Power IC Model Library

PSpice Models for the Power Electronics Designer

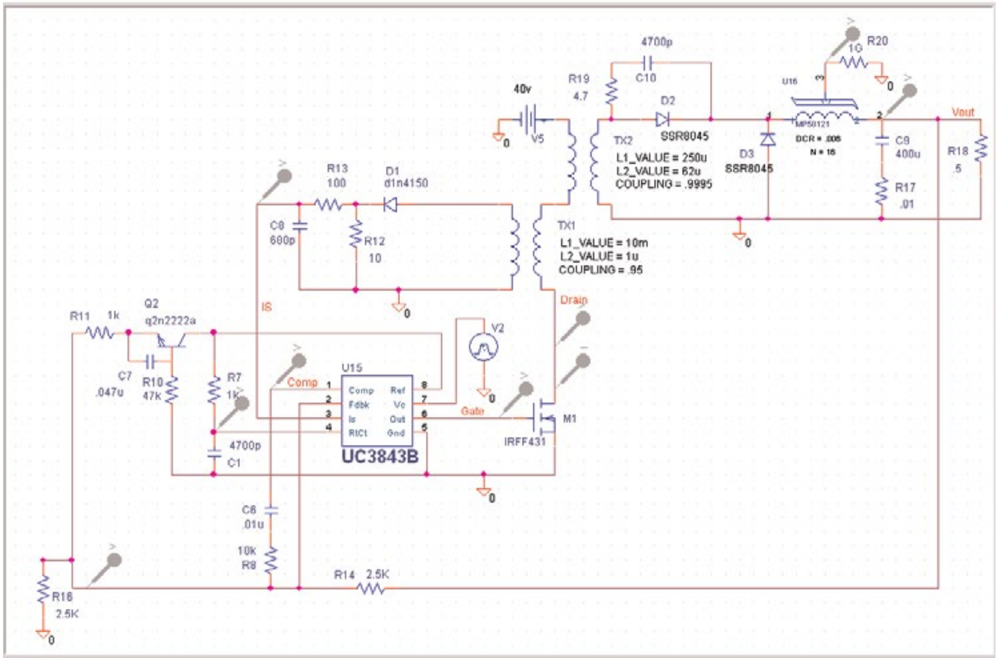
Power IC Model Library Key Features

- 1,700 pre-configured, ready-to-use PSpice models
- Bench verified accuracy for real-world simulation
- Analyze large signal effects such as 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 limitations
- Useful application circuit example schematics in multiple file formats to accelerate your designs
- Symbol library containing both OrCAD Capture and Microsim models (OLB, SLB files)

EMA Design Automation®

AEi Systems

The Power IC Model Library incorporates over 1,700 high fidelity time-domain PSpice models for power electronic designs. It gives designers capabilities previously unavailable for many popular parts—the ability to plug in a model, representative of the actual IC, and simulate the switching performance under actual operating conditions.



A 50W forward converter circuit using UC3843B PWM and Magnetics MPP58121 core models. The simulation explores the circuit operation as the transformer coupling is changed. The simulation can generate start up or steady state output depending on the settings

Simulate Switching Performance Under Actual Operating Conditions

The Power IC Model Library, a product of AEi Systems, is specially designed for the Cadence® PSpice® analog and mixed signal simulator. Nonlinear characteristics such as propagation delay, switching speed, drive capability and maximum duty cycle/current limits, startup phenomena 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, hysteric current control, low voltage, and low operating current.

Model accuracy and fidelity is assured by AEi System's proprietary relationships with the majority of the top analog IC manufacturers. The models in the Power IC Model Library are verified with bench data under start-up, steady state, line, and load transient conditions. This ensures the models exhibit the kind of accuracy you expect from PSpice.

Power IC Model Library
Improved Simulation Speed and Accuracy

Power IC Model Library Contents

Over 1,700 Power IC Models including:

- o Phase shift
- o Voltage and current mode PWM controllers
- o Switching regulators
- o Motor controllers
- o Power factor correction
- o Power MOSFET drivers
- o Automotive Stimulus Models for ISO 76750/ISO 7637-2 and FMC1278 including :
  - C1220 (load dump)
  - C1230A (power cycling)
  - C1250 (voltage offset)
  - C1260A-E (Voltage dropout)
- o Popular parts including:
  - UCC3895 and UCC380x
  - HS117/LM117
  - UC384x, UC152x, and UC182x
  - LT124x
  - UC1846 and UC1854
  - IR2110
- o Linear ICs:
  - AD813x Differential Amps
  - AD8333 Phase Shifter
  - AD8331 VGA
  - ADx36/x37 DC-RMS Converter
- o Nonlinear Magnetic Cores
- o Transformers
- o Opto-Couplers
- o Models from leading manufacturers including:
  - Texas Instruments
  - Intersil
  - ON Semiconductor
  - Linear Technology
  - International Rectifier
  - Micrel
  - Vishay
- o Spark Gap, Fluorescent Tube and Dead Time Controllable FET Driver Models

System Requirements:

OrCAD Capture/PSpice:
Version 15.9 or greater

Microsim Schematics/PSpice:
Version 8.0

AEi Systems® has developed proprietary relationships with the majority of the top analog IC manufacturers and is the sole developer of power IC models in many cases. These relationships ensure that the models exhibit the accuracy you expect from a PSpice simulation.

Simulate Your Entire Power System

The library includes example application circuits for most of the IC models. Symbols, model libraries and schematic sets are available for both Cadence OrCAD® Capture and Microsim Schematics. You can perform high-speed cycle-by-cycle simulation to show true large-signal performance, simulate current-mode control using the latest accurate modeling techniques, run CCM and DCM converter simulations, analyze control systems including loop gain, input filter design and analysis, and measure power stage loss and stress analysis for all major components. In summary, you can simulate your entire power system.

The PSpice Simulator Provides Speed Increases for Complex Power Supply Simulations

Some newly added models in the Power IC Model Library for PSpice incorporate recent PSpice built-in component and building block advances that allow ements. Together they greatly reduce the model's simulation runtime, an important factor in SMPS analysis. Most simulations run in just a few minutes. These improvements result in speed increases of 10 to 40% for power supply simulations.

Key Benefits of the Power IC Model Library

- Analyze large signal effects like start-up transients, power stage semiconductor stress, and step-load response
- 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.

About AEi Systems

The Power IC Model Library is a product of AEi Systems, a leader in the technology of SPICE and PSpice modeling, and Worst Case Circuit Analysis.

AEi Systems (www.aeng.com) specializes in the design and analysis of power conversion circuits for satellite systems, and also performs analysis for commercial and military, ground based and airborne, analog, digital and mixed mode applications.

For More Information

EMA Design Automation provides exclusive distribution and support throughout North America for the AEi Systems Power IC Model Library. For sales and pricing information contact EMA Design Automation, a Cadence Channel Partner:

EMA Design Automation, Inc.
225 Tech Park Drive
Rochester, New York 14623

Phone: 877.362.3321
eMail: info@ema-eda.com
Web: www.ema-eda.com