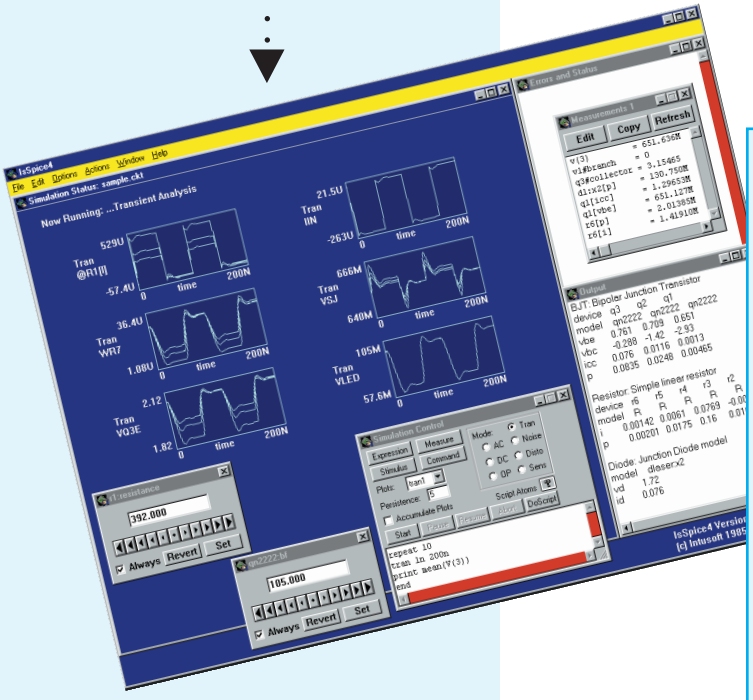
intusoft

Now EVERYONE can experience SPICE...

ICAP/4

The Virtual Circuit Design Lab



Because Intusoft Gives You

What's A Virtual Circuit Design Lab?

ICAP/4 is a complete circuit design and analysis package which features schematic entry, IsSpice4 (the native analog and mixed signal simulator), extensive device libraries, and powerful waveform processing, all integrated in one easy-to-use interactive environment. ICAP/4 is called "The Virtual Circuit Design Lab" because it can simulate all types of system, board, and IC level designs right on your computer. The designs can contain mixed mode circuits, sampled data systems, and mixed technology (electrical, mechanical, physical, etc.) elements - all within the same simulation, and all using a top-down simulation methodology.

Some SPICE programs claim to be easy-to-use. Some claim to be powerful. Intusoft software is BOTH.

If you're learning to use SPICE, Intusoft can show you how to get started on the path of least resistance. (We know how - we are the only SPICE vendor that offers SPICE training classes and publishes books on SPICE.)

If you're a power user, IsSpice4 will give you superior performance which will help you tackle the toughest analog and mixed signal circuits, and put you ahead of your competitors.

With features like design validation, failure analysis, AHDL models, OLE and Visual Basic scripting interfaces, IsSpice4 takes you beyond SPICE versions of the past and competing solutions of the present.

Since IsSpice4 is built with the latest 32-bit compilers and Win9x/NT interface guidelines, you can maximize your investment in new operating system software. Check out our competitors - their programs are years behind and still have DOS based remnants in their interfaces!

So whether you're a beginner or a power user, Intusoft gives you the combination of features you want at a price you can afford!

- Software and technology that truly increase your productivity
- New products that keep you ahead of the competition
- A company that is focused on simulation tools, NOT broad line EDA tool suites
- Service and Support that are proven and UNMATCHED

SpiceNet Schematic Entry

Intusoft believes the schematic features shown here are fundamental requirements for any competitive product. Some simulator vendors don't even offer a schematic, while some of these features are available on some products but not others. The combination of features listed on the following pages, along with those listed below, are not available from any other suppliers.

- Tight Schematic/Simulator Integration
- Hierarchical Buses
- Unlimited Undo/Redo
- EDIF output for Schematic/PCB interfacing
- AutoBridging (automatic analog <-> digital interface) with logic family definition
- Integration of different types of part data in a single schematic database (Monte Carlo, Fault, SPICE, packaging)
- Integration of different circuit design configurations in one schematic database
- Display of waveforms and other SPICE data DIRECTLY on the schematic and updated during the simulation
- Hierarchical push and pop to/from subcircuits
- Internet/Intranet based SPICE simulation
- Mechanism for simulation of multiple sets of SPICE analyses and automatic recording of the results

Underlying these features is the Intusoft ActiveX™ architecture, which enables our products to span the Internet and Intranet, and provide super computer power through a distributed simulation architecture.

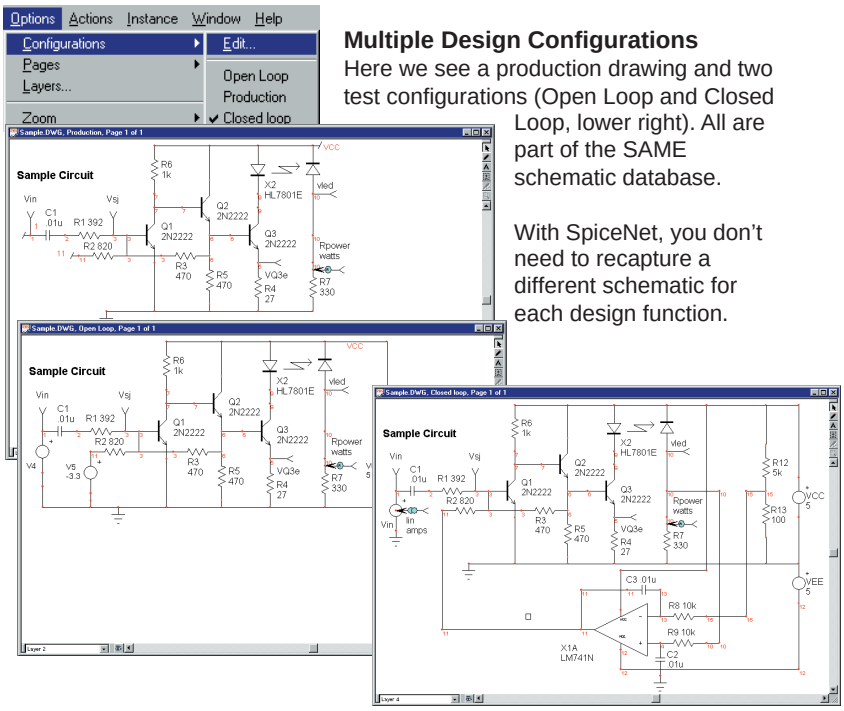
Discussed next are just some of the unique features in the SpiceNet schematic entry package. A fully functional version of SpiceNet is posted on the Intusoft Web Site at *WWW.INTUSOFT.COM*. **You can download it and give SpiceNet a try!**

Ease-of-Use and POWER to grow

ICAP/4 gives you everything you need to do your design job, quickly and easily. At the heart of the ICAP/4 system is the SpiceNet schematic entry program. SpiceNet works seamlessly with the IsSpice4 circuit simulator, giving you a fast, easy-to-use solution for capturing, simulating, and debugging analog and mixed signal circuits.

Full-Featured Schematic Editing

- Multiple design configurations allow you to place design variations and production drawings in a single schematic database
- Multiple schematic layers allow you to reconfigure and resimulate your designs quickly and easily
- Hierarchical drawing capability with automatic subcircuit creation and graphical navigation
- Mechanical data entry capability allows you to use a single schematic for simulation and PCB layout
- Full window cut/copy paste functions enable you to easily reuse existing designs
- Unlimited undo/redo functions save you time and help you avoid costly mistakes
- On-line help with an comprehensive SPICE reference guide and Intusoft's unique Multimedia Movies guide you through the simulation process
- Clipboard copy function allows you to include your design in MS Word or other Windows applications

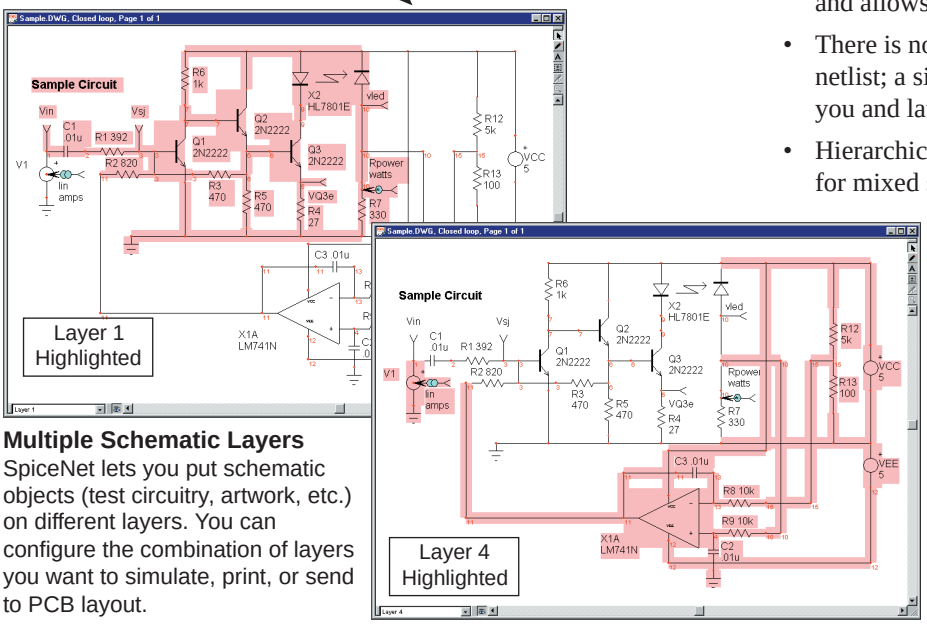


Multiple Design Configurations
Here we see a production drawing and two test configurations (Open Loop and Closed Loop, lower right). All are part of the SAME schematic database.

