

AEi Systems
Power IC Model Library™
for PSpice®

Model Documentation

Version 3.0a, February, 2010
© 2005-2010 AEi Systems, LLC.
All Rights Reserved.

Trademarks

The AEi Systems logo and “Power IC Model Library” are trademarks of AEi Systems, LLC. OrCAD, OrCAD Capture, and PSpice, are registered trademarks of Cadence Design Systems, Inc.

All other brand and product names mentioned herein are used for identification purposes only and are registered trademarks, trademarks, or service marks of their respective holders.

Copyright notice

Except as permitted under the United States Copyright Act of 1976, no part of this publication may be reproduced or distributed in any form or by any means, or stored in a data base or retrieval system, without the prior written permission of AEi Systems, LLC.

As described in the license agreement, you are permitted to run one copy of the AEi software on one computer at a time. Unauthorized duplication of the software or documentation is prohibited by law. Corporate Program Licensing and multiple copy discounts are available.

Contact information

EMA technical support	(585) 334-6001 option 5	EMA technical support e-mail	techsupport@ema-eda.com
AEi Systems support	(310) 216-1144	Technical support e-mail	info@aeng.com http://www.aeng.com

Table of Contents

Chapter 1 - Overview	4
Welcome	4
What's Included – Getting Started.....	6
Installing the Power IC Model Library	6
Installing the Power Library for Older PSpice versions.....	7
Model Documentation and Support.....	9
Chapter 2 - Model Discussions	10
Model Usage.....	10
Using the Models with Other Simulators.....	11
Chapter 3 - Using the Power IC Model Schematic Examples	13
Schematic Examples.....	13
Types of Simulations	14
Simulation Convergence – Quick Fix.....	15
Simulation Convergence	16
General Discussion	17
DC Convergence Solutions	18
Transient Convergence Solutions.....	20
Modeling Tips.....	21
Chapter 4 - Library Listings	23
Power FET Drivers.....	23
Linear ICs.....	24
Power ICs	24
Semiconductors.....	29
Magnetics	30
Generic Model Templates	31
Chapter 5 - References	32
General.....	32

Chapter 1 - Overview

Welcome

Thank you for purchasing the AEi Systems Power IC Model Library for PSpice.

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, AEi Systems introduced the Power IC Model Library for PSpice. This library represents a major breakthrough for SMPS designers who use PSpice.

AEi Systems has spent years developing accurate and robust models for the components that are used in power designs. We test our models thoroughly so you can have confidence in the model's operation and results. Useful examples are also provided for most of the models.

The library incorporates a comprehensive set of large signal hyper-accurate cycle-by-cycle simulation models for Pulse Width Modulation (PWM), Switching Regulators, Phase Shift Controllers and other Power ICs. 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.

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, to name just a few.

Summary of Benefits:

- 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.

The models utilize analog behavioral elements, special Boolean logic elements and other specially designed function blocks. Together they greatly reduce the model's simulation runtime, while maintaining "better than data sheet max/min" accuracy, an important factor in SMPS analysis.

Components are normally modeled to match "Typical" part performance at room temperature. Some temperature performance variations are also taken into account.

What's Included – Getting Started

The Power IC Model Library includes over 300 PSpice syntax compatible models in multiple model library files (See Chapter 4, Library Listings). Example schematics in native format and symbols for both OrCAD Capture and MicroSim Schematics are included.

Files included and their location after installation:

PSpice Power Library Folder

- License files and documentation

PowerLib Folder

Library Folder - Models and Symbols

- PSpice Model Libraries Files for both Capture and Schematics (.LIB)
- Symbol Files for both Capture and Schematics (.OLB, .OLJ, .SLB)

Examples Folder – OrCAD Capture Schematics

Various Folders (by manufacturer)

- Capture schematic files (.OPJ, .DSN, .SCH)

MicroSim Folder - MicroSim Schematics Files

Various Folders (by manufacturer)

- Schematics Files (.SCH, .NET, .ALS, etc.)

Documentation Folder - Manual and Model Reference Documentation

Installing the Power IC Model Library

The Power IC Model Library installation utility will install the library and other files onto your computer regardless of the version of PSpice you are using.

For PSpice versions 10.0 or higher the utility will also automatically configure OrCAD to use the libraries. For older versions of PSpice please follow the instructions provided in the next section.

Please see the relevant portions of your OrCAD or MicroSim User's Guide for details on how to incorporate new models and/or symbols into the design environment.

Symbol and model libraries generally use the name of the IC manufacturer whose parts are modeled in the library. The models in a .LIB file have corresponding symbols in the same named symbol file. Please see Chapter 4, Library Listings, for information on where specific models and symbols are located.

Master Library File

The **Power_EMA_AEI.LIB** file contains a list of all of the library files and may be used to include all of the models in the Power IC Model Libraries in a manner similar to the NOM.LIB file (Nom.Lib is distributed with PSpice).

In some cases, common sub-blocks called by a subcircuit in one library might be located in another library. That's why it is best to use the Power_EMA_AEI.LIB master library file to incorporate models into your simulations and schematic environment.

Important Note: It is recommended that Power IC Model Library files and symbols NOT be placed in the same directory as the OrCAD delivered files in order to avoid any naming conflicts that may arise in the future.

MicroSim Schematics

While the MicroSim Schematics files are included in the relevant OrCAD schematics folders a separate folder set is included for MicroSim Schematics users that only includes the relevant *.SCH file, without any OrCAD Capture related files.

You may have to edit the Library Settings using the Editor Configuration function in the Options menu in order to get MicroSim Schematics to recognize the .SLB symbol files.

Uninstalling the Power Library Files

1. Use the Add/Remove Program feature in the Window Control Panel to uninstall the Power IC Model Library.
2. After the uninstalling is complete, go to the Simulation Settings window and select the Power IC Model Libraries from the list in the Configured Files section.
3. Delete the Power_EMA_AEI.lib item from this list, by clicking on the "X" symbol button just above the list.
4. This will ensure that Power IC Model Library and its components are completely removed from all your future PSpice projects.

Installing the Power Library for Older PSpice versions

Installing the Power Library Files for PSpice versions prior to version 10.0

1. Open a new or an existing PSpice project.
2. From the PSpice Menu, choose New Simulation Profile (provide a new name for the profile) or Edit Simulation Profile.
3. In the Simulation Settings window, chose the Configuration Files tab.

