# ICAP™ for Viewlogic

## *The Wait for Integrated Simulation Power Is Over*
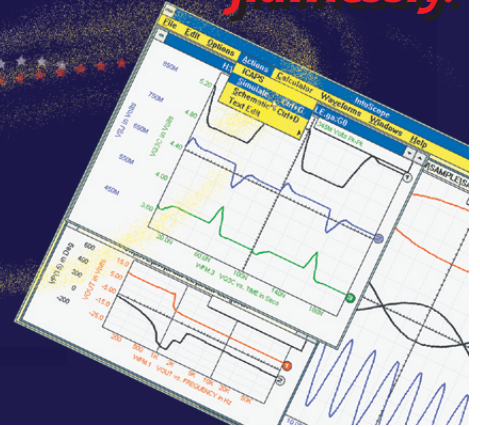
# Intusoft Links With Viewlogic

## OLE Technology Integrates Analog/Mixed-Signal Simulation with Schematic Entry

Presently, the most common schematic/SPICE interface is text based. This is clearly insufficient from the integration and interaction points of view. In addition, most schematic entry programs do not fully support all of the advanced SPICE features available today. Consequently, analog and mixed-signal simulation has been difficult to perform when using a third party schematic tool. Intusoft is proud to announce that the wait for *SEAMLESS INTEGRATION* and *EASE OF OPERATION* is over!

Intusoft has introduced a new product called **ICAP**™ for Viewlogic®. The new system uses Object Linking and Embedding (OLE) technology to transparently integrate analog and mixed-signal capabilities into Viewlogic's Workview Office™. Everything you need to perform circuit simulation is provided, including the powerful and proven IsSpice4 simulator, extensive SPICE model libraries, and a unique waveform data processor called IntuScope (See box below). IsSpice4 gives you full SPICE 3/XSPICE analysis power, along with a native mixed-mode (analog/digital) simulation capability. Simply put, **ICAP**™ provides the easiest and most powerful way to integrate analog and mixed-signal simulation with Viewlogic's schematic entry tools.

### How Intusoft Makes the Magic Happen

**ICAP**™ uses OLE techniques to communicate with ViewDraw and to integrate key simulation related features into Workview Office. The block diagram (above right) shows the communication paths. **ICAP**™ includes a new "magical" component, called the **Intusoft Integration Module**. The Integration Module attaches a host of features to ViewDraw in the form of Dynamic Link Libraries (DLLs), new menu items, and a floating toolbar. It is this new technology that allows you to **run SPICE simulations directly from your ViewDraw schematic**! No netlist editing and virtually no knowledge of arcane SPICE syntax is needed anymore. Finally, you can simulate using the same schematic program that you use for PCB layout.

The Integration Module adds custom SPICE attribute dialogs for each element, a quick part change and resimulate feature, SPICE model editing, special dialogs for SPICE analysis
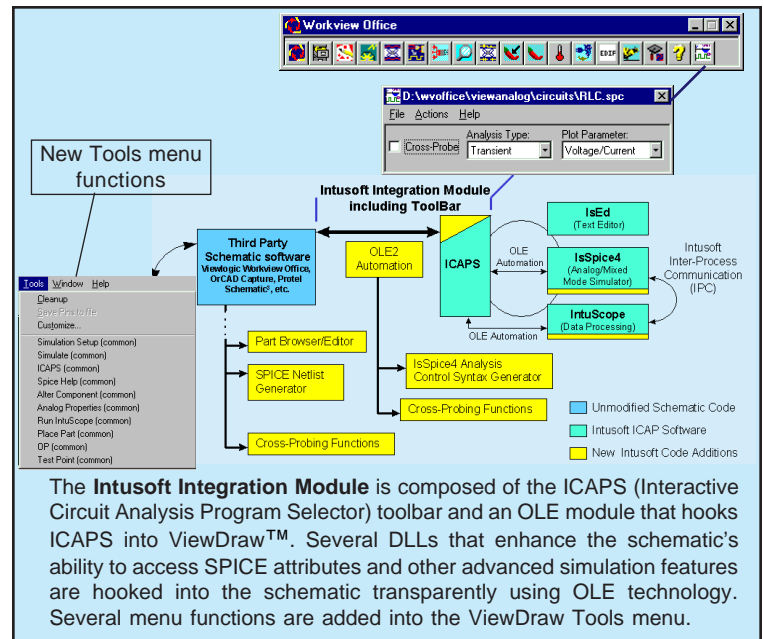


The **Intusoft Integration Module** is composed of the ICAPS (Interactive Circuit Analysis Program Selector) toolbar and an OLE module that hooks ICAPS into ViewDraw™. Several DLLs that enhance the schematic's ability to access SPICE attributes and other advanced simulation features are hooked into the schematic transparently using OLE technology. Several menu functions are added into the ViewDraw Tools menu.

statements, Monte Carlo analysis options, waveform and operating point cross-probing, and the ability to launch and control the simulator from the schematic. The result is that **ICAP**™ makes ViewDraw *VASTLY EASIER TO USE* for analog and mixed-signal simulation.