With SpiceNet, you don't need to recapture a different schematic for each design function.

Time Saving Features

- Auto-label placement makes it easy to capture a good-looking schematic rapidly
- Automatic on-the-fly connectivity saves you time - there's no need to place connectivity dots
- Intelligent part attribute editor simplifies data entry and allows you to edit any part graphically or via a list
- There is no need to separately generate a SPICE netlist; a simple menu function does everything for you and launches the simulation right from SpiceNet
- Hierarchical Bus support gives you added flexibility for mixed signal designs



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 to PCB layout.

Interactive Symbol Editing

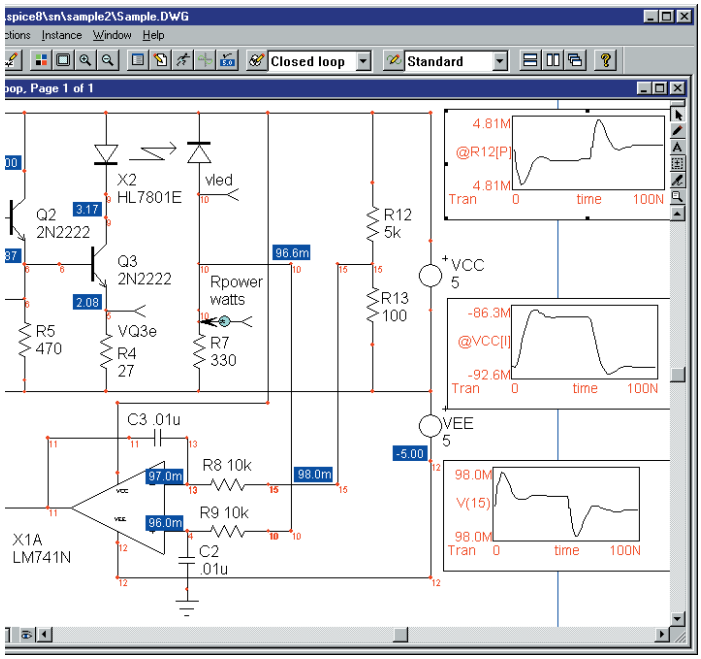
- On-line symbol editor and Symbol Wizard make it easy to add or reconfigure symbols
- Quick symbol modify and replace function gives you the schematic appearance you want
- Symbol shapes including arcs, ellipses, Beziers, polylines, fills, metafiles, and colored bitmaps enable you to make production schematics that stand out

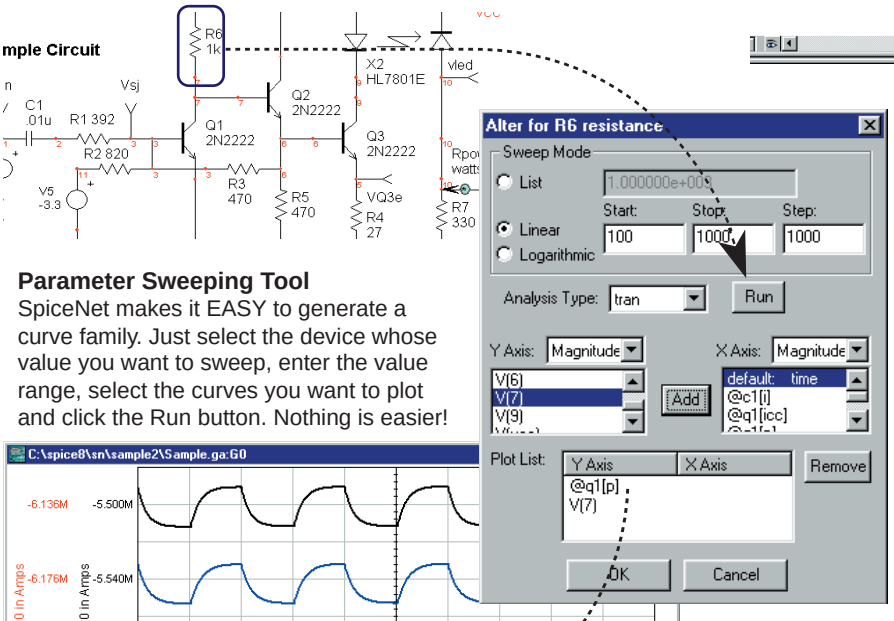
Understand Your Design
with tools that perform

While other EDA vendors claim to offer the benefits listed here, only Intusoft has the product which **REALLY delivers the results**. Combining the features shown here with a user-centered interface, Intusoft provides a combination of products and services that are unbeatable and truly surpass other competitor's offerings. If you're new to SPICE, there's only one company you can rely on to give you the assistance and productivity you need. That's Intusoft.

Unequaled SPICE Power and Ease-Of-Use

- Parameter Sweeping tool allows you to sweep the value of any component, or the circuit temperature, and view a curve family of the results
- Perform linear, logarithmic, or list-based sweeps and plot any waveform
- Enter data for all SPICE analyses using "fill-in-the-blank" dialogs; there's no need to know SPICE syntax
- Change any model parameter in the design or add a new model based on an existing device
- Monte Carlo tolerance dialog lets your schematic database hold custom tolerances for all parts
- Support for ALL vendor supplied SPICE models, regardless of format or syntax variations
- Simulation fully controlled from any task (schematic, text editor, waveform viewer, and simulator)





Parameter Sweeping Tool
SpiceNet makes it EASY to generate a curve family. Just select the device whose value you want to sweep, enter the value range, select the curves you want to plot and click the Run button. Nothing is easier!

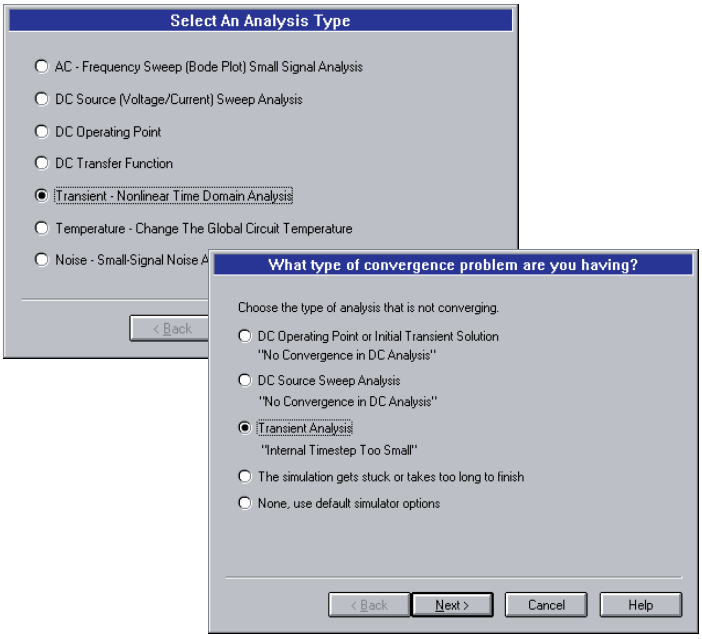
Superior Waveform Cross-Probing

- Cross-probing lets you select a net and see the trace displayed right on the schematic or in the IntuScope waveform analyzer
- Resizable waveforms can be placed directly on the schematic; no separate display dialogs or programs are needed
- Test point symbols let you see current or power from any device or any node voltage.
- Detailed or brief operating point information can be displayed on the schematic
- Cross-probe node voltages, device currents and power dissipation
- Cross-probe data from the AC, DC, or Transient analyses
- Print or copy schematics with waveforms and operating point data to the clipboard; great for documentation purposes

NO NEED to Know any SPICE syntax

SpiceNet is specially designed for use with the IsSpice4 simulator. It contains many enhancements that make simulation and analysis effortless. NO schematic is easier to use for analog and mixed signal circuit simulation.

