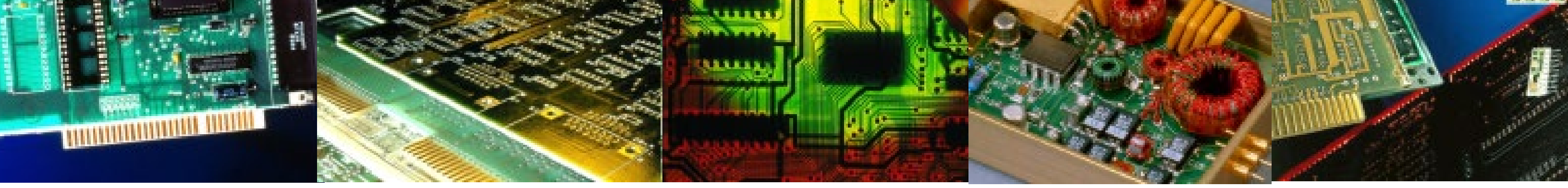


Power IC Model Library™
for PSpice® version 4.4

AEi Systems presents The Power IC Model Library for PSpice. The library now incorporates over 1,200 high fidelity and correlated PSpice models for power electronic design and simulation. The Power IC Model Library for PSpice includes:

- PSpice Model Libraries (.lib files)
- Useful application circuit example schematics (opj)
- Full Symbol sets for both Microsim Schematics and OrCAD Capture (olb, slb files)
- OrCAD Capture and Microsim Schematics Support

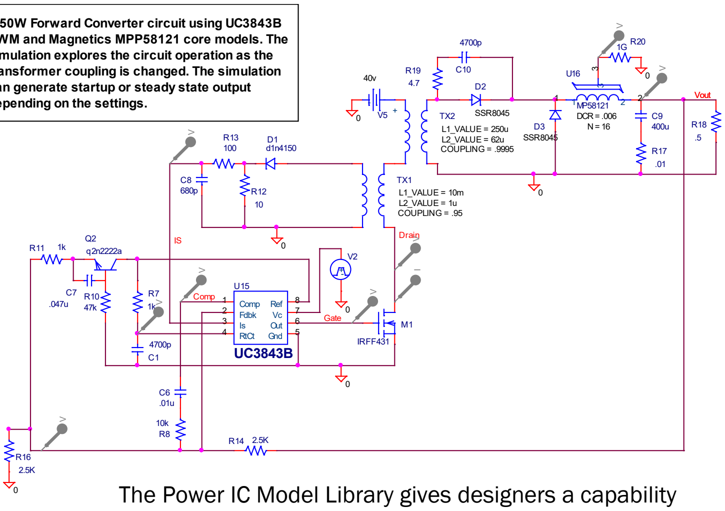
The Power IC Model Library for PSpice, created by AEi Systems, includes model netlists in PSpice syntax, schematic symbols for both MicroSim Schematics and OrCAD Capture, and an expanding set of example application circuits for many of the IC models. New examples include switch boost converters for LCD and white LED applications, high-voltage hot swap controllers, and current-mode notebook power controllers.

The product is compatible with PSpice version 17.x, 16.x, 15.9, and Microsim Schematics, as well as the latest OrCAD Capture version 17.x/16.x.

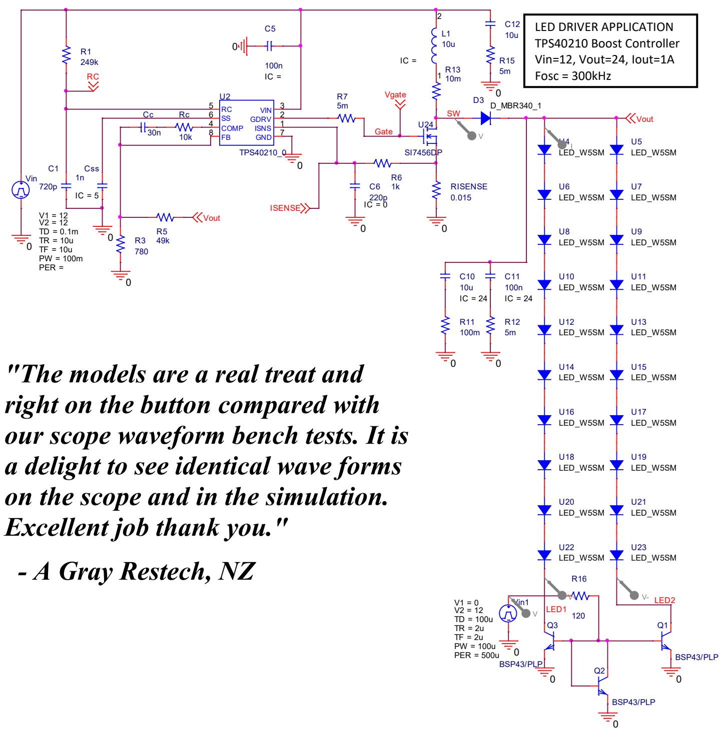
The Power IC Model Library for PSpice includes models for:

- Phase shift, Voltage and current mode PWM controllers, Switching regulators, Motor controllers, Power factor correction, and Power MOSFET drivers
- Popular parts: RH and ISL Opamps and references, UCC3895, xx117, UC384x, UC152x, UCC380x, LT124x, UC182x, UC1846, TL431, IR2110, UC1854
- Linear ICs: AD813x Differential Amps, AD8333 Phase Shifter, AD8331 VGA, ADx36/x37 DC-RMS Converter
- Nonlinear Magnetic Cores, Transformers, Opto-Couplers, MOVs
- TI, Intersil, ON Semiconductor, Linear Technology, International Rectifier, Micrel, Vishay and more
- Automotive EMC Transient Signal Generators (FMC1278/ISO 7637-2/ISO 76750-2)
- Spark Gap, Fluorescent Tube and Dead Time Controllable FET Driver Models
- Ford EMC transient sources (CI 210, 220, 221, 222, 230, 231, 250, 260 and 280)

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 startup or steady state output depending on the settings.