4. Choose Library from the Category list, seen on the left hand side.
5. Click on the Browse button in the Details section and browse to the location in your machine where the Power IC Model Library was installed during the installation process. By default the library and its components are saved within C:\Program Files\PSpice Power Library
6. Locate “Power_EMA_AEI.lib” which is located in \PSpice Power Library\PowerLib\Libraries and click Open.
7. Click on Add as Global and the models in the Power IC Model Libraries (which now will appear on the top of the list under Configured Files section) are now ready to use in your PSpice projects.
8. Click OK to exit the Simulation Settings window.

Model Documentation and Support

Each model in the AEI Systems Power IC library is tested under various conditions using multiple test circuits for specific functions and then again in one or more full application test circuits. The results are compared to data sheet performance and in most cases actual bench measurements.

AEI Systems strives to produce models that meet all key performance metrics and exhibit “typical” performance as specified in the data sheet or performance that is within given Max-Min specifications.

Adobe Acrobat .PDF files containing documentation for various models in the library are included in the Documentation folder on the distribution CD.

The documentation discusses the model development and architecture and contains the results of the testing and verification process.

The support section of AEI System’s web site, <http://www.aeng.com/support.asp>, contains additional model documentation and various white papers on simulation and modeling.

Documentation for other models that are not on the distribution CD may or may not be available. Please inquire with AEI Systems directly at info@aeng.com.

If you purchased this library from EMA Design Automation, you may get support in any of the following ways:

EMA Resource Center	http://support.ema-eda.com
e-mail	techsupport@ema-eda.com
Telephone	585-334-6001 option 5

Chapter 2 - Model Discussions

Model Usage

The majority of the models in the Power IC Model Libraries are transient time-domain models. That includes the FET Drivers and majority of the controller models (parts without an “s” extension in the model name).

The controller models are normally pin-for-pin compatible with the actual physical part. All key functions of the actual chip are modeled with the exception of variations with temperature. This includes startup nonlinearities and other transient phenomenon. Smoke alarm parameters are not currently implemented. While over-current and other protection functions are modeled, stimulus or supply voltages that exceed the data sheet minimums or maximums may produce unreliable results.

The models are characterized for typical operation at room temperature.

The transient models can be used in all types of simulations including startup, line transient, load transient, and steady state, provided that the external circuit and stimulus are properly adjusted.

The transformer and semiconductor models can be used in either transient or frequency domain simulations.

For switching circuit simulations linearized models are required in order to perform frequency domain simulations. The switching based transient controller models can not be used in frequency domain simulations. Models that are linearized fall into the classification of “state space” models. While a variety of state space models are available on the Internet,

the Power IC Model Library includes Boost, Buck, Forward, and Flyback “PWM” blocks. See references 1 and 2 for more information on these blocks.

Frequency domain versions of various PWM controllers are included in the Power IC Model Library. Many will have an “s” appended to their model name as in the LT1242s and UC1845As. Several example .AC simulations are included (UC1842STATESPACE.DSN, UC1843ASTEST.DSN, NCP1000AVGTEST.DSN, etc.). Please see the list of models, “Power_Library_List_3.0.pdf” for specific model names and functionality.

Using the Models with Other Simulators

In order to produce models that are accurate but run in a reasonable time frame, AEl Systems uses a combination of actual semiconductors and behavioral modeling constructs to model power ICs. Two constructs are utilized frequently; the switch with hysteresis and If-Then-Else expressions.

Not all versions of PSpice support a switch with hysteresis. While the S_ST Short-Transition switch model, which emulates the Berkeley SPICE 3 S element, is now available, older versions of PSpice (prior to v9.2.3) did not support the hysteresis effect. A subcircuit, shown below, is utilized in order to provide compatibility with all versions of PSpice without compromising performance.

```
.subckt SWhyste NodeMinus NodePlus Plus Minus PARAMS: RON=1 ROFF=100MEG
VT=1.5 VH=.5
S5 NodePlus NodeMinus 8 0 smoothSW
EBctrl 8 0 Value = { IF ( V(plus)-V(minus) > V(ref), 1, 0 ) }
EBref ref1 0 Value = { IF ( V(8) > 0.5, {VT-VH}, {VT+VH} ) }
Rdel ref1 ref 70
Cdel ref 0 100p IC={VT+VH}
Rconv1 8 0 10Meg
Rconv2 plus 0 10Meg
Rconv3 minus 0 10Meg
.model smoothSW VSWITCH (RON={RON} ROFF={ROFF} VON=1 VOFF=0)
.ends SWhyste
```

If-Then-Else expressions in the E and G elements are also used for various logic and controlling functions. In them, mathematical equations using Boolean combinations of node voltages and branch currents are utilized.

For example,

```
GB1 33 2 Value= { IF ( V(5) > 2.5 & V(11) > 4.3 , -.014 , 0 ) }
EB19 44 0 Value= { IF ( V(22)<1 , 2 , IF ( V(30)<1 & V(42)<1 , 2 , V(30) ) ) }
```

If you are trying to use the models with other SPICE based simulators you will have to make sure that the simulator supports these extensions to the basic SPICE primitive set.

If you need one or more of the models translated to another SPICE syntax please contact AEi Systems directly.

Chapter 3 - Using the Power IC Model Schematic Examples

Schematic Examples

The Power IC Model Library for PSpice includes a number of application test circuit examples. Some are simple; others are fairly complicated and mimic the actual applications circuit found in the data sheet. All should run in a few minutes or less on most computers.

The example schematic designs can be found in two folders on the distribution CD. Under the PowerLib folder is an Examples folder. That contains the schematics for the OrCAD Capture system. The MicroSim folder contains approximately the same set of examples for the MicroSim Schematics environment.

A master library file has been created called Power_EMA_AEI.lib. It is in the Libraries folder along with the rest of the model libraries. You can refer to this file in the in order to include all of the models in the parts database for either OrCAD Capture Library Configuration or the MicroSim Schematics Library and Include Files... function (Analysis menu)

The MicroSim Schematics editor may not be able to find the symbol library files depending on where you choose to install them. You can correct this problem by pointing to the symbol library file on your hard disk using the Editor Configuration... function in the Options menu.

Most parts have an equivalent test circuit matched to their name. In some cases, where there is a family of parts there may be fewer test circuits than parts, but the parts are normally

interchangeable, in so far as the test circuit is concerned, allowing the same test circuit to serve the part family.

The test circuits will allow you to explore the basic functionality of the model, if not its entire range of features.