Analysis and Convergence Wizards



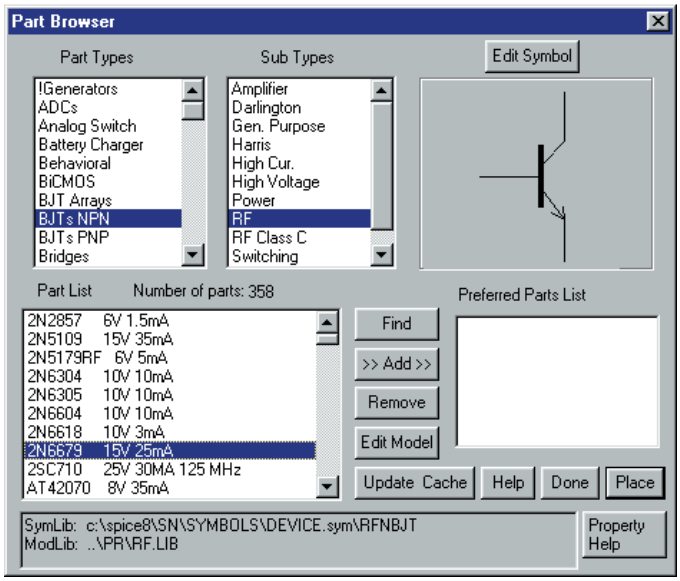
Two new powerful Wizards are included in SpiceNet. Together, they eliminate the SPICE syntax complexities that engineers have complained about for years.

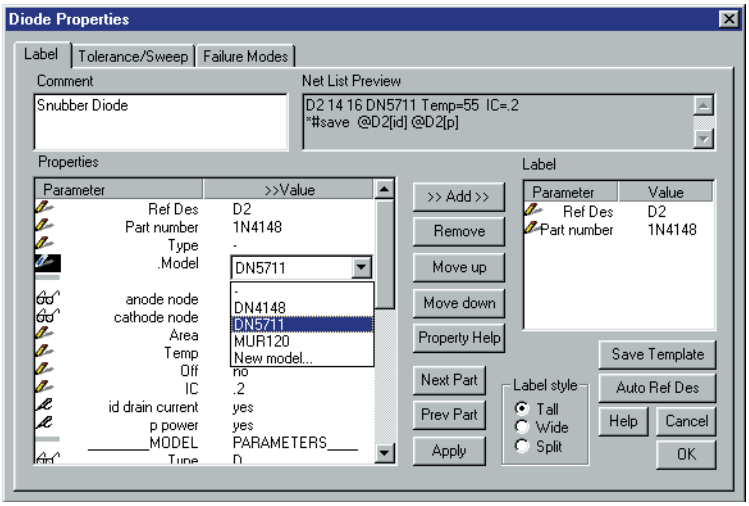
The Convergence Wizard makes it easy to solve your toughest convergence problems without having to know SPICE syntax. Based on your answers to simple questions, the Convergence Wizard intelligently optimizes key .OPTIONS variables for you.

The Analysis Wizard helps novice and power users specify all key SPICE analyses without the use of a SPICE reference guide.

Easy Part Selection

- Easily find any of the 14,000+ parts in the ICAP/4 model libraries using the hierarchical Graphical Parts Browser
- Search by part number, model name, part type or description
- Access model or symbol libraries from any drive or directory
- No need to preselect symbol or model libraries; all parts are available, saving you time and search hassles
- Preferred Parts list lets you get to frequently used parts from a pull-down menu





Powerful Part Editing Features

- Edit All SPICE instance and model parameters from a single properties dialog (shown left)
- Versatile part labels allow you to configure custom schematic labels
- Enter part attributes, Monte Carlo tolerances, and part failure modes in the same dialog
- Edit subcircuit netlists right from the schematic
- Special attribute support for AHDL models
- Add your own dialog features through the DLL interface (source code provided)

Precise, Accurate, Fast
these are the words

IsSpice4 provides a quantum leap in performance over other analog and mixed mode simulators. It is the first commercially available version of SPICE based on Berkeley SPICE 3F and XSPICE.

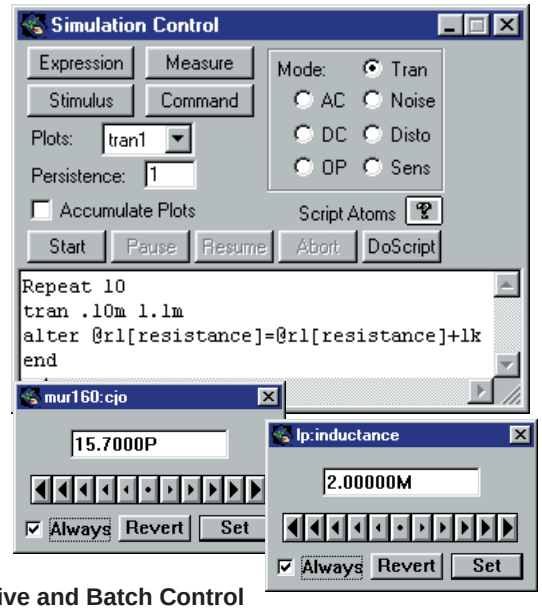
IsSpice4 allows you to explore circuit performance by interactively running different analyses and sweeping any circuit variable. With the ability to simulate electrical, sampled-data, mechanical, physical, thermal, and other systems, **IsSpice4 is the ONLY true native mixed mode SPICE 3 based simulator.** The advanced features of IsSpice4 allow all types of applications to be simulated: switch mode power supplies, mixed signal ASICs, RF communication systems, interconnects, control systems, and mixed mechanical/physical systems.

IsSpice4 - State-of-the-Art SPICE

- Full SPICE3 and XSPICE compliant simulator with numerous interface and algorithmic enhancements
- Native mixed mode simulation - IsSpice4 includes an event driven logic simulator that supports mixed analog, digital and DSP circuitry
- Tight coupling with the SpiceNet schematic entry tool and the IntuScope waveform analysis tool
- Interactive Operation - IsSpice4 operates interactively, and frees you from the restrictive batch style of older SPICE simulators
- Interactive Command Language - Comprehensive set of functions for batch style control of the simulator
- **Visual Basic Scripting** - Drive IsSpice4 using VB scripts from popular programs like Excel
- **ActiveX Interface** - Develop your own OLE/ActiveX interfaces

IsSpice4 Elements and Models

- Passive Elements: Resistors, Capacitors, Inductors, Coupled Inductors, Transmission Lines (Ideal, Lossy, and Distributed RLCG), Switches with hysteresis, Independent sources
- Active Elements: Diodes, BJTs, JFETs, MOSFETs (Level 1-8), GaAs Mesfets
- Digital and AHDL Models: Digital primitives, State Machine, Frequency Divider, RAM, Sampled-Data Filters, Nonlinear VCOs, Laplace Equations
- Behavioral Modeling: In-line math equations, Table models, If-Then-Else statements, Boolean logic expressions, Dependent sources
- Pspice® parameter passing syntax compatibility



Interactive and Batch Control
IsSpice4 gives you complete interactive control over the simulation process. Any circuit variable can be swept, individually or in groups. You can even write your own scripts to drive the simulator to perform multiple analyses, measurements, and value changes.

Advanced Models

- AHDL Models and C Subroutines; Create models based on a powerful nonproprietary AHDL using C
- Support for nonelectrical applications and top-down design
- Models that interface with files (wav, pwl), the operating system, or hardware (analog output boards/data acquisition boards)
- Three types of digital/mixed mode modeling
- Lossy (distributed) transmission lines with frequency dependent losses (skin effect and dielectric loss)
- MOSFET: BSIM1, BSIM2, BSIM3 version 2, 3.1, and 3.2 and SOI models
- MESFET: Statz, Curtis-Enttenburg, Parker-Skellern, and HEMT