The **ICAP™ Simulation** toolbar gives you quick access to various simulation tasks such as netlist editing, interactive simulation options, and graphical data processing.

You can build schematics with symbols you have created in ViewDraw or use the full symbol library set provided with **ICAP**™. You can run an IsSpice4 simulation directly from ViewDraw via the Simulate function in the Tools menu. The simulation waveforms are displayed in *REAL TIME* as the simulation runs. There is no need to wait for the simulation to finish in order to see the results.

**ICAP**™ is *COMPLETELY INTERACTIVE*. If you click on a node or part in the ViewDraw schematic you will see the waveform pertaining to the selected item. This process is called cross-probing and it makes analyzing your design effortless. You may even cross-probe while the simulation is running! You can control the types of analyses (AC, DC, Operating Point or Transient) and the types of waveforms (node voltages, device currents, or device power dissipation) that you cross-probe. Full data analysis including cursor measurements, FFTs, and comprehensive waveform math are just a mouse click away.

**With the introduction of *ICAP*™ for Viewlogic, Intusoft finally brings integrated analog and mixed-mode simulation to Workview Office at a price you can afford, and with the ease of use you have come to expect from Intusoft.**

## ICAP™ Includes:

- IsSpice4, the Interactive Native Mixed-Mode (analog/digital) SPICE 3F/Xspice based simulator
- Bigger analog libraries than ANY other vendor, 10,000+ SPICE models (includes vendor IC models)
- IntuScope, the most powerful graphical waveform analyzer (FFTs, math functions, and more.)
- SpiceMod, the SPICE Modeling Program
- Full ViewDraw Symbol Library support
- The New Intusoft Integration Module
- Special RF or Power Supply model libraries

# SPECIAL DIALOGS FOR SPICE
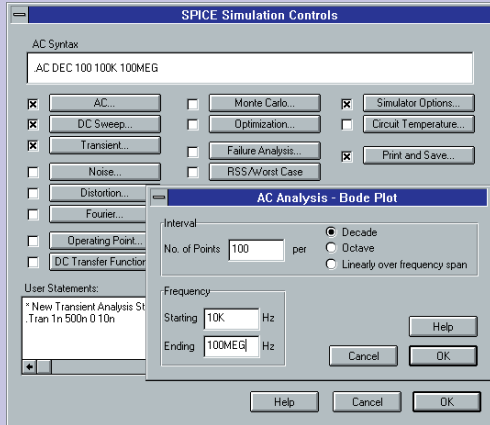


Custom SPICE dialogs make running an analysis painless and provide you with access to all of IsSpice4's powerful features.

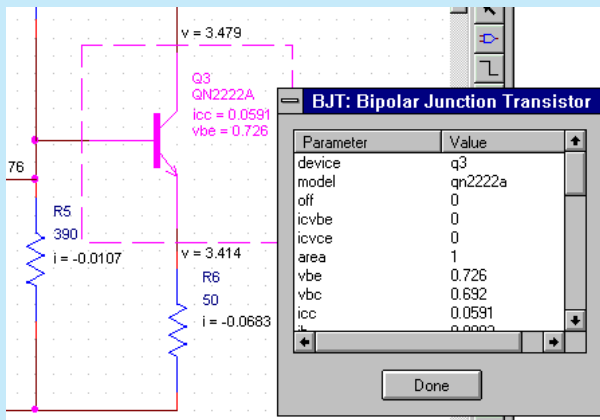**Benefits: No need to learn SPICE syntax.**