Types of Simulations

Simulation Run Times

The time it takes to run a switch mode power supply simulation is directly related to a number of factors:

- Your computer's performance
- Complexity of the model and external circuitry
- Length of the simulation
- Time step of the simulator

The last item is driven by a number of factors including the stimulus and loading, soft-start and compensation components, the simulator .OPTION tolerances, the TMAX timestep setting, and the overall frequency content of the circuit (edge speeds).

It is not uncommon for SMPS simulations to take 15-60 minutes each. However, most of the example simulations run in just a few minutes.

Startup Simulations

Startup simulations can take a long time to run depending on the conditions and compensation settings. You can recognize these simulations in the examples because they normally don't use the UIC (use initial conditions) transient directive and use VCC/stimulus settings that start at zero and are pulsed on to their terminal voltage. In addition, initial conditions on input, output or compensation capacitors are not utilized.

Steady State, Line, Load Transient Simulations

In order to speed up the simulation of these types of analyses it is best to try to set initial conditions on key storage elements. This helps to get the circuit running at or near steady state almost immediately. This is as opposed to running a startup type simulation and delaying the data taking interval until steady is achieved. To do this, the UIC option is used and IC= directives are inserted, especially on input output, and compensation capacitors.

Correct initial conditions can allow a circuit that would normally take several milliseconds of time to start to get to steady state in a few hundred microseconds.

Incorrect initial conditions, whether they are set by the `.NODESET`, `.IC`, or `IC=` directives can cause the PSpice simulation to take much longer than if it were started from a zero voltage state due to transient residues or can cause convergence problems. In some circuits the initial conditions on compensation components can be very sensitive with respect to simulation settling time, with a few tenths of a volt making a huge difference in settling/runtime.

Except in cases where the compensation is internal, the models in the Power IC Model Library are setup to allow you to achieve steady state by setting initial conditions on elements external to the part.

Simulation Convergence – Quick Fix

If you encounter a convergence problem change the `.OPTIONS` settings you are using to the following:

- **Abstol = 0.01u** **(Default=1p)**
- **Vntol = 10u** **(Default=1u)**
- **Gmin=0.1n** **(Default=1p)**
- **Reltol = 0.01** **(Default=0.001)**
- **ITL4 = 500** **(Default=10)**

This should cure most simulation convergence problems unless there is an error in your circuit description.

Switching simulations refer to simulations which have a significant number of repetitive cycles, such as those found in SMPS simulations. Most of the simulation you perform with the Power IC Models will be of this type.

SMPS simulations can experience a large number of rejected time points. Rejected time points are due to the fact that PSpice has a dynamically varying time step which is controlled by constant tolerance values (`Reltol`, `Abstol`, `Vntol`). An event that occurs during each cycle, such as the switching of a power semiconductor, can trigger a reduction in the time step value. This is caused by the fact that PSpice attempts to maintain a specific accuracy, and adjusts the time step in order to accomplish this task. The time step is increased after the event, until the next cycle, when it is again reduced. This time step hysteresis can cause an excessive number of unnecessary calculations. To correct this problem, we can regress to a SPICE 2 methodology and force the simulator to have a fixed time step value.

To force the time step to be a fixed value, set the `Trtol` value to 25, i.e. `.OPTIONS TRTOL=25`. The default value is 7. The `Trtol` parameter controls how far ahead in time SPICE tries to jump. The value of 25 causes PSpice to try to jump far ahead. Then set the

Tmax value (maximum allowed time step) in the .TRAN statement to a value which is between 1/10 and 1/100 of the switching cycle period. This has the opposite effect; it forces the time step to be limited. Together, they effectively lock the simulator time step to a value which is between 1/10 and 1/100 of the switching cycle period, and eliminate virtually all of the rejected time points. These settings can result in over a 100% increase in speed!

Note: In order to verify the number of accepted and rejected time points, you may issue the .OPTIONS ACCT parameter and view the data at the end of the output file.

If this does not help the simulation converge proceed to the next section which has more details.

Simulation Convergence

The answer to a nonlinear problem, such as those in the SPICE DC and Transient analyses, is found via an iterative solution. For example, PSpice makes an initial guess at the circuit's node voltages and then, using the circuit conductances, calculates the mesh currents. The currents are then used to recalculate the node voltages, and the cycle begins again. This continues until all of the node voltages settle to values which are within specific tolerance limits. These limits can be altered using various .Options parameters such as Reltol, Vntol, and Abstol.

If the node voltages do not settle down within a certain number of iterations, the DC analysis will issue an error message such as “No convergence in DC analysis”, “Singular Matrix”, or “Source Stepping Failed”. PSpice will then halt the run because both the AC and transient analyses require an initial stable operating point in order to proceed. During the transient analysis, this iterative process is repeated for each individual time step. If the node voltages do not settle down, the time step is reduced and PSpice tries again to determine the node voltages. If the time step is reduced beyond a specific fraction of the total analysis time, the transient analysis will issue the error message, “Time step too small,” and the analysis will be halted.

Convergence problems come in all shapes, sizes, and disguises, but they are usually related to one of the following:

- Circuit Topology
- Device Modeling
- Simulator Setup

The DC analysis may fail to converge because of incorrect initial voltage estimates, model discontinuities, unstable/bistable operation, or unrealistic circuit impedances. Transient analysis failures are usually due to model discontinuities or unrealistic circuit, source, or

parasitic modeling. In general, you will have problems if the impedances, or impedance changes, do not remain reasonable. Convergence problems will result if the impedances in your circuit are too high or too low.

The various solutions to convergence problems fall under one of two types. Some are simply band-aids which merely attempt to fix the symptom by adjusting the simulator options. Other solutions actually affect the true cause of the convergence problems.

The following techniques can be used to solve a large number of convergence problems. When a convergence problem is encountered, you should start at solution 0 and proceed with the subsequent suggestions until convergence is achieved. The sequence of the suggestions is structured so that they can be incrementally added to the simulation. The sequence is also defined so that the initial suggestions will be of the most benefit. Note that suggestions which involve simulation options may simply mask the underlying circuit instabilities. Invariably, you will find that once the circuit is properly modeled, many of the “options” fixes will no longer be required!

General Discussion

Many power electronics convergence problems can be solved with the .OPTIONS Gmin parameter. Gmin is the minimum conductance across all semiconductor junctions. The conductance is used to keep the matrix well conditioned. Its default value is 1E-12mhos. Setting Gmin to a value between 1n and 10n will often solve convergence problems. Setting Gmin to a value which is greater than 10n may cause convergence problems.

PSpice does not always converge when relaxed tolerances are used. One of the most common problems is the incorrect use of the .Options parameters. For example, setting the tolerance option, Reltol, to a value which is greater than .01 will often cause convergence problems.