Convergent, Solid, Compatible
that describe IsSpice4

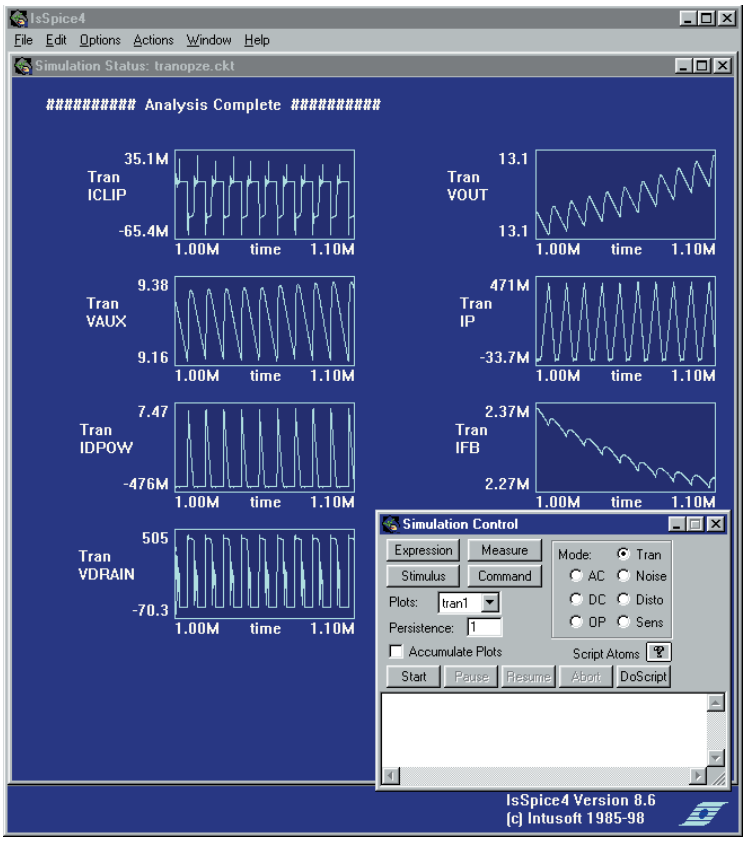
Intusoft has spent hundreds of man-hours improving IsSpice4. And although IsSpice4 is based on SPICE 3, Intusoft has greatly enhanced the program over and above the public domain version; adding an interactive interface, providing superior analysis and model support, and improving the convergence algorithms - all for a price no one can match. IsSpice4 is simply the best and most productive SPICE program on the market today.

Analysis Support

- AC, DC, transient, noise, Fourier, distortion, DC/AC sensitivity, Impedance, transfer function, and Pole-Zero analyses, plus Temperature variations on individual elements
- Monte Carlo Analysis, Circuit Optimization/Performance Analysis
- **RSS, EVA (Extreme Value), Sensitivity and Worst Case** measurements for AC, DC, Operating Point and Transient analyses
- **Design Validation** for automatic design verification
- **Simulation Templates** for architecting your own simulation procedures
- **Test Designer™** - Fault Analysis and Software Test Set Design
- **SpiceFarm™** - Internet/Intranet SPICE

Additional Interactive, AHDL & Mixed Mode Features

- Real-time display of voltage, current and power dissipation and hundreds of other circuit quantities
- Set simulation breakpoints and Stress Alarms to automatically monitor circuit conditions
- Interactively run analyses without having to edit the netlist or restart the simulator; add, delete, or rescale waveforms on the real-time display
- Sweep parameters one at a time or in groups with great ease
- Start, stop, pause, change, or resume any analysis on demand



IsSpice4's Real Time Display
The Real Time display allows you to see what's happening with your design during the simulation. Any circuit quantity can be displayed.

Convergence and Speed Improvements

- Automatic Gmin stepping/Source stepping algorithms
- Pseudo-transient algorithm UIC algorithm
- Improved predictor-corrector, latency, and bypass algorithms
- Improved program defaults
- Special circuit debugging options
- Full Gear integration option

Versatile Design Validation: What YOU always

The *Design Validation* feature, found in ICAP/4Windows, ICAP/4 Professional, and Test Designer, keeps track of multiple circuit configurations, analysis setups and test measurements. It then runs an entire suite of tests automatically and creates a report that summarizes the results and pass-fail grades. This powerful capability extends SPICE well beyond its current single simulation oriented boundaries.

How Design Validation Works

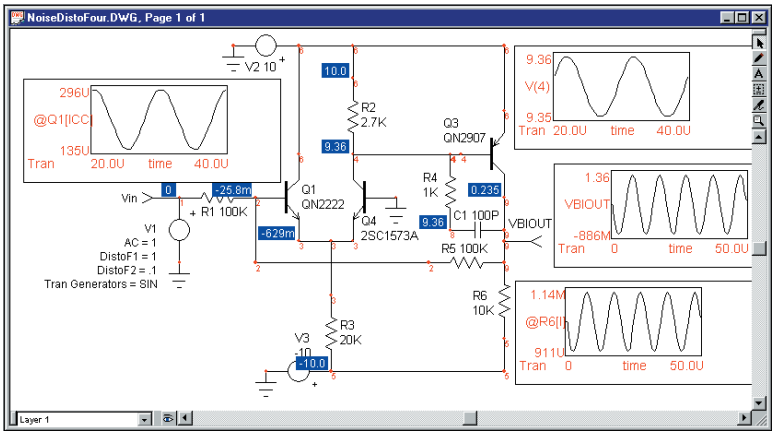
You begin by using ICAP/4 to capture different circuit configurations or variations. For example, one version may have a different set of values from another, or different test circuitry. In the schematic shown below, two configurations have been captured; one with a sinusoidal generator (Config1, shown below) and one with a square wave generator (Config2, not shown). Each circuit configuration can be assigned one or more SPICE analyses. The combination of an analysis and a configuration is referred to as a "Test".

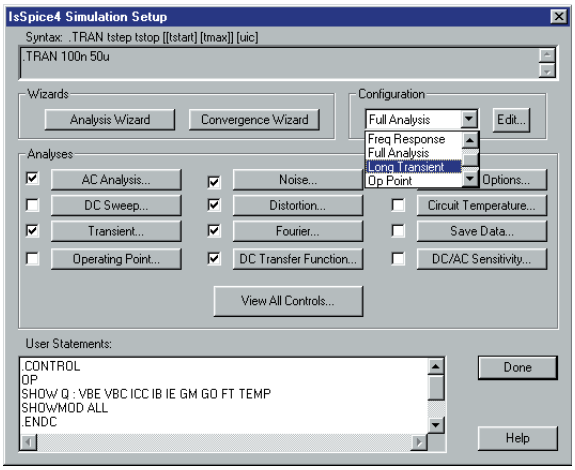
You then assign one or more measurements to each test. The measurements can be made on any number of nodes or devices, and are setup using an easy-to-use Wizard approach (shown lower right).

A summary of the tests and measurements for this example is shown on the next page. ICAP/4 automatically simulates the multiple test configurations you have created, and returns the measurement results to SpiceNet. The results are organized into a report containing the measured (simulated) result and the pass/fail limits which are used to grade the performance.

ICAP/4 runs all the simulations and processes all the results WITHOUT the need for any user interaction.

The report (next page) shows which tests passed or failed based on tolerances assigned to measurements taken in a nominal run. Different pass/fail limits can be easily assigned, but defaults are inserted for you. The upper report shows a nominal case. The lower reports shows the pass/fail grades on the operating point after the resistor R2 is changed to 5K.



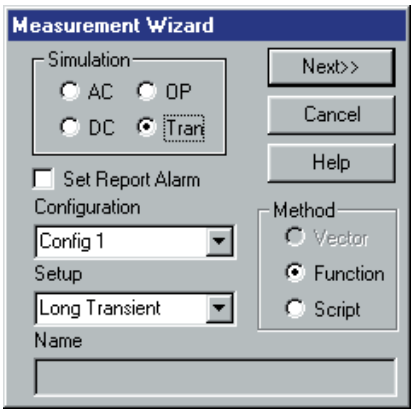


Multiple Simulation Setups

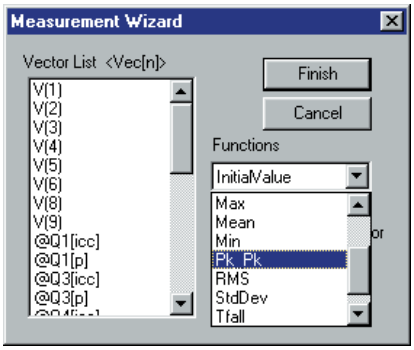
The Simulation Setup dialog makes specifying SPICE analyses easy. Just click on the desired analysis button and fill in the simple dialog. All of the syntax details are handled for you. SpiceNet even allows you to save several sets of analyses. You can then combine the different analysis groups with various circuit configurations to make multiple test configurations.

Measurement Wizards

It's easy to specify the measurements you want to make for each test. Just point and click to record the data you want. Advanced users can take advantage of IsSpice4's powerful ICL scripting capabilities to make specialized measurements.



All of the commonly required measurements are available, including cursor and event-based measurements.



wanted SPICE To Be able to DO.