# QUICK CHANGE AND RESIMULATE

ICAP adds a simple yet powerful quick change feature to ViewDraw. With it, you can sweep any part value, or the circuit temperature, and automatically plot a curve family.
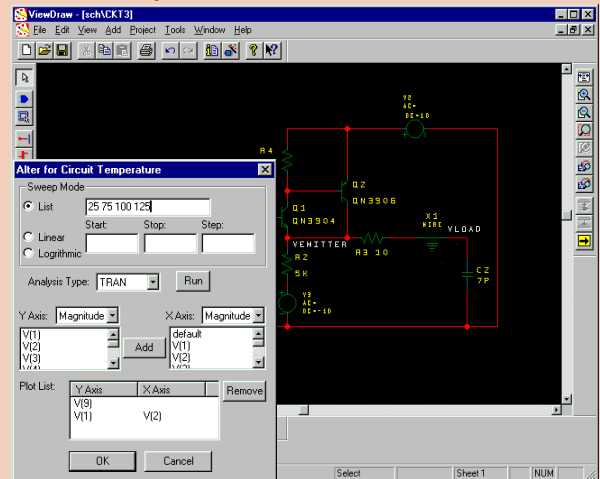


**Benefits: Optimize your designs with powerful parameter sweeping capabilities.**

# OPERATING POINT CROSS-PROBING



You can select different kinds of information to cross-probe. Here, in ViewDraw, a dialog displays each part's operating point information as the part is selected. The operating point voltages and currents are also displayed DIRECTLY on the schematic.

**Benefits: Easy data analysis and circuit debugging.**

# REAL TIME WAVEFORM DISPLAY
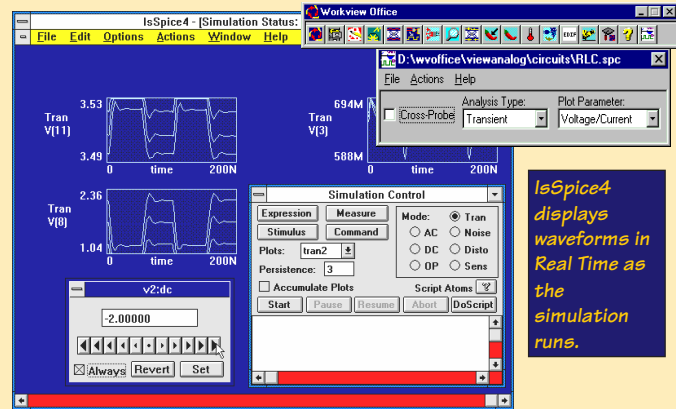


*IsSpice4 displays waveforms in Real Time as the simulation runs.*

The **ICAP™ Toolbar** (top right) provides ViewDraw users with easy access to model editing, SPICE simulation, cross-probing options, Monte Carlo options, and real time waveform processing.

**Benefits: Instant simulation feedback. No need to wait!**

# CROSS-PROBING VOLTAGES, CURRENTS, AND POWER

The Integration Module allows you to click on any node or device in the ViewDraw schematic. The corresponding node voltage, device current, or device power dissipation waveform will be immediately displayed. The combined ICAP™ for Viewlogic system is completely interactive. At any time, you can analyze simulation results or cross-probe waveforms, <u>even while the simulation running</u>.
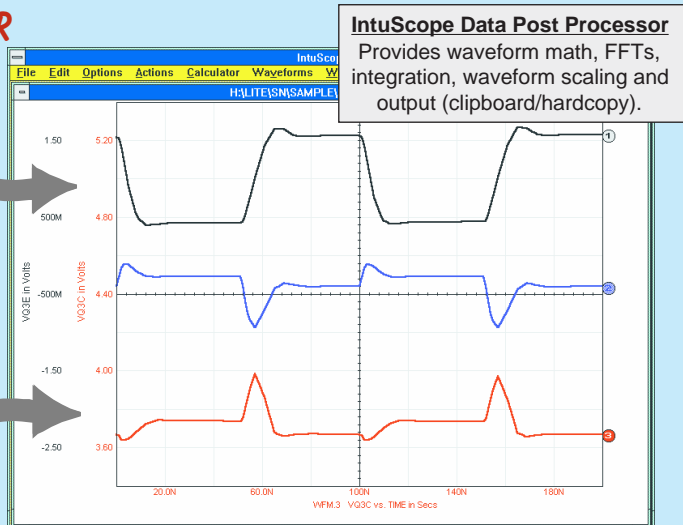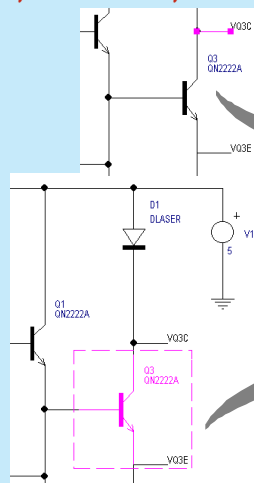
**IntuScope Data Post Processor**
Provides waveform math, FFTs, integration, waveform scaling and output (clipboard/hardcopy).