Setting the value of Abstol to 1u will help in the case of circuits that have currents which are larger than several amps. Again, do not overdo this setting. Setting Abstol to a value which is greater than 1u may cause more convergence problems than it will solve.

After you've performed a number of simulations, you will discover the options which work best for your circuit. Very often various options will be needed as the circuit topology is developed. Invariably, you will find that after you have debugged your circuit representation, and if your components are well modeled, most of the options can be removed.

If all else fails, you can almost always get a circuit to simulate in a transient simulation if you begin with a zero voltage/zero current state. This makes sense if you consider the fact that the simulation always starts with the assumption that all voltages and currents are zero. The simulator can almost always track the nodes from a zero condition. Running the simulation will often help uncover the cause of the convergence failure.

The above recommendation is only true if your circuit is constructed properly. Most of the time, minor mistakes are the cause of convergence problems. Error messages will help you track down the problems, however, a good technique is to scan each line of the netlist and look for anomalies. It may be tedious, but it's a proven way to weed out mistakes.

Not all convergence failures are a result of the PSpice software! Convergence failures may identify many circuit problems. Check your circuits carefully, and don't be too quick to blame the software.

DC Convergence Solutions

0. Check the circuit topology and connectivity.

Common mistakes and problems:

- Make sure that all of the circuit connections are valid. Also, verify component polarity.
- Check for syntax mistakes. Make sure that you used the correct SPICE units (i.e. MEG instead of M(milli) for 1E6).
- Make sure that there is a DC path from every node to ground.
- Make sure that voltage/current generators use realistic values, especially for rise and fall time
- Make sure that dependent source gains are correct, and that E/G element expressions are reasonable. If you are using division in an expression, verify that division by zero cannot occur or protect against it with a small offset in the denominator.

1. Increase I_{TTL1} to 400 in the .OPTIONS statement.

Example: `.OPTIONS ITL1=400`

This increases the number of DC iterations that PSpice will perform before it gives up. In all but the most complex circuits, further increases in I_{TTL1} won't typically aid convergence.

2. Add .NODESETs

Example: `.NODESET V(6)=0`

View the node voltage/branch current table in the output file. PSpice produces one even if the circuit does not converge. Add .NODESET values for the top level circuit nodes (not the subcircuit nodes) that have unrealistic values. You do not need to nodeset every node. Use a .NODESET value of 0V if you do not have a better estimation of the proper DC voltage. Caution is warranted, however, for an inaccurate Nodeset value may cause undesirable results.

3. Add resistors and use the OFF keyword.

Example: D1 1 2 DMOD OFF
 RD1 1 2 100MEG

Add resistors across diodes in order to simulate leakage. Add resistors across MOSFET drain-to-source connections to simulate realistic channel impedances. This will make the impedances reasonable so that they will be neither too high nor too low. Add ohmic resistances (RC, RB, RE) to transistors. Use the .Options statement to reduce Gmin by an order of magnitude.

Next, you can also add the OFF keyword to semiconductors (especially diodes) that may be causing convergence problems. The OFF keyword tells PSpice to first solve the operating point with the device turned off. Then the device is turned on, and the previous operating point is used as a starting condition for the final operating point calculation.

4. Use PULSE statements to turn on DC power supplies.

Example: VCC 1 0 15 DC
 becomes VCC 1 0 PULSE 0 15

This allows the user to selectively turn on specific power supplies. This is sometimes known as the “Pseudo-Transient” start-up method. Use a reasonable rise time in the PULSE statement to simulate realistic turn on. For example,

V1 1 0 PULSE 0 5 0 1U

will provide a 5 volt supply with a turn on time of 1 μ s. The first value after the 5 (in this case, 0) is the turn-on delay, which can be used to allow the circuit to stabilize before the power supply is applied.

5. Add UIC (Use Initial Conditions) to the .TRAN statement.

Example: .TRAN .1N 100N UIC

Insert the UIC keyword in the .TRAN statement. Use Initial Conditions (UIC) will cause PSpice to completely bypass the DC analysis. You should add any applicable .IC and IC= initial conditions statements to assist in the initial stages of the transient analysis. Be careful when you set initial conditions, for a poor setting may cause convergence difficulties.

AC Analysis Note: Solutions 4 and 5 should be used only as a last resort, because they will not produce a valid DC operating point for the circuit (all supplies may not be turned on and circuit may not be properly biased). Therefore, you cannot use solutions 4 and 5 if you want to perform an AC analysis, because the AC analysis must be preceded by a valid operating point solution. However, if your goal is to proceed to the transient analysis, then solutions 4 and 5 may help you and may possibly uncover the hidden problems which plague the DC analysis.

Transient Convergence Solutions

0. Check circuit topology and connectivity.

This item is the same as item 0 in the DC analysis.

1. Set RELTOL=0.01 or 0.005 in the .OPTIONS statement.

Example: `.OPTIONS RELTOL=0.01`

This option is encouraged for most simulations, since the reduction of Reltol can increase the simulation speed by 10 to 50%. Only a minor loss in accuracy usually results. A useful recommendation is to set Reltol to 0.01 for initial simulations, and then reset it to its default value of .001 when you have the simulation running the way you like it and a more accurate answer is required. Setting Reltol to a value less than .001 is generally not required.

2. Set ITL4=500 in the .OPTIONS statement.

Example: `.OPTIONS ITL4=500`

This increases the number of transient iterations that SPICE will attempt at each time point before it gives up. Values which are greater than 500 or 1000 won't usually bring convergence.

3. Reduce the accuracy of ABSTOL/VNTOL if current/voltage levels allow it.

Example: `.OPTION ABSTOL=1N VNTOL=1M`

Abstol and Vntol should be set to about 8 orders of magnitude below the level of the maximum voltage and current. The default values are Abstol=1p and Vntol=1u. These values are generally associated with IC designs.

