



# ICAP/4Consumer

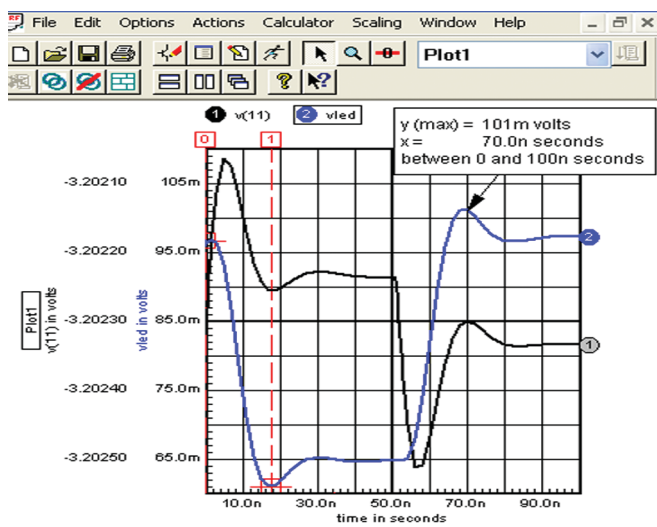
Use your computer to build electronic designs and display their operation!

Looking to build electronic circuitry and watch its operation right on your computer? ICAP/4 Consumer is an easy to use program that does exactly that. ICAP/4 Consumer enables you to quickly construct any type of analog, digital or mixed analog-digital design, apply electrical stimulus, and view signal waveforms throughout the design.

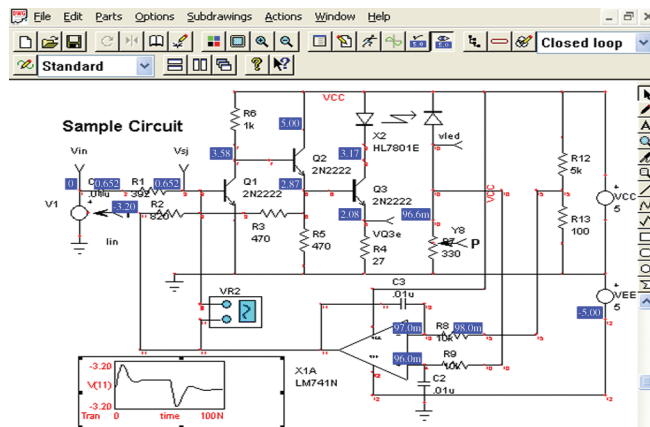
ICAP/4 Consumer features "SpiceNet," a powerful design entry system that enables you to place almost any type of electronic component on your computer screen. Then, you simply wire the connections between components, apply electrical stimulus (i.e., battery, AC outlet voltage, digital signal), specify the type of design simulation you want, and push a button to simulate the circuit's operation. Simply point-and-click your mouse on any wire or component in the design and signal waveforms instantly appear in the versatile waveform viewing tool (IntuScope).

Part values can be tweaked, wires can quickly be rerouted, parts can be changed or added, then a new circuit simulation enables signal waveforms to be viewed again.

ICAP/4 Consumer is also a great educational tool. It teaches anyone interested in electronic design how to use today's leading design-simulation technology, as used in colleges and industry. ICAP/4 Consumer is menu (and icon) driven, giving it a familiar look and feel. It's a robust program based on ICAP/4's powerful IsSpice4 kernel and today's leading circuit-simulation technology created by Intusoft.



IntuScope provides ultimate waveform viewing, scaling, math operations and 100+ waveform processing functions.



Probe any wire or device for waveforms on the schematic or in the IntuScope waveform viewer. DC levels in blue.

