved and recalled for view or regression testing. What's more, the industry's only Digital Data Viewer numerically charts buss data (binary, octal, hexadecimal or mnemonic format) in synchronization with clock edges in IntuScope.

Part Browser

ICAP/4's hierarchical parts browser makes finding parts easy. Search by part number, model name, part type, subtype or description. Create a preferred part list for convenient part placement.

Model Libraries - Best in the SPICE World!

ICAP/4 provides a wealth of models for IC, digital, mechantronic, behavioral, discrete, RF, power, and mixed-mode design. An optional C-code modeling tool is also available. Special GFT models enable measurement of open loop properties without breaking a closed loop topology, particularly beneficial for SMPS and IC design. Library size, depending upon product, ranges from 15,500 to 23,800 parts.

With hundreds of part types well organized within ICAP/4's parts browser, finding a desired part is fast. Most ICAP/4 products supply "SpiceMod," a tool for easy creation and test of 14 model types from vendor datasheets. Library Manager assists SpiceMod in part verification, but also organizes and archives models, including comparison and search capability.

Why ICAP/4 Leads the Industry

Easy-To-Use:
ICAP/4 provides: In-application help, tutorial movies, New User's Tutorial, plus "Getting Started" and "IsSpice4 User's" guides.

Affordable:
ICAP/4's scalable product line is decisively affordable and designed to easily meet your budget. Upgrade to a higher end product at any time by only paying the price difference.

Unequaled Power of ICAP/4

Schematic Layering:
SpiceNet's unique layering and configuration system takes the frustration out of managing complex hierarchical designs.

Simulation Control:
Observe simulation progress with real-time waveforms. Modify component parameters interactively with ICAP/4's intuitive graphical interface. Modify existing scripts for customized simulation and waveform processing.

Eliminates Product Defects:
ICAP/4 tests designs using several analyses that ensure high reliability and manufacturability. Fault injection forces 3 types of part failures to analyze faulty circuit behavior. Test Designer combines test synthesis to ICAP/4Professional software. Most ICAP/4 offerings feature a powerful IBIS2SPICE converter.

Unequivocal Strength:
The unique and vastly enhanced algorithms of IsSpice4 ensure fast and accurate simulation of analog, mixed-signal, mixed-systems, RF, and power design for boards and ICs.

ICAP/4 Family of Products

- **ICAP/4Rx** - Entry level, but with full IsSpice4 power. Includes 15,500+ models. Features standard SPICE analyses, pole-zero analysis and single component sweep.
- **ICAP/4Windows** - Increases part count to 21,900+ models. Adds distortion, noise, AC & Transient sensitivity, transfer function, Monte Carlo, worst case, extreme value, optimization, nested parametric sweep and single fault injection. Includes graphical interface to interactively modify any circuit parameter. Features Library Manager.
- **ICAP/4Windows Power Deluxe** - Features ICAP/4 Windows but with additional power library of 1,100+ devices for power electronic designers. Also includes SpiceMod and Library Manager for parts management.
- **ICAP/4Windows RF Deluxe** - Features ICAP/4 Windows with special RF library of 600 parts. Also includes SpiceMod and Library Manager.
- **Magnetics Designer** - Industry's leading tool for the design and synthesis of virtually any inductive device. Includes wizards for easy setup. Outputs SPICE models and winding specification. Includes saturable core model.
- **Power Supply Designer** - Combines ICAP/4Windows Power Deluxe and Magnetics Designer.
- **ICAP/4 Professional** - Combines ICAP/4 Windows with Power and RF Deluxe. Total library count 23,300+. Also includes full automated fault analysis.
- **Test Designer** - ICAP/4's apex product suite for failure analysis, acceptance testing and fault isolation. Adds this powerful capability to everything from ICAP/4 Professional, including 23,800+ models.

First year of maintenance included with all ICAP/4 products