4. Realistically Model Your Circuit; add parasitics, especially stray/junction capacitance.

The idea here is to smooth any strong nonlinearities or discontinuities. This may be accomplished via the addition of capacitance to various nodes and verifying that all semiconductor junctions have capacitance. Other tips include:

- Use RC snubbers around diodes.
- Add capacitance for all semiconductor junctions (3pF for diodes, 5pF for BJTs if no specific value is known).
- Add realistic circuit and element parasitics.
- Watch the real-time waveform display and look for waveforms that transition vertically (up or down) at the point during which the analysis halts. These are the key nodes which you should examine for problems.
- If the .Model definition for the part doesn't reflect the behavior of the device, use a subcircuit representation. This is especially important for RF and power devices such as RF BJTs and power MOSFETs. Many model vendors cheat and try to "force fit" the SPICE .MODEL statement in order to represent a device's behavior. This is a sure sign that the vendor has skimmed on quality in favor of quantity. Primitive level 1 or 3 .MODEL statements CAN NOT be used to model most

devices above 200MEGhz because of the effect of package parasitics. And .MODEL statements CAN NOT be used to model most power devices because of their extreme nonlinear behavior. In particular, if your vendor uses a .MODEL statement to model a power MOSFET, throw away the model. It's almost certainly useless for transient analysis.

5. Reduce the rise/fall times of the PULSE sources.

Example: VCC 1 0 PULSE 0 1 0 0 0
becomes VCC 1 0 PULSE 0 1 0 1U 1U

Again, we are trying to smooth strong nonlinearities. The pulse times should be realistic, not ideal. If no rise or fall time values are given, or if 0 is specified, the rise and fall times will be set to the TSTEP value in the .TRAN statement.

6. Add UIC (Use Initial Conditions) to the .TRAN line.

Example: .TRAN .1N 100N UIC

If you are having trouble getting the transient analysis to start because the DC operating point can't be calculated, insert the UIC keyword in the .TRAN statement (skip initial transient solution). UIC will cause PSpice to completely bypass the DC analysis. You should add any applicable .IC and IC= initial conditions statements to assist in the initial stages of the transient analysis. Be careful when you set initial conditions, for a poor setting may cause convergence difficulties.

Modeling Tips

Device modeling is one of the hardest steps encountered in the circuit simulation process. It requires not only an understanding of the device's physical and electrical properties, but also a detailed knowledge of the particular circuit application. Nevertheless, the problems of device modeling are not insurmountable. A good first-cut model can be obtained from data sheet information and quick calculations, so the designer can have an accurate device model for a wide range of applications.

Data sheet information is generally very conservative, yet it provides a good first-cut of a device model. In order to obtain the best results for circuit modeling, follow the rule: "Use the simplest model possible". In general, the SPICE component models have default values that produce reasonable first order results. Here are some helpful tips:

- Don't make your models any more complicated than they need to be. Overcomplicating a model will only cause it to run more slowly, and will increase the likelihood of an error.
- Remember: modeling is a compromise.
- Don't be afraid to pull apart your circuit and test individual sections or even models, especially the ones you did not create.

- Create subcircuits which can be run and debugged independently. Simulation is just like being at the bench. If the simulation of the entire circuit fails, you should break it apart and use simple test circuits to verify the operation of each component or section.
- Document the models as you create them. If you don't use a model often, you might forget how to use it.
- Be careful when you models which have been produced by hardware vendors. Many have limitations on the operating point bounds for which they can be used.
- Semiconductor models should always include junction capacitance and the transit time (AC charge storage) parameters.
- If the .Model definition for a large geometry device doesn't reflect the behavior of the device, use a subcircuit representation.
- Be careful when using behavioral models for power devices. Many models are not thoroughly tested and work at one operating point but are highly inaccurate at other operating points.
- And lastly, there is no substitute for knowing what you're doing!!

Chapter 4 - Library Listings

Please see the file Power_Library_List_3.0.pdf for an electronic version of these listings.

An asterisk (*) or red entry in the left column indicates that the model has been recently added.

Power FET Drivers

Power MOS/IGBT Drivers	Vendor	Library	Part Description	Application Schematic File Name
IR2110	IR	IR_Driver	Hi and Lo Side Drivers	IR2110Test
IR2110S	IR	IR_Driver	Hi and Lo Side Drivers	IR2110STest
RIC7113	IR	IR_Driver	Hi and Lo Side Drivers	RIC7113Test
SI4724CY	Vishay	Vishay	N-Channel Synchronous MOSFETs with Break-Before-Make, See SI4724CY.pdf	SI4724CYTest
SI4768CY	Vishay	Vishay	N-Channel Synchronous MOSFETs with Break-Before-Make	SI4768CYTest
SI4770CY	Vishay	Vishay	N-Channel Synchronous MOSFETs with Break-Before-Make	SI4770CYTest
Sic710DD	Vishay	Vishay	Half-Bridge FET Driver	Sic710DD
Sic720DD	Vishay	Vishay	Half-Bridge FET Driver	SIC720DD
SIP41101	Vishay	Vishay	Half-Bridge FET Driver	SIP41101
HP2100	Intersil	Intersil	100VDC – 2A Half Bridge Driver	HP2100Test
HP2101	Intersil	Intersil	100V Half Bridge N-Channel	HP2101Test
HP6601B	Intersil	Intersil	MOSFET Driver, Dual N-Channel	HP6601BTest
HP6602B	Intersil	Intersil	Synchronous Rectified Buck MOSFET	HP6602BTest
MIC4416	Micrel	Micrel	1.2A-Peak Low-Side MOSFET Driver	
MIC4417	Micrel	Micrel	1.2A-Peak Low-Side MOSFET Driver	
MIC4420	Micrel	Micrel	6A-Peak Low-Side MOSFET Driver	
MIC4429	Micrel	Micrel	6A-Peak Low-Side MOSFET Driver	
MIC4421	Micrel	Micrel	9A-Peak Low-Side MOSFET Driver	
MIC4422	Micrel	Micrel	9A-Peak Low-Side MOSFET Driver	
MIC4421A	Micrel	Micrel	9A-Peak Low-Side MOSFET Driver	
MIC4422A	Micrel	Micrel	9A-Peak Low-Side MOSFET Driver	
MIC4423	Micrel	Micrel	3A-Peak Low-Side MOSFET Driver	
MIC4424	Micrel	Micrel	3A-Peak Low-Side MOSFET Driver	MIC4424, MICREL_Test
MIC4451	Micrel	Micrel	12A-Peak Low-Side MOSFET Driver	
MIC4452	Micrel	Micrel	12A-Peak Low-Side MOSFET Driver	
TPS2834 *	TI	TI_Power	Synchronous-Buck MOSFET Drivers With Deadtime Control	TPS2834_5_App
TPS2835 *	TI	TI_Power	Synchronous-Buck MOSFET Drivers With Deadtime Control	TPS2834_5_App
UCC37323 *	TI	TI_Power	Dual 4 A Peak High Speed Low-Side Power MOSFET Drivers	UCC37324 Test Circuit
UCC37324 *	TI	TI_Power	Dual 4 A Peak High Speed Low-Side Power MOSFET Drivers	UCC37324 Test Circuit
UCC37325 *	TI	TI_Power	Dual 4 A Peak High Speed Low-Side Power MOSFET Drivers	UCC37324 Test Circuit