**ICAP/4 Consumer** provides an integrated and easy-to-use design environment that enables you to:

- Draw a complete schematic from a vast collection of analog, digital and systems-level components
- Simulate a wide variety of circuits, useful for hobby, school, home, car, shop and work.
- Perform fast simulation of circuit behavior and instantly view signal waveforms. Quickly modify your design and view waveforms again in three different modes.
- Great for things like home and automotive electronics, hobby circuits, power supply, electromagnetic design, power converter, digital counter, amplifier, noise generator, digital signal control, signal processor and more.
- Sweep a component through a range of values, or even a semiconductor's temperature or design temperature. Quickly view signal waveform variation for each change.
- Perform an extensive set of waveform calculator-like functions and view instant results (i.e., add 2 waveforms, subtract, multiply, divide, integrate, log, average value, frequency spectral analysis, filtering and more).

ICAP/4 Consumer provides a vast collection of 3,000+ components such as resistor, capacitor, motor, lamp, fuse, thermistor, pressure sensor, photo/laser diode, switch, digital devices, fuse, transistor, battery, lamp, LED, transformer, rectifier, connector, crystal, DC converter, coax line, spring, multiplier, divider, operational amplifier, variable resistor, relay and more. Design size is limited to approximately 100 parts.

IntuScope enables signal plots to be customized with many different line styles, colors and text labels. Multiple cursors can be added and any two displayed at once. They also provide a wealth of numerical measurements on waveforms.

# Makes Circuit Design Easy!

## Easy, Affordable and Upgradeable

- Quickly get up and running with an extensive in-application help system, movies and softbound manuals.
- Go through provided design examples to learn circuit design.
- Receive free software upgrades from Intusoft's website.

## Simulation Environment

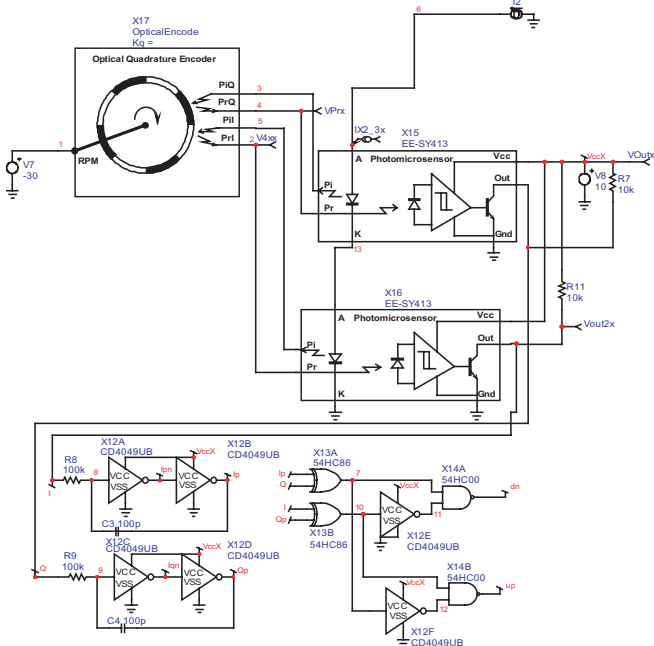
- IsSpice4: SPICE 3F5 and XSPICE based simulation kernel provides big gains in simulation accuracy and speed.
- AC, transient, operating point, DC sweep, Fourier, temperature and pole-zero analysis provided.
- Sweep a passive component value, semiconductor's temperature or global circuit temperature to create a curve family.
- Observe simulation progress with real-time marching waveforms.