# *ICAP*™ for Viewlogic

## *Brings state-of-the-art simulation power to your desktop*

**ICAP** includes IsSpice4, the first commercially available version of SPICE based on Berkeley SPICE 3F.5 and Georgia Institute of Technology's XSPICE. IsSpice4 provides a quantum leap in performance over other analog and mixed-mode simulators. With the ability to simulate electrical, sampled-data, mechanical, physical, and thermal elements, **IsSpice4 is the ONLY true interactive native mixed-mode SPICE 3 based simulator.**

Intusoft has spent hundreds of man-hours improving SPICE. And although IsSpice4 is based on SPICE 3, Intusoft has greatly enhanced the program over and above the public domain version. We have added more features, an interactive graphical user interface, superior analysis and model support, and improved convergence algorithms - All for a price no one can match. IsSpice4 is simply the best and most affordable SPICE based simulator on the market today.

## *AT LAST!! TRANSPARENT SIMULATION DIRECTLY FROM THE SCHEMATIC*

| Feature | Benefit/Description |
|---|---|
| Launch a simulation directly from the schematic | Easily run a simulation (with a single menu function) without learning a new program interface |
| Cross-probe simulation results from schematic or layout | Quickly display the correct circuit waveforms; Display voltages, currents, power, and operating point info by clicking on a node or schematic symbol |
| Symbol Library Support | Symbols are provided for all SPICE models |
| Support for ALL SPICE Analysis statements and part attributes | No need to learn SPICE syntax; Special Help dialogs for All SPICE analyses and elements are provided |
| Real Time Waveform Display | IsSpice4 displays instant feedback as the simulation runs. You don't have to wait for the simulation to finish in order to see the results |
| 2 simple steps to change a value and resimulate | Seamless integration makes it easy to see the results of a value change |
| Complete SPICE netlist generation | No need to edit the SPICE netlist, Schematic and simulation always agree automatically |
| Parameter Passing | Generic models/subcircuits are supported |
| Monte Carlo support | Tolerance/Yield and Worst Case analysis are supported |
| Support for AHDL Models and user defined C code subroutines | Special attribute dialogs for user generated models allow ASIC, Board and System level designers to create custom SPICE models |

**For More Details** about the components of ICAP including IsSpice4, IntuScope, SpiceMod, and the Intusoft SPICE Model Libraries, please contact Intusoft, your local Intusoft or Viewlogic dealer, or Intusoft's Web Site.

## *Hardware/Software Requirements*

- x86, Windows 95 or Windows NT
- 16 megabytes (32 megabytes recommended)
- ViewDraw 7.3 or greater

## *Different ICAP Options*

**ICAP**™ for Viewlogic is available in two versions. The **RF Designer's version** has a special SPICE library for RF parts, while the **Power Supply Designer's version** has a special SPICE library for power electronic parts. **This is the only difference.**

- **RF Model Library -** A specialized SPICE Model Library containing over 600 RF parts (BJTs, MMICs, Beads, Mesfets and More).

- **Power Supply Designer's Library -** A specialized SPICE model library containing over 400 parts including PWM ICs, Magnetic Cores, PFC ICs, and more.

## *Product Support*

*Other Vendors talk About Support... Intusoft Delivers!*

- *Intusoft Newsletter -* Industry Standard SPICE publication with application notes and models.

- **FREE SPICE Models -** The Intusoft technical support staff makes SPICE models FREE of charge for registered customers.

- **Web/Internet -** Intusoft maintains a home page on the World Wide Web and a forum on the CompuServe® Information Service.

- **30 Day Money Back Guarantee**

## *Additional Support Products*
- **SPICE Reference Books - "Simulating With SPICE", "SPICE Applications Handbook", "SMPS Simulation With SPICE 3", "A SPICE Cookbook"**

- **Tutorial Classes and Training -** Intusoft offers full on-site training by the engineers who developed the software, not a third party source like other EDA companies offer.

**http://www.intusoft.com**

**intusoft**

P. O. Box 710
San Pedro, CA
90733-0710 **U.S.A.**

**(310) 833-0710 - Tele**
**(310) 833-9658 - Fax**
**sales@intusoft.com**

*the absolute leader in affordable simulation tools*