Linear ICs

Linear	Vendor	Library	Part Description	Application Schematic File Name
AD524S	AD	ADI_Linear	Analog Multiplier, See AD524S.pdf	NA
AD534T	AD	ADI_Linear	Instrumentation Amp, See AD534T.pdf	NA
AD536A	AD	ADI_Linear	Integrated Circuit True RMS to DC Converter	AD536A Test dB, AD536A Test
AD636	AD	ADI_Linear	Low Level, True RMS to DC Converter	AD636C, AD636C_dB
AD637	AD	ADI_Linear	High Precision, Wideband RMS to DC Converter	dB Test, Test
AD736	AD	ADI_Linear	Low Cost, Low Power, True RMS to DC Converter	Test (AD736G)
AD737F	AD	ADI_Linear	Low Cost, Low Power, True RMS to DC Converter	Test (AD737F)
AD8000 *	AD	ADI_Linear	1.5 GHz Ultra-High Speed Op Amp with Power-Down	AD8000 Test
AD8003 *	AD	ADI_Linear	Triple, 1.5 GHz Op Amp	AD8003
AD8099	AD	ADI_Linear	Op-amp, See AD8099.pdf	AD8099Test
AD8133 *	AD	ADI_Linear	Triple Differential Driver With Output Pull-Down	AD8133_Closed
AD8137	AD	ADI_Linear	Low Cost, Low Power 12-Bit Differential ADC Driver	AD8137
AD8139	AD	ADI_Linear	Ultra Low Noise Fully Differential ADC Driver	AD8139
AD8206	AD	ADI_Linear	Single-Supply 42V System Difference Amplifier	AD8206
AD8214 *	AD	ADI_Linear	High Voltage Threshold Detector	AD8214 Test
AD8330	AD	ADI_Linear	Low Cost DC to 150 MHz Variable Gain Amplifier	AD8330_App
AD8331, AD8331_LNA, AD8331_VGA	AD	ADI_Linear	Single VGA with Ultralow Noise Preamplifier and Programmable Rin	AD8331
AD8332	AD	ADI_Linear	Dual VGA with Ultralow Noise Preamplifier and Programmable Rin	AD8332
AD8333	AD	ADI_Linear	DC to 50 MHz Dual I/Q Demodulator and Phase Shifter	ad8333p, AD8333TEST2, AD8333TEST3
AD8334	AD	ADI_Linear	Quad VGA with Ultralow Noise Preamplifier and Programmable Rin	AD8334
AD8335 *	AD	ADI_Linear	Quad Low Noise, Low Cost Variable Gain Amplifier	AD8335_APP
ADA4860	AD	ADI_Linear	High Speed, Low Cost, Op Amp	Test (ADA4860-1)
ADA4861	AD	ADI_Linear	High Speed, Low Cost, Triple Op Amp	Test (ADA4861-3)
ADA4862	AD	ADI_Linear	High Speed, G = +2, Low Cost, Triple Op Amp	Test (ADA4862-3)

Power ICs

Power IC Models	Vendor	Library	Part Description	Application Schematic File Name
HS117RH	Intersil	Intersil	Radiation Hardened Adjustable Positive Voltage Linear Regulator	HS117, HS117_AC
ISL6225	Intersil	Intersil	PWM Controller, Dual, Regulated Output Voltage 0.9V-5.5V	ISL6225Avg
ISL6520a	Intersil	Intersil	PWM Controller, +5V Input, VOUT 0.8V Min @ 1.5%, 300kHz	ISL6520ATRAN
ISL6520Assa	Intersil	Intersil	Average model	ISL6225AVG
ISL6721	Intersil	Intersil	Single-Ended Current Mode PWM controller	ISL6721TRAN
ISL6721Av	Intersil	Intersil	Single-Ended Current Mode PWM controller, Average model	ISL6721AVG
ISL6740	Intersil	Intersil	PWM controller for half bridge and bus converter, See ISL6740switching.pdf	NA
ISL6740av	Intersil	Intersil	Average model, See ISL6740average.pdf	ISL6740Avg
ISL6741av	Intersil	Intersil	PWM controller for hard-switched full bridge and push-pull applications, Average model	ISL6741Avg
LT1242	Linear Tech	LT_Power	High Speed Current Mode Pulse Width Modulators	LT1242Test
LT1242S	Linear Tech	LT_Power	State space average model	
LT1243	Linear Tech	LT_Power	High Speed Current Mode Pulse Width Modulators	LT1243
LT1243S	Linear Tech	LT_Power	State space average model	
LT1244	Linear Tech	LT_Power	High Speed Current Mode Pulse Width Modulators	LT1244
LT1244S	Linear Tech	LT_Power	State space average model	
LT1245	Linear Tech	LT_Power	High Speed Current Mode Pulse Width Modulators	LT1245
LT1245S	Linear Tech	LT_Power	State space average model	
ML4863	Microlinear	Microlinear	Boost Regulators for Battery Powered Applications	ML4863Test