Uses and Benefits

- Automatically validate and evaluate circuit changes against design specifications
- Easily simulate and verify multiple circuit configurations
- Provide a framework for automatic test set software design
- Make automated measurements, under a variety of conditions, without manual interaction
- Identify the components that are outside of their 'safe operating area'

Features

- Report with pass/fail indicator shows you instantly how far your measurement is out-of-tolerance
- User-defined pass/fail limits may be asymmetrical; tolerances can be in absolute terms or percent
- Freely define different test configurations and scenarios
- Automatically perform a variety of measurements on each test configuration
- Quick edit allows you to make circuit changes and resimulate to see the results

Engineers are under pressure to achieve better quality designs which can be brought to market faster and at a lower manufacturing cost. The need to check the robustness of a design now extends beyond the verification of nominal system performance, into areas such as the impact of component tolerances and failures, yield, design changes, and manufacturing options.

Your job demands require that you make the best design within a given set of specifications, and that you respond faster to changes in requirements.

To support these tasks, it is essential for the simulation technology to be able to facilitate the production of tolerance and failure independent systems for which critical aspects of the design can be identified prior to prototyping.

Through wide-ranging and automatic testing, Intusoft's ICAP/4 helps to ensure the robustness of a design and allows you to realize the substantial benefits that can arise out of early yield, failure, and traditional SPICE analysis. The Design Validation feature enables any part of your system to be tested and verified under any condition, including out-of-spec performance variations based on any design change or variation.

With ICAP/4, test strategies can be improved and debugged by enabling virtual production studies to be undertaken without the need to build a prototype or even wait until the design process has been completed. ICAP/4's Design Validation feature provides a productivity boost over manual analysis, and dramatically reduces the time-consuming evaluation of large volumes of data.

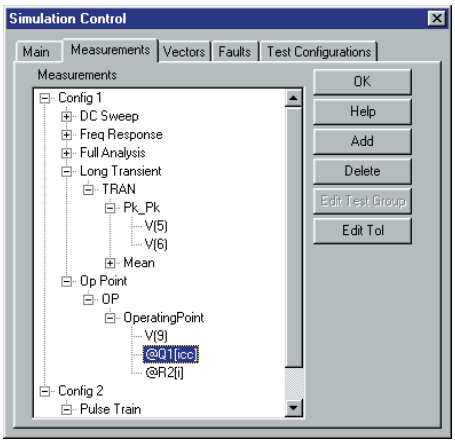
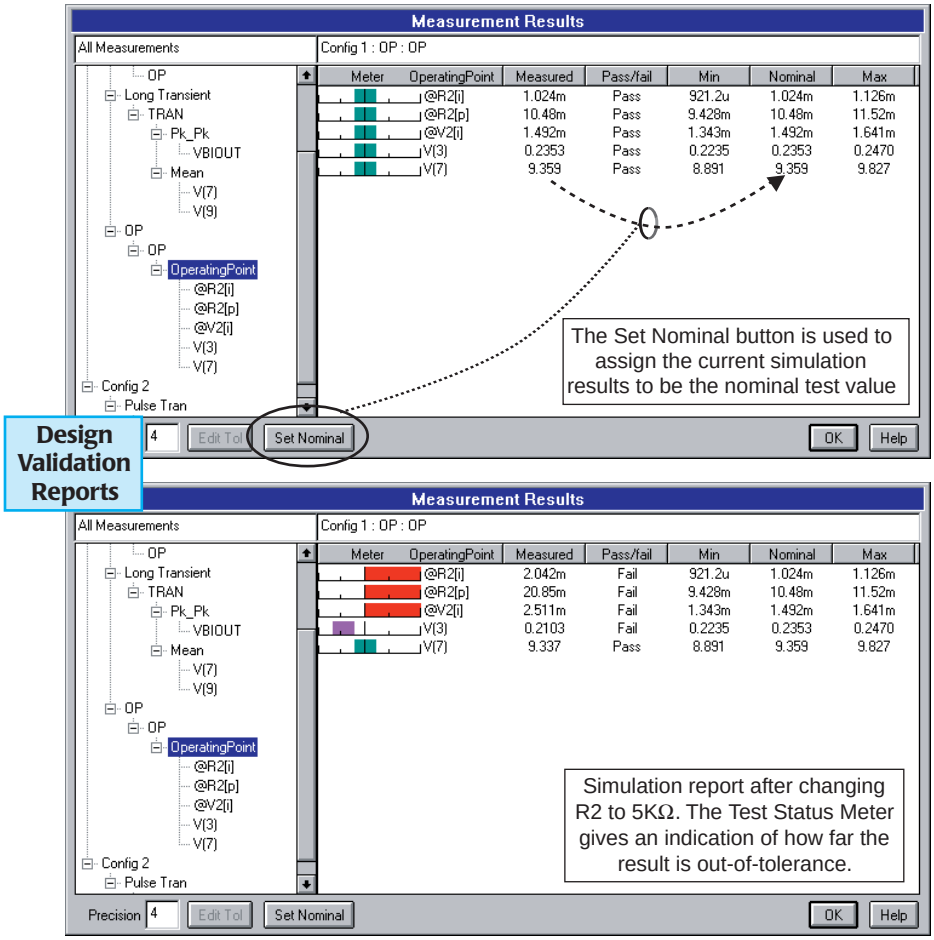


Table of Measurements
The above table illustrates the pairings of analyses and measurements with a configuration in order to define a test.



Power Supply Designer's Library specialized Models, tools,

SMPS applications today are much more demanding than ever. Today's designs require increases in switching frequency, higher efficiency and lower standby current. State space based models simply do not reveal many important nonlinear factors that influence these performance characteristics. To address the needs of today's power supply designer, Intusoft has introduced a special SPICE model library. It contains magnetics models and a comprehensive set of large signal cycle-by-cycle simulation models for Pulse Width Modulation (PWM) ICs. The library represents a major breakthrough for SMPS designers.

State Space Is Just the Beginning

The power library includes a "unified" state space PWM model that works for AC, DC, and transient analyses in continuous and discontinuous modes. However, the switch level models give designers a capability they have never had before; the ability to plug in a model, representative of the actual IC, which simulates the total switching performance.

Advanced Model Flexibility Gives You Control

Nonlinear PWM characteristics such as propagation delay, switching speed, drive capability and maximum duty cycle/current limits are all accurately modeled. You can directly compare the performance of components from different vendors and analyze the effects of different implementations such as peak current mode control, hysteretic current control, low voltage, and low operating current, to name just a few.

Models Include:

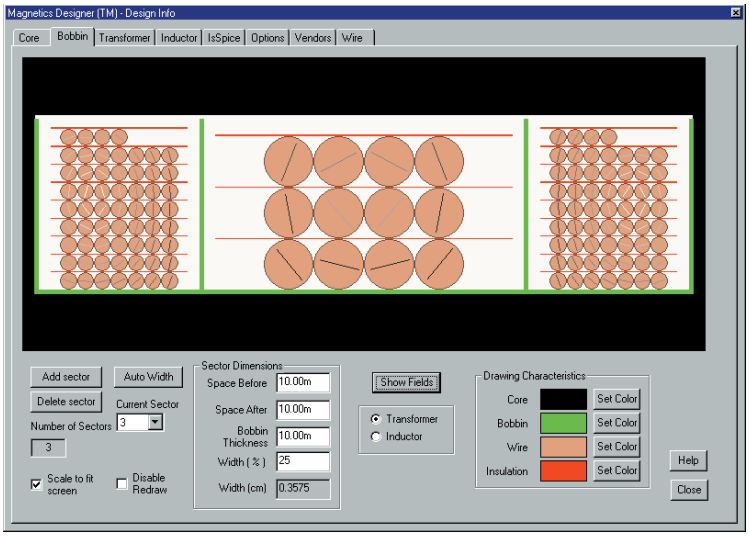
- Over 28 PWM IC models with full switching accuracy
- 'Unified' state space models
- Nonlinear Magnetic Cores
- Transformers
- Power MOSFET Drivers
- Other ICs: Motor controller, Power factor correction, Regulators

Power Supply Designer's Model Library

- Analyze large signal effects like start-up transients, power stage semiconductor stress, and step-load response
- Explore different approaches to transformer, converter, filter, and control structures
- Compute component stresses and test for excessive power dissipation
- Compare circuit characteristics with linear and nonlinear magnetics
- Analyze in both time and frequency domains
- Simulate and analyze your entire power supply without ANY limitations.