## Elements and Models

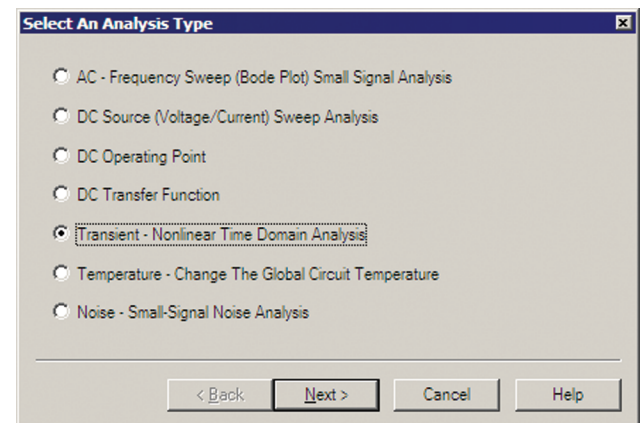
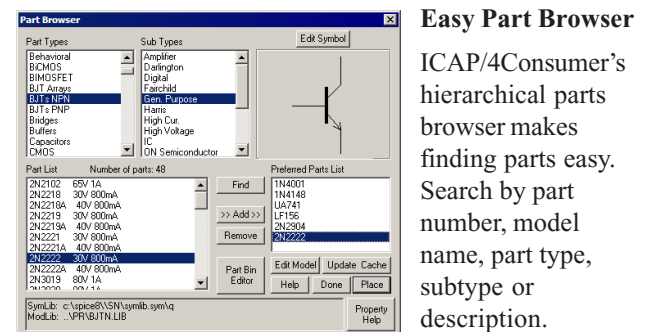
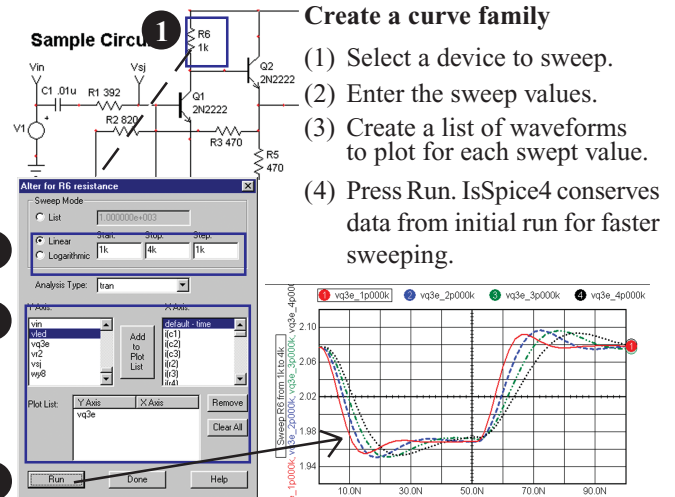
- 3,000+ models, including 600+ digital parts and 200 model types. Nonlinear magnetic, SCR, IGBT, PWM (average), MOV, lamp, fuse, thermistor, pressure sensor, photo/laser diode, 1,500+ vendor IC models and configurable generic models.
- Behavioral Modeling: math expression, Boolean, if-then-else & table.
- Resistors, capacitors and inductors accept expressions. Use trigonometric functions; passed parameters; and simulation time, frequency or circuit temperature in an expression.
- Accommodates SPICE and Pspice elements.

## Graphics and Post Processing

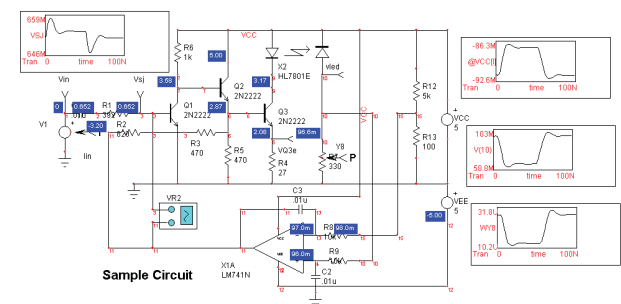
- Cross-probing displays waveforms instantly on schematic and in the IntuScope waveform viewer.
- Operating points, waveforms and custom artwork displayed directly on the schematic. Copy/paste schematic or waveform graph into any publication for documentation purposes.
- Instantly update previous waveform data by overwriting old waveforms, appending to them, or create a new graph.



Many example circuits and lessons are provided



Use wizards to setup your analysis.



Cross-probe any node voltage, or device current and power to display the waveform on the schematic, or in the IntuScope waveform viewer.