Power IC Models	Vendor	Library	Part Description	Application Schematic File Name
TL494 *	TI	TI_Power	PWM Control Circuit	TL494 STEADY STATE, TL494 STARTUP
TL494avg *	TI	TI_Power	Average TL494 model	TL494AVG BODE
TPS40007 *	TI	TI_Power	Synchronous Buck,300kHz	TPS40007_APP
TPS40007Avg *	TI	TI_Power	Synchronous Buck,300kHz Average model	TPS40007AVG
TPS40009 *	TI	TI_Power	Synchronous Buck,600KHz	TPS40009
TPS40009Avg *	TI	TI_Power	Synchronous Buck,600KHz	TPS40009AVG
TPS40040 *	TI	TI_Power	Synchronous Buck Converter,300kHz	TPS40040_1_APP
TPS40040Avg *	TI	TI_Power	Synchronous Buck Converter,300kHz State-Space Average model	TPS40040avg
TPS40041 *	TI	TI_Power	Synchronous Buck Converter,600kHz	TPS40040_1_APP
TPS40041Avg *	TI	TI_Power	Synchronous Buck Converter,600kHz State-Space Average model	TPS40041avg
TPS40042 *	TI	TPS40042 (in Examples\TI\TPS40042)	Low Pin Count, Low Vin, Synchronous Buck DCDC Controller with Tracking	tps40042
TPS40055 *	TI	TI_Power	Wide-input Synchronous Buck Controller	TPS40055
40055MOD2 *	TI	TI_Power	State Space Model	TPS40055_SSA
TPS40060 *	TI	TI_Power	Wide-Input Synchronous Buck Controller	TPS40060_AppV2
TPS40060_Avg *	TI	TI_Power	Wide-Input Synchronous Buck Controller State-Space Average model	TPS40060 Average
TPS40061 *	TI	TI_Power	Wide-Input Synchronous Buck Controller	TPS40061_AppV2
TPS40061_Avg *	TI	TI_Power	Average model	TPS40061 Average
TPS40075 *	TI	TI_Power	Midrange Input Synchronous Buck Controller With Voltage Feed-Forward	TPS40075_APPV1
TPS40075avg *	TI	TI_Power	Feed-Forward State-Space Average model	TPS40075 Average Packaged
TPS40090 *	TI	TI_Power	High Frequency Multi-phase Controller; Parameter TRI=1	TPS40090_1_APP
TPS40090avg *	TI	TI_Power	High Frequency Multi-phase Controller State-Space Average model	TPS40090avg
TPS40091 * (use 40090, Set TRI = 0)	TI	TI_Power	difference between the two models is that the passed model parameter TRI=1 for TPS40090 (disables tri-state feature) and TRI=0 for the TPS40091 (enables tri-state)	TPS40090_1_APP (average same as '090)
TPS40140 *	TI	TPS40140 (in Examples\TI\TPS40140)	Stackable 2 Channel Multiphase or 2 Channel Independent Output Controller	tps40140
TPS40180 *	TI	TPS40180 (in Examples\TI\TPS40180)	Stackable 2 Channel Multiphase or 2 Channel Independent Output Controller	tps40180
TPS40190 *	TI	TI_Power	Synchronous Buck Converter	SLUU232
TPS40190Avg *	TI	TI_Power	Synchronous Buck Converter Average Model	TPS40190avg
TPS40192 *	TI	TI_Power	Synchronous Buck w/P Good, 600KHz Separate State-Space Average model simulation provided	TPS40192_AppV4, TPS40192AVG
TPS40193 *	TI	TI_Power	Synchronous Buck w/pP Good ,300KHz Separate State-Space Average model simulation provided	TPS40193_APPV3, TPS40193 AVG
TPS40195 *	TI	TI_Power	Synchronous Buck w/P Good and Sync Separate State-Space Average model simulation provided	TPS40195_SubV2, Average
TPS40200 *	TI	TI_Power	Wide-Input Non-Synchronous Buck Controller	APPLICATION (requires tps40200app.lib)
TPS40200_avg *	TI	TI_Power	Wide-Input Non-Synchronous Buck Controller State-Space Average model	TPS40200 average
TPS40210_0 *	TI	TI_Power	Current Mode Boost	LED_Application, TPS40210 Boost App
TPS40210Avg *	TI	TI_Power	Current Mode Boost State-Space Average model	TPS40210AVG
TPS40211 *	TI	TPS40211 (in Examples\TI\TPS40211)	Wide Input Range Current Mode Boost Controller	tps40211_trans_steady
TPS40211_Startup *	TI	TPS40211 (in Examples\TI\TPS40211)	Wide Input Range Current Mode Boost Controller	tps40211_trans_start
TPS40211Avg *	TI	TPS40211 (in Examples\TI\TPS40211)	Wide Input Range Current Mode Boost Controller State Space Average Model	tps40211_avg
TPS40222 *	TI	TI_Power	1.6A 1.25Mhz Buck State-Space Average model	TPS40222NEW
TPS40222_Avg *	TI	TI_Power	1.6A 1.25Mhz Buck State-Space Average model	TPS40222_AVG
TPS51100 *	TI	TI_Power	3A Sink/Source DDR Regulator	TPS51100_APP (two versions, one for AC, one for Transient)
TPS51113 *	TI	TPS51113 (in Examples\TI\TPS51113)	4.5V to 13.2V Synchronous Buck Controller with High Current Gate Driver, 300kHz	tps51113
TPS51116 *	TI	TPS51116 (in Examples\TI\TPS51116)	DDR1, DDR2, DDR3 Switcher and LDO	tps51116
TPS51117 *	TI	TI_Power	Synchronous Step-down On-Timer Controller	TPS51117 Steady State,TPS51117 Start UP
TPS51124H *	TI	TI_Power	Synchronous Step-down On-Timer Controller	BUCK_EVM_SLUU252, INTEL
TPS51124A *	TI	TI_Power	Synchronous Step-down On-Timer Controller	