Magnetics Designer

Transformer and Inductor Design Made Easy



Magnetics Designer Tool

- Synthesizes designs for any type of layer or sector wound transformer or inductor
- Analyzes and reports all key performance criteria
- Generates an accurate and detailed SPICE model of the magnetic, including all parasitics
- Produces a winding sheet and performance report with all operational characteristics
- Comes with a database of over 7,000 cores
- DC to RF, Planar to Telephone pole
- Uses a combination of analytical and FEA solution algorithms to optimize and analyze the magnetic design for you
- Synthesis capability saves countless hours and makes your design time more productive

RF Device Library and Applications Support

Accurate Models Enable Your RF Simulations

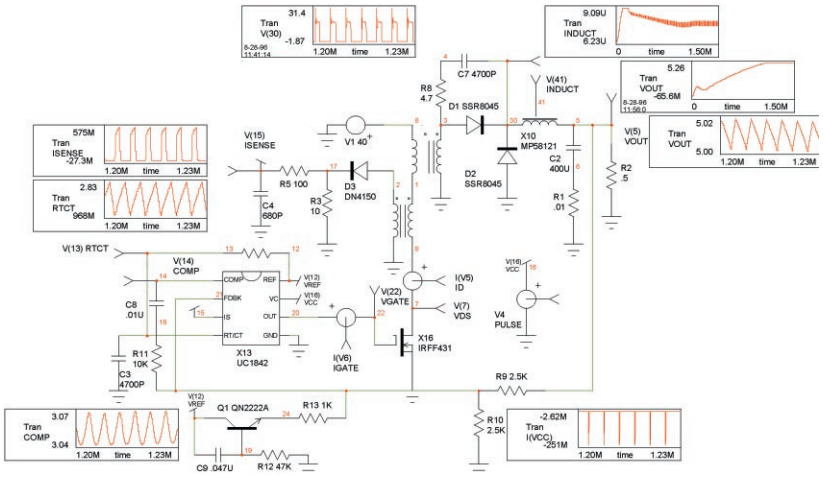
Because of the difficulties in creating SPICE based RF transistor models, most SPICE vendors have been unable to provide accurate models at a reasonable cost. This has made simulation for RF circuits difficult, if not impossible, for the average RF engineer.

Using special custom software and new optimization techniques, the modeling problems associated with developing RF device models have been solved. Unlike other RF SPICE models, the Intusoft models do not use unrealistic parameters to “force” fit device behavior to the standard Gummel-Poon model. A custom subcircuit approach is utilized to account 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. Intusoft is one of the only CAE vendors to offer specialized SPICE models for Power and RF devices.

RF Device Library

- Models can be used with any Berkeley SPICE compatible simulator
- BJT models can be used in both transient and small-signal linear simulations
- Over 600 RF models included
- Wide variety of device types covered
- RF Bipolar transistors, RF beads, PIN diodes
- MMICs, JFETs, Power MOSFETs,
- GaAs MESFETs, and ideal couplers



Many types of switched mode power supply and RF/Microwave simulations are possible with ICAP/4 software.

SpiceMod: SPICE Modeling Made Easy

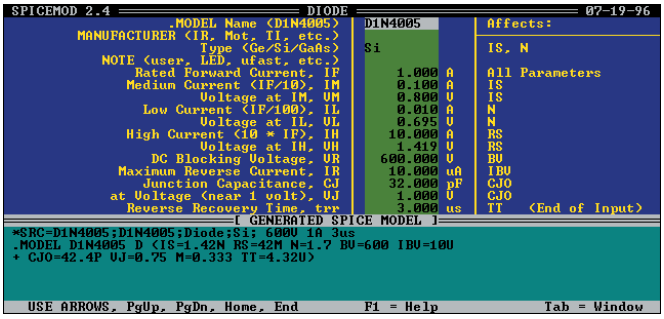
Making Models From Data Sheets

Finally, there is a simple program available to alleviate the difficulties of SPICE model development. SpiceMod, the SPICE modeling spreadsheet, gives you the power to create an unlimited number of SPICE models for thousands of semiconductors. With SpiceMod, you won't have to stop your simulation activities because you don't have a model, you won’t have to spend countless frustrating hours learning the intricacies of SPICE model parameters, and you won’t have to make any laboratory measurements. All you need is the manufacturer’s data sheet and just a few minutes of your time.

SpiceMod produces accurate models that can be used with any Berkeley SPICE compatible program. SpiceMod is also integrated with ICAP/4, Intusoft’s analog and mixed signal circuit design system. The models and subcircuits created in SpiceMod can be immediately used in your SpiceNet schematics and IsSpice circuit simulations.

SpiceMod makes models for:

- Diodes, Zeners, BJTs, power BJTs, Darlington BJTs,
- JFETs, MOSFETs, power MOSFETs,
- SCRs, GTOs, Sidacs, Triacs, and IGBTs



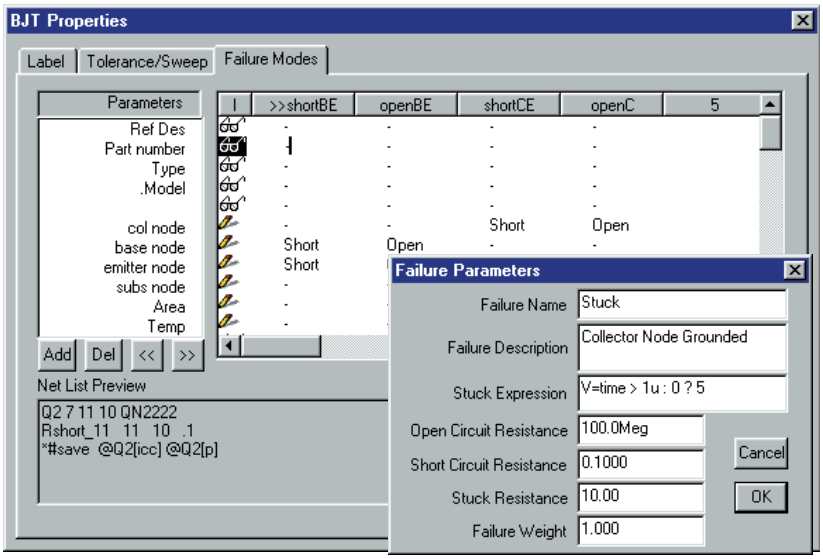
SpiceMod takes in manufacturer’s data sheet parameters and generates SPICE models.

Powerful FAILURE analysis makes your designs BETTER

The Failure Analysis features included in ICAP/4 provide powerful and timesaving functions which are invaluable to design, test, and reliability engineers. You can select any part and choose from a predefined failure mode list or define your own failure modes. ICAP/4 will automatically alter the simulation model for your part without changing the appearance or connectivity of the schematic. There's no need to edit any SPICE subcircuits or to insert wire shorts, resistive opens, etc. in the schematic. ICAP/4 takes care of all of that for you. You can then analyze the effects of hard faults, out-of-tolerance conditions, and anomalous device behaviors. With this knowledge, you can define test requirements, strategies and diagnostics.

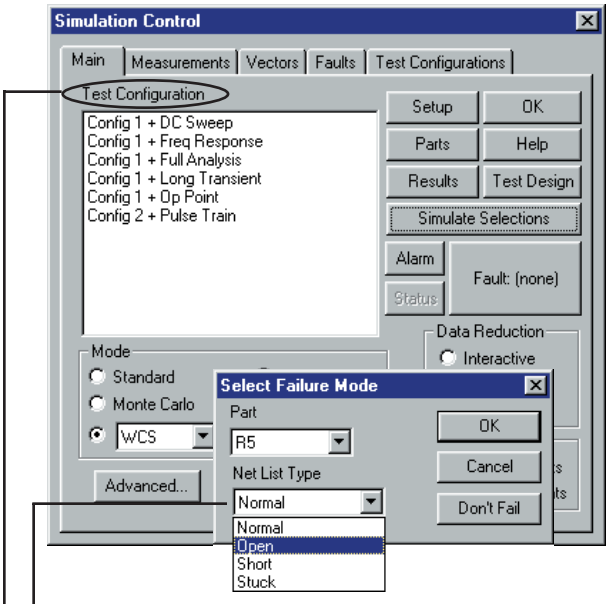
Failure Analysis

- Includes Stress, RSS, Extreme Value, and Worst Case analyses
- Simulate failures, out-of-tolerance conditions, or other user-defined part behavior ("what-if" scenarios)
- Provides a basis for test software development
- Helps troubleshoot circuit malfunctions
- Select Failure Mode dialog allows you to quickly select any defined fault for any component and run a simulation
- When linked with ICAP/4's Design Validation feature, you can perform "what-if" scenarios and make multiple automated measurements
- When linked with Design Validation, you also get a report of the pass/fail statistics and a visual test status meter (see Design Validation report, page 9)



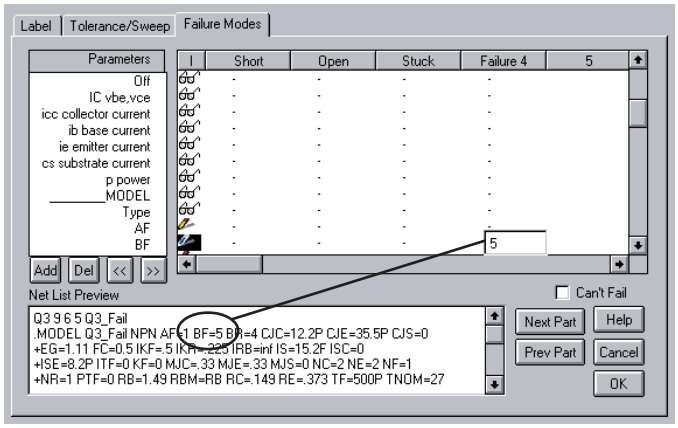
Fault Mode Definition

- Special fault definition dialog contains predefined part failure modes as per the CASS standard
- A variety of faults can be defined and inserted in any analysis
- Failure Parameters dialog allows you to define open, short, and stuck circuit resistance values
- User-defined faults can include expressions of time and other circuit parameters
- All SPICE attributes, including component values, model parameters, and subcircuit connections, can be faulted or set out-of-tolerance



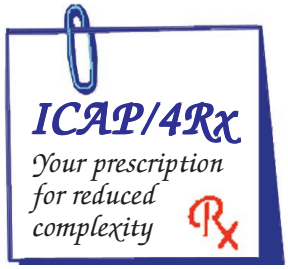
Simulation and Failure Selection
The Failure Mode dialog makes it easy to select and simulate the effects of any failure mode for a particular part in your design.

