

Why Power Supply Designer is for You

To help meet with the complexity of today's power electronics, Intusoft features its Power Supply Designer product. This is a complete software solution that enables fast power supply design, simulation, verification and production yield analysis. These critical processes speed the design cycle and heighten product reliability.

Power Supply Designer integrates Intusoft's "SpiceNet" schematic entry and design management system, with the proven "IsSpice4" analog and mixed-signal simulation kernel. The package also features: "IntuScope," ICAP/4's powerful waveform viewing and signal processing tool; "Magnetics Designer" transformer and inductor design and synthesis tool; a wealth of advanced design verification analyses; over 22,700 component models; and model development and library management tools.

The IsSpice4 simulator is widely recognized for its handling of complex design, such as power supplies with tight feedback, switching circuitry, and nonlinear magnetic devices. The 4th-generation SPICE kernel embodies a tremendous amount of algorithmic enhancement for speed, convergence and simulation accuracy. The product's vast provision of simulation models includes a dedicated library of 1,100+ power devices, plus unique models that employ state-of-the-art technology for SMPS design. In all, this dynamic solution will more than meet your demand for power design, or any other type of analog, mixed-signal or mixed-systems circuitry.

Power Supply Templates

Power Supply Designer provides 30 SMPS schematic templates to make the design process quick and effective. Forward, Flyback and Push-Pull comprise typical design topologies.

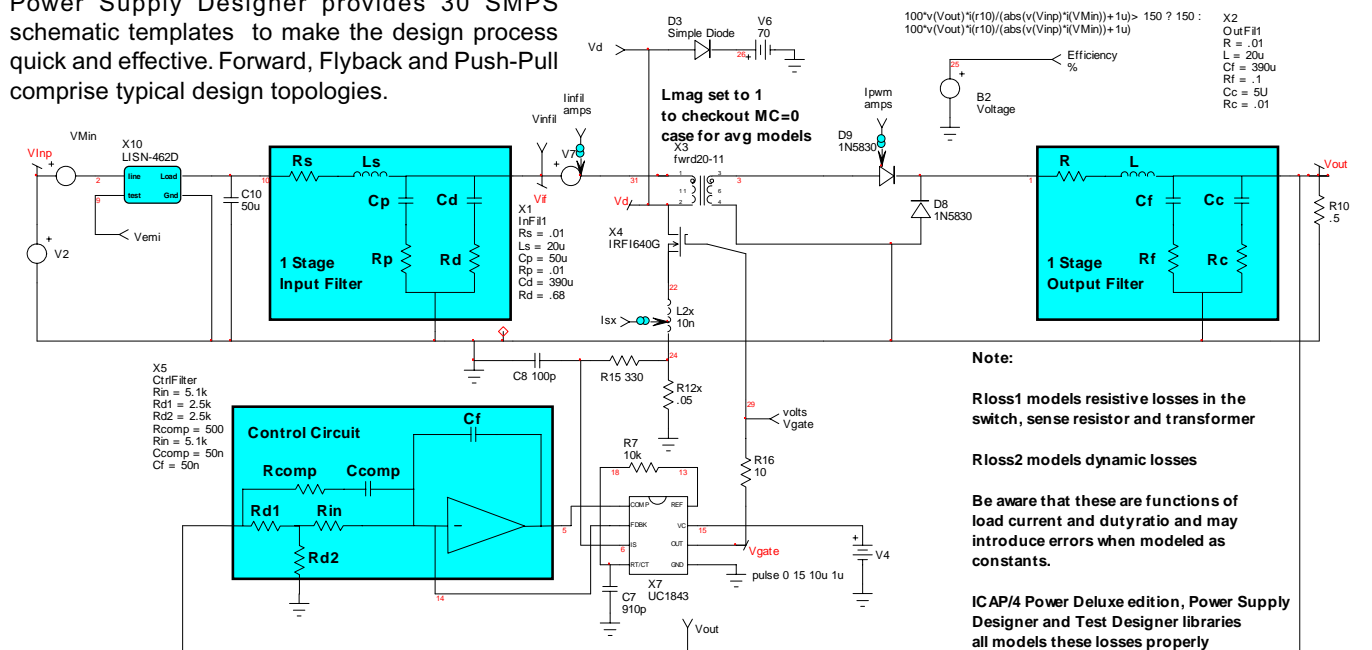
Top Down Design

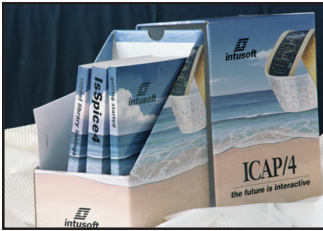
Designing a power supply from the top down begins with a high-level design framework, followed by more detailed circuit complexity. SpiceNet draws from a variety of sophisticated behavioral models that provide different levels of accuracy and simulation speed at the top level. More detailed models can be successively substituted into new schematic layers and configurations. This layering capability is flexible and creates unique overlays of reusable circuitry, then combines desired layers to form different schematic configurations (i.e., for debug, detailed design, test, PCB layout, documentation, etc.)

DSP Designer

This revolutionary product enables digital control and other aspects of DSP design to be visually simulated as transfer functions and equivalent assembly code instructions using special device models. Corresponding waveforms include Bode plots of hidden parameters within Z-transformations. The product is also valuable for verifying DSP assembly language traditionally programmed through a development board interface, though even this process will be automated with DSP Designer.