Power IC Models	Vendor	Library	Part Description	Application Schematic File Name
TPS51163 *	TI	TPS51163 (in Examples\TI\TPS51163)	4.5V to 13.2V Synchronous Buck Controller with High Current Gate Driver, 600kHz	tps51163
TPS51200 *	TI	TI_Power	Sink/Source DDR Termination Regulator	TPS51200_APP, TPS51200_AVG
TPS51315 *	TI	TPS51315 (in Examples\TI\TPS51315)	3V to 14V, 10A Synchronous Step Down Converter with D-CAP™ Mode	tps51315
TPS54140 *	TI	TPS54140 (in Examples\TI\TPS54140)	3.5V to 42V Input, 1.5 A Step Down SWIFT™ Converter with Eco-Mode™	tps54140
TPS54160 *	TI	TPS54160 (in Examples\TI\TPS54160)	3.5V to 60V, 1.5A Step Down SWIFT™ Converter with Eco-	tps54160
TPS54283 *	TI	TI_Power	Non Synchronous Converter W/integrated HS FET,300kHz 2A	3P3V AVERAGE, 5V AVERAGE
TPS54286 *	TI	TI_Power	Non Synchronous Converter W/integrated HS FET,600kHz 2A	3P3V AVERAGE, 5V AVERAGE
TPS54383 *	TI	TI_Power	Non Synchronous Converter W/integrated HS FET,300kHz 3A	TPS54x8x, TPS54383AVG5V
TPS54386 *	TI	TI_Power	Non Synchronous Converter W/integrated HS FET,600kHz 3A State-Space Average model	3P3V AVERAGE, 5V AVERAGE
TPS54418 *	TI	TPS54418 (in Examples\TI\TPS54418)	2.95V to 6V Input, 4A, 2MHz Synchronous Step Down SWIFT™ DCDC Converter	tps54418_trans
TPS61020 *	TI	TPS61020 (in Examples\TI\TPS61020)	Adjustable, 1.5-A Switch, 96% Efficient Boost Converter with Down-Mode, QFN-10	tps61020_trans
TPS61020_Avg *	TI	TPS61020 (in Examples\TI\TPS61020)	Adjustable, 1.5-A Switch, 96% Efficient Boost Converter with Down-Mode, QFN-10 Average Model	tps61020avg
TPS62240_Trans *	TI	TPS6224x (in Examples\TI\TPS6224x)	2.25MHz 300mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62240_trans
TPS62242_Trans *	TI	TPS6224x (in Examples\TI\TPS6224x)	2.25MHz 300mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62242_trans
TPS62243_Trans *	TI	TPS6224x (in Examples\TI\TPS6224x)	2.25MHz 300mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62243_trans
TPS62260_Avg *	TI	TI_Power	2.25MHz 600mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62260_Avg
TPS62260 *	TI	TI_Power	2.25MHz 600mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62260_trans
TPS62261 *	TI	TI_Power	2.25MHz 600mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62261_trans
TPS62262 *	TI	TI_Power	2.25MHz 600mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62262_trans
TPS62263 *	TI	TI_Power	2.25MHz 600mA Step-Down Converter in 2x2mm SON/TSOT23 Package	tps62263_trans
TPS62290 *	TI	TPS62290 (in Examples\TI\TPS6229x)	2.25MHz 1A Step-Down Converter in 2x2mm SON	tps62290_pspice_trans
TPS62291 *	TI	TPS62291 (in Examples\TI\TPS6229x)	2.25MHz 1A Step-Down Converter in 2x2mm SON	tps62291_trans
TPS62293 *	TI	TPS62293 (in Examples\TI\TPS6229x)	2.25MHz 1A Step-Down Converter in 2x2mm SON	tps62293_trans
TPS65563_0 *	TI	TI_Power	Integrated Photo Flash Charger and IGBT Driver	tps65563
UA723	TI	TI_Power	Precision Voltage Regulator	UA723Test
UC1524	TI	TI_Power	Advanced Regulating Pulse Width Modulators	UC1524Test
UC1524A	TI	TI_Power	Advanced Regulating Pulse Width Modulators	
UC1525	TI	TI_Power	Advanced Regulating Pulse Width Modulators	
UC1525A	TI	TI_Power	Advanced Regulating Pulse Width Modulators	
UC1637	TI	TI_Power	Switched Mode Controller for DC Motor Drive	UC1637SplitSupply, UC1637SingleSupply
UC1823	TI	TI_Power	High Speed PWM Controller	UC1823Test
UC1823A	TI	TI_Power	High Speed PWM Controller	
UC1824	TI	TI_Power	High Speed PWM Controller	UC1824Test
UC1825	TI	TI_Power	High Speed PWM Controller	UC1825Test
UC1825A	TI	TI_Power	High Speed PWM Controller	
UC1832 *	TI	TI_Power	Precision Low Dropout Linear Controller	UC1832_3_APP
UC1833 *	TI	TI_Power	Precision Low Dropout Linear Controller - See LoadStep	UC1832_3_APP
UC1842	TI	TI_Power	Current Mode PWM Controller	
UC1842A	TI	TI_Power	Current Mode PWM Controller	UC1842StateSpace, UC1842Test
UC1842AS	TI	TI_Power	State Space Average Model	
UC1842S	TI	TI_Power	State Space Average Model	
UC1843	TI	TI_Power	Current Mode PWM Controller	
UC1843A	TI	TI_Power	Current Mode PWM Controller	
UC1843AS	TI	TI_Power	State Space Average Model	UC1843ASTest
UC1843S	TI	TI_Power	State Space Average Model	
UC1844	TI	TI_Power	Current Mode PWM Controller	
UC1844A	TI	TI_Power	Current Mode PWM Controller	
UC1844AS	TI	TI_Power	State Space Average Model	
UC1844S	TI	TI_Power	State Space Average Model	
UC1845	TI	TI_Power	Current Mode PWM Controller	
UC1845A	TI	TI_Power	Current Mode PWM Controller	
UC1845AS	TI	TI_Power	State Space Average Model	
UC1845S	TI	TI_Power	State Space Average Model	
UC1846	TI	TI_Power	Current Mode PWM Controller	UC1846TRAN, UC1846_Package
UC1846SSA *	TI	TI_Power	Current Mode PWM Controller State Space Average Model	
UC1847 *	TI	TI_Power	Current Mode PWM Controller	

Power IC Models	Vendor	Library	Part Description	Application Schematic File Name
UC1871	TI	TI_Power	Resonant Fluorescent Lamp Driver	UC1871Test
UC1872	TI	TI_Power	Resonant Fluorescent Lamp Ballast Controller	UC1872Test
UC1875	TI	TI_Power	Phase Shift Resonant Controller	UC1875Test
UC1876	TI	TI_Power	Phase Shift Resonant Controller	
UC1901 *	TI	TI_Power	Isolated Feedback Generator	UC1901
UC1901Avg *	TI	TI_Power	Isolated Feedback Generator	UC1901 AVERAGE
UC3842B	On Sem	TI_Power	Current Mode, See UC384x.pdf	
UC3843B	On Sem	TI_Power	Current Mode, See UC384x.pdf	UC3843BFW
UC3844B	On Sem	TI_Power	Current Mode, See UC384x.pdf	
UC3845B	On Sem	TI_Power	Current Mode, See UC384x.pdf	UC384XFlyback
UC3854Bs	TI	TI_Power	Enhanced High Power Factor Preregulator, State space	UC3854Test
UC3854s	TI	TI_Power	Enhanced High Power Factor Preregulator, State space	
UC39432 *	TI	TI_Power	Precision Analog Controller Same model for transient and AC simulations	uc39432_app
UCC1806	TI	TI_Power	Low Power, Dual Output, Current Mode PWM Controller	UCC1806Test
UCC28019 *	TI	TI_Power	Continuous conduction Mode PFC Same model for transient and AC simulations	UCC28019 Packaged, UCC28019Avg
UCC2813-0 *	TI	UCCX813-0 (in Examples\TIUCCX813-X)	Low Power Economy BiCMOS Current Mode PWM	uccx813-0
UCC2813-1 *	TI	UCCX813-1 (in Examples\TIUCCX813-X)	Low Power Economy BiCMOS Current Mode PWM	uccx813-1
UCC2813-2 *	TI	UCCX813-2 (in Examples\TIUCCX813-X)