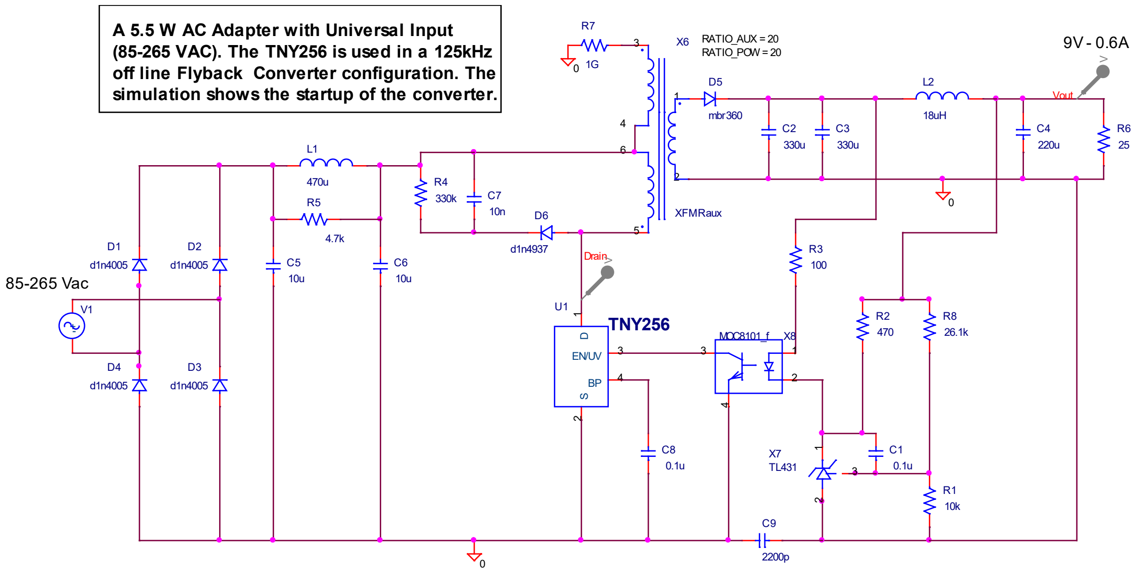


The Power IC Model Library gives designers a capability they have not had before for many popular parts: the ability to plug in a model representative of the actual IC and simulate the switching performance under actual operating conditions.



"The models are a real treat and right on the button compared with our scope waveform bench tests. It is a delight to see identical wave forms on the scope and in the simulation. Excellent job thank you."

- A Gray Restech, NZ



"AEi Models provide exceptional results. Their models correlate consistently with our hardware!" - L.J., On Semiconductor

All models are created by AEi Systems, the world leader in SPICE model development. The Power IC Model library represents a tremendous value, as single models ordinarily cost between $2,000 and $15,000 each to develop.

The Power IC Model Library, coupled with OrCAD PSpice, makes a very compelling offering for engineers looking to simulate their power designs and gives users a feature set that no other analog tool vendor can match.

SMPS applications today are much more demanding than ever. Today's designs require increases in power IC functionality, switching frequency and system interaction. State space average based models simply do not reveal many important nonlinear factors that influence these performance characteristics.

The Power IC Library enables you to 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, generate line and load step responses, and measure power stage loss and stress for all major components.

Model accuracy and fidelity is a paramount concern. AEi Systems has proprietary relationships with the majority of the top analog IC manufacturers and is the sole source of Power IC models in many cases. These relationships ensure that the models exhibit the kind of accuracy you expect from PSpice.

The models in the Power IC Model Library are verified with bench data under start-up, steady state, line, and load transient conditions.

Nonlinear characteristics such as propagation delay, switching speed, drive capability, maximum duty cycle/current limits, and start-up 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, hysteretic current control, low voltage, and low operating current, to name a few.

NEW FOR VERSION 4.4

- Diodes, FETs, Capacitors
- References and Linear Regulators (RH117/RH137, RH1009/RH1021, ISL75051, LM4050)
- Opamps and Comparators (RH1014, RH1498)
- SIC Diodes
- Ferrite Beads
- LISNs

For More Information

More details are available at www.AENG.com/PSpice.asp including a list of the components and documentation detailing the models' accuracy and performance.

Requirements

- OrCAD Capture/PSpice version 15.9 or greater
- Microsim Schematics/PSpice version 8.x

Call us today to order your copy!

To order the Power IC Model Library for PSpice, call your local salesperson, or contact:

ae systems
ANALYTICAL **HEAVY** LIFTING

AEi Systems, LLC
Phone: 310.216.1144
Email: info@AENG.com