The new Simulation Control dialog allows you to define various test configurations (combinations of circuit configurations and IsSpice4 analyses). You can then simulate one or more test configurations interactively or in batch mode.



User-Defined Faults
The Failure Modes definition dialog in SpiceNet allows you to define your own faults. Here a BJT model is given a low beta. Note that the corresponding .Model statement is generated automatically.

cost effective ICAP/4Rx, Priced Right Not under Powered



Only Intusoft GIVES you ALL the features you NEED at a PRICE you can AFFORD!

If you've been looking to upgrade your evaluation version of SPICE, or are thinking about getting started with SPICE, we'd like to suggest a system that's easy to learn but powerful enough to tackle all of your design jobs. It's called **ICAP/4Rx** and it's a complete circuit simulation system that includes everything you need to simulate all types of system, board, and IC level circuit designs.

The ICAP/4Rx system strikes a perfect balance between ease-of-use and power. It has the power to handle tough designs, and removes the steep learning curve associated with using SPICE. And it does this without compromising power as some other workbench systems do, and at a reasonable cost (unlike OrCAD's™ Pspice®).

ICAP/4Rx is an integrated circuit simulation system including our 5th generation schematic entry tool, SPICE simulator, waveform post processor, and comprehensive model libraries. It is reduced in complexity in order to provide you with the easiest simulation environment to use.

Benefits To New Users

- Simple schematic interface makes it easy to get results instead of headaches
- Analysis Wizard alleviates the need to learn SPICE syntax
- Convergence Wizard helps you solve common simulation problems
- Integrated schematic-simulator solution makes you instantly productive
- Our mature 4th generation SPICE 3 program, IsSpice4, is time-tested and gives you accurate answers you can trust
- Graphical waveform analysis with multiple windows and cursor measurements gets the most out of SPICE
- Reduced complexity makes using ICAP/4Rx effortless
- Lowest cost solution gets you started along the right path

Elements and Models

- All IsSpice4 elements
- Digital gates including State Machines
- Comprehensive SPICE Model Library (5000+ parts)
- Advanced Behavioral Modeling: Math expressions, If-Then-Else, Table & Laplace models
- AHDL Models, BSIM3 version 3
- Advanced SPICE 3F.5 Convergence Algorithms
- Includes All vendor supplied IC models

General

- **No Hardware Protection Key**
- Compatible with the SPICE netlists produced by most schematic capture programs
- Low Cost Upgrade to ICAP/4Windows
- Schematics can be used with any version of ICAP/4
- Includes the power of Intusoft with all its services and support

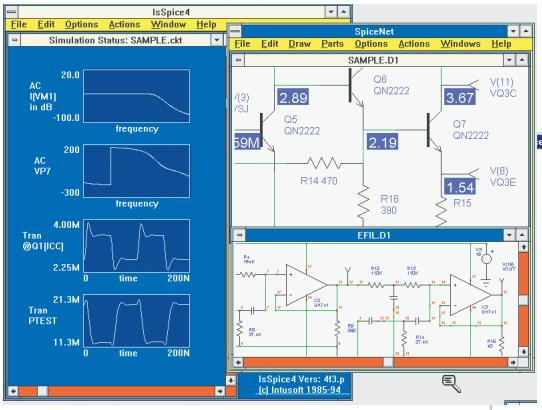
ICAP/4Rx For Sale On the Web

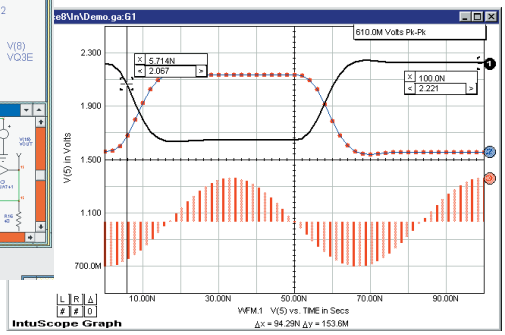
You can now purchase and download ICAP/4Rx from the Web. By going to **www.intusoft.com/OrderRx.htm** you can start simulating immediately without any waiting.

You'll get the latest version of ICAP/4Rx with unlimited circuit size, superior mixed-signal IsSpice4 engine and larger SPICE model libraries. With these new features and the ease with which you can get started, it's the perfect time to step up to professional simulation power. ICAP/4Rx is both easy-to-use and powerful, and now it's available instantly.

Simulation Environment

- Unlimited circuit size allows you to simulate large designs
- Proven state-of-the-art SPICE 3 and XSPICE-based simulator
- Perform all major SPICE analyses: AC, DC, Operating Point, Transient, Fourier, Temperature and Parametric "What-If " analyses
- Full Mixed-Signal simulation
- Interactive analyses and parameter sweeping with ICL Scripts
- Real time waveform display of ALL voltages, currents, & power dissipations
- Waveform and operating point voltage displayed directly on the schematic





What Makes Intusoft Different ?

What do you really get for your money? You're right to wonder about lower cost simulation alternatives. They may seem like they're complete. But they're NOT. Intusoft gives you EVERYTHING you NEED at a price you can AFFORD. Here's a brief look at what the other guys are missing.

Feature	Intusoft IsSpice4	OrCAD Pspice	Electronic WorkBench
Berkeley SPICE2, SPICE3 and XSPICE compatibility	Yes	No	SPICE3 Only
True interactive interface w/autosweeping & retrace	Yes	No	Limited
Design Validation (auto-simulate/verify multiple tests)	Yes	No	No
Failure, RSS, EVA, & Worst Case analyses	Yes	No	No
Analysis & Convergence Wizards	Yes	No	No
AutoSweeping/Curve family Wizard	Yes	Yes	Limited
Behavioral Modeling (If-Then-Else, Laplace, etc.)	Yes	Yes	Very Limited
Schematic - simulator integration	Very Tight	Very Poor	Tight
Edit ALL SPICE attributes from the schematic	Yes	Very Limited	Limited
Interactive Script Language (ICL) for SPICE	Yes	No	No
AHDL models and model development kit	Yes	No	No
Integration with OrCAD, Protel, and Viewlogic	Yes	OrCAD Only	No
ActiveX/Visual Basic scripting interfaces for SPICE	Yes	No	No
Display waveforms/OP data directly on the schematic	Yes	No	No
Full-featured waveform/data processing tool	Yes	Yes	No
Number of Model Types *, Total Models	**446**, 14,000+	**130**, 10,000+	**79**, 9,000+
Special Power, RF, & Mechatronic model libraries	Yes	No	No
Data sheet modeling tool	Yes	Yes	No
Low cost entry solution with upgrade path	Yes	Yes	No
Magnetics and Filter design tools	Yes	No	No
Offers their own training classes and SPICE seminars	Yes	Yes	No
Number of SPICE reference books published	4	1 **	0
Tutorial Multimedia Movies to get you going quickly	Yes	No	No
Shipping SPICE3 compatible SPICE for how long	Over 5 Years	Still Not	2 Years

✔ ICAP/4 Productivity

✔ Superior SPICE

✔ Advanced Features

✔ Ease-of-Use

✔ AHDL/C Code

✔ OLE Integration

✔ Biggest Libraries

✔ Affordability

✔ Knowledge & Experience

Table contents as of 5/1/99. * Includes different types of models MADE BY the SPICE vendor. It does not include models supplied to the SPICE vendors by hardware/op-amp vendors (such as Analog Devices, TI, elantec, etc.)
**Author does not work for Microsim/OrCAD™ anymore.