Special model abstraction for power supply design features ICAP/4's ultra-fast hysteretic state-average models, switch level models, custom non-linear magnetic devices and passive devices containing 2nd-order effects. Valuable GFT injection models are also used, which provide a balance of nulling and injection between voltage and current. These revolutionary models measure open-loop control system characteristics while preserving the loop closed, thus maintaining correct loading and DC bias.





ICAP/4Windows Power Deluxe features a 4th-generation SPICE simulation kernel, plus complete design and verification software.

Magnetics Designer can design virtually any single phase, layer wound, inductor or transformer from 10 Hz to 5 MHz.

Production Readiness

Component tolerances can be varied in a number of ways to mimic design performance in production. Any real component is subject to change through its tolerance. Therefore, Power Supply Designer's pre-production verification employs analyses such as Monte Carlo, worst case, extreme value, fault injection and component stress to simulate actual parametric and temperature variances that occur in real circuitry. Monte Carlo and extreme value analyses respectively vary component tolerances in a statistical and worst-case fashion. Other analyses such as DC, AC and transient sensitivity, plus noise, distortion, pole-zero and design optimization, also serve to maximize design reliability prior to the expense of a prototype board or IC. Parametric variation on parts while viewing real-time waveforms is powerful and fast, plus requires no schematic intervention. Expressions, or variation as a function of another part's variation, can quickly be assigned to components. Fast single or nested part and temperature sweeps with curve family also speed the debug cycle.

Prior to production, the IntuScope waveform viewing tool minimizes debug time by enabling fast signal plots. You can click any node or part on the schematic, or from a list to view waveforms. Toggle between run types, plot any variable against another (e.g., power vs. temp.), and select (or modify) 150 waveform processing functions. Previously viewed waveforms can be updated in 3 selectable modes. And if your power design contains digital busses, IntuScope's one-of-a-kind Digital Data Viewer charts buss data in binary, octal, hexadecimal and mnemonic format, in synchronization with system clock waveform edges.

Accurate Models Play a Key Role in Design Reliability

Accurate component models address circuit behavior such as switching, snubbing of loadline, power stage loss, inrush current and EMI filter performance. Difficult to simulate using conventional SPICE tools, Power Supply Designer uses sophisticated capability with its IsSpice4 kernel to simulate in both time and frequency these important design properties that are characteristic to power and switchmode design.

A gamut of second-order effects found in real magnetic devices are accommodated with Power Supply Designer's "Magnetics Designer" software. Magnetics Designer is the industry's de-facto standard for synthesizing non-linear magnetic models. Apart from its ease of use and highly graphical approach, key parameters like high-frequency losses caused by gap fields that intercept the wire, layer-to-layer proximity losses, leakage inductance, AC resistance (including proximity and skin effects), interwinding capacitance and several other properties are accurately modeled. Thousands of core models are also furnished, including saturable elements when non-linear magnetic properties are required.

The IsSpice4 kernel adds an extensive Hardware Descriptive Language to its model library, as represented by some of the complex power components, logic devices and mixed-engineering system blocks it contains. Further, a rich set of switching regulators, transistors, FETS, diodes, PWMs, ICs, magnetic devices and other power components are supplied. "SpiceMod" enables fast model development for 13 types of discrete devices, based on manufacturers' datasheet entry, while Library Manager assists in verifying a new model's performance with SpiceMod's host of test circuits. Library Manager also removes the burden of parts management by providing a number of helpful features that orchestrate the organization and archival of models.

Simulate Your Entire Power Supply System

- Complete switching power supply design solution
- Custom magnetic design and synthesis capability
- Automatically assign and monitor a wide range of electrical measurements and their limits. Results displayed as colored out-of-limit histograms
- Transient and AC analyses on switchmode control design, plus 9 verification analyses
- **DSP Designer** enables a visual simulation of Z transforms, assembly instructions and other DSP functions.
- Full-featured graphical design entry, extensive simulation and debug tool suite, and powerful waveform analyzer
- Simulates large-signal performance on high-gain feedback and switching circuitry using powerful modeling techniques
- Runs CCM and DCM converter simulations
- Analyzes control systems, including loop gain, input filter analysis, power stage loss, inrush limiting, snubbing of loadline and EMI waveform conversion
- Features pole-zero analysis for insight into circuit stability, plus optimization for auto calculation of parts based on a design objective
- Features powerful convergence wizard for tough design

What is Included in Power Supply Designer?

ICAP/4Windows Power Deluxe SpiceNet design entry, IsSpice4 simulator, IntuScope waveform analyzer, 9 verification analyses, SpiceMod discrete part modeler & Library Manager.

Magnetics Designer Transformer and inductor design and synthesis. Outputs standard Spice model and winding specification.

Model Libraries More than 22,700, including dedicated Power Library. Additional parts easily created with SpiceMod model building tool. Includes over 30 power supply design templates, several test schematics and Library Manager.

Why You Need Power Supply Designer

Saves You Money

Find hard and soft problems before production.

Saves You Time

Simulate before prototyping. Use power supply templates instead of designing from scratch.

Measures the Immeasurable

Vary component parameters on-the-fly and perform full manufacturability analysis. Perform stability analysis on sensitive circuitry.

Improves Safety

Explore and eliminate failures that produce hazardous conditions.