Models, Models, Models

Who REALLY Has The MOST Models?

There are major differences in the model libraries supplied by various EDA software vendors. These differences can dramatically affect the results of your simulation. Models are one of the most important parts of a simulation package. We encourage you to compare our libraries with other vendors' libraries and ask questions. Samples are available.

Intusoft prides itself on having the most comprehensive SPICE model libraries in the industry. While many companies offer model libraries with large numbers of parts, the user will invariably find that

Superior Products Support & Service

more than half of the models are for diodes and BJTs. Thousands of models are useless unless the **type of device** you need is available. Intusoft realizes that engineers design circuits with more than diodes and transistors. In our library you'll find a wide variety of part types, in addition to industry standard devices. Intusoft models the full spectrum of an engineer's toolbox and has the MOST MODEL TYPES of any SPICE program in the entire EDA industry.

At Intusoft, quality is paramount. Intusoft's models for complex parts, which our competitors model with simple SPICE primitive elements, utilize a subcircuit macro approach or sophisticated AHDL. In short, many of our competitor's models produce poor and inaccurate results!

Intusoft also realizes that your modeling needs don't end with the purchase of our software. In fact, they just begin. That's why Intusoft has a **FREE Modeling Service**. If you own our software and need a model, Intusoft will model the device for you. At No Charge. This is an invaluable resource - one you can't do without. This service is NOT offered by any of our competitors!

Just Some of the Models Our Competitors Don't Offer

Shown below is a list of some of the models that you WON'T find in the model libraries of our competitors. However, you will find them in ICAP/4, already made and ready to use.

System Blocks: Slew rate, Hysteresis, Z-transform, Sampled-Data, Calculus
Sources: Batteries, AM, PSK, FSK, 3 Phase, Sound (.Wav files), Random noise, NTSC, Stair-case, Dead-time, File data, Current-limited power supply
Power Electronics: PWMs (cycle-by-cycle models), Regulators, Switches with hysteresis, Power Factor IC, Transformers, Nonlinear Magnetics
RF Parts: MMICs, RF Beads, Circulator, BJTS, GaAs Mesfets
Mechanical: Motors, Hydraulics, Relays
Diodes: Photo, Laser, Bridges, PIN, Soft Recovery, SBS, Tunnel, Gunn
Semiconductors: Darlington BJTs, IGBTs w/flyback diode, Varistors, Sidacs, Dual-Gate Fets, UJTs
Other: Vacuum Tubes, Fuses, Spark Gaps, Thermistors, Fluorescent tubes, Connectors, Sensors
Model Templates
Generic Zener, Diode, Op-Amp, etc.
AHDL based models
BSIM3 version 3
State Machine, VCOs
SOI MOSFET model

How Do The Models From Different Vendors Compare??

Berkeley SPICE 3 has just over 20 different built-in models, such as diodes and resistors, which are called primitive elements. IsSpice4 also adds over 40 more built-in models which are made with our HDL (Hardware Description Language).

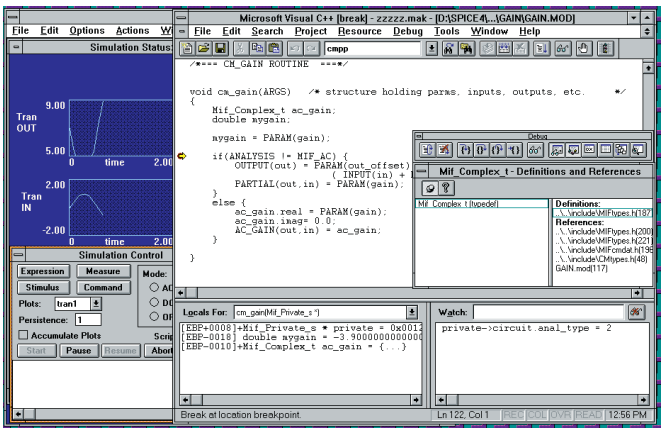
This fact means that the majority of electronic devices in the model libraries of SPICE vendors are built using subcircuits, or macromodels. These macromodels are made using various combinations of the primitive elements. For example, an op-amp usually has 15 - 75 primitive elements.

There are literally dozens of ways to emulate an electronic device that is not built directly into SPICE. Therefore, the same part from two different vendors may produce ENTIRELY DIFFERENT results. Some subcircuit topologies exhibit more accurate characteristics than others. Herein lies the difference between SPICE libraries from different vendors. NOT all models are created equal.

CMSDK: AHDL Model Development

Advanced AHDL Modeling Power

Intusoft offers a variety of modeling options including a sophisticated nonproprietary AHDL (Analog Hardware Description Language) based on C. The language is based on the publicly available XSPICE program. XDL models are like traditional SPICE models, except they are created using C subroutines. The code describing the model's behavior is linked to the simulator via an external file (Windows DLL) rather than being bound within the executable program. This allows new models to be added to the simulator, and old models changed externally, without having to recompile IsSpice4. You can create analog, digital, and mixed mode elements (i.e., mechanical, radiation, array processing, and system-level transfer functions, inter-program interfaces, etc.). New models are added to IsSpice4 using the Intusoft Code Modeling Software Development Kit. The kit is required only for model development, however, any IsSpice4 program can use the newly developed models. The process is easy to learn because you don't need to learn a proprietary language or custom development tools.



Write and Debug XDL Models Using Microsoft Visual C++®

Screen shots reprinted with permission from Microsoft Corporation

Intusoft and IsSpice4 FOCUSED on Simulation

Product Matrix *Features Included*	ICAP/4 Windows	ICAP/4 Windows RF Deluxe Option	ICAP/4 Windows Power Deluxe Option	ICAP/4 Rx	Power Supply Designer	ICAP/4 Professional	Test Designer
SpiceNet schematic Entry, IntuScope Waveform analysis tool, pgs. 2-5	✔	✔	✔	✔	✔	✔	✔
IsSpice4 simulator, pgs. 6-7	✔	✔	✔	✔	✔	✔	✔
Design Validation, pgs. 8-9	✔	✔	✔	✔	✔	✔	✔
Power Supply Designer's Library, pg. 10			✔		✔	✔	✔
Magnetics Designer, pg. 10					✔		
RF Device Library, pg. 11		✔				✔	✔
SpiceMod Modeling Tool, pg. 11		✔	✔		✔	✔	✔
Single Point Failure analysis, RSS, EVA, & Worst Case analyses, pg. 12	✔	✔	✔		✔	✔ plus automated failure analysis	✔ plus automated failure analysis
Low Cost Entry Solution, pg. 13				✔			
CMSDK AHDL model development kit, pg.15							
Model Libraries, pgs. 14-15	10,000+	13,000+	13,000+	5,000+	13,000+	13,000+	14,000+

Table contents as of 5/1/99. Magnetics Designer is sold separately or as part of the Power Supply Designer product. SpiceMod is sold separately or as part of the ICAP/4Windows Deluxe option. CMSDK sold separately. Power Supply Designer includes ICAP/4Windows Power Deluxe and Magnetics Designer.

Need More Details On Intusoft's Products?

www.intusoft.com

Web Site
SPICE CD-Rom
Call, Email, or Fax

(310) 833-0710 - Tele
(310) 833-9658 - Fax
info@intusoft.com - Internet

the absolute leader in affordable simulation tools

Hardware/Software Requirements

ICAP/4 Version 8.x - All versions

- x86, Pentium, P6, or Digital Alpha
- Windows 9x or Windows NT 3.5x, 4.x, or 5.x
- 16 megabytes RAM

Power Supply Designer™

- Same requirements as ICAP/4 above

Test Designer™

- Same requirements as ICAP/4 above

Magnetics Designer™

- Same requirements as ICAP/4 above

SpiceMod™

- DOS 5.0 or greater. Runs under the DOS prompt in Windows 9x and NT.

AHDL Model Development - CMSDK

- Requires Microsoft Visual C++™ 4.x - 5.x for model development only

Support

Other Vendors Talk About Support... Intusoft Delivers!

- **FREE SPICE Models -** The Intusoft technical support staff makes SPICE models FREE of charge for registered customers. Please inquire directly with Intusoft
- **Maintenance is required on all purchases**
- ***Intusoft Newsletter* -** FREE publication with SPICE application notes and models.
- **Tutorial Classes and Training -** Intusoft offers full on-site training by the engineers who developed the software
- **SPICE Reference Books -** "SPICE APPLICATIONS HANDBOOK", "A SPICE COOKBOOK", and "SMPS Simulation With SPICE3"
- **Web/Internet -** Intusoft maintains a home page on the World Wide Web along with the SpiceFarm™ simulation service
- **30 Day Money Back Guarantee**

intusoft

P.O. BOX 710
San Pedro, CA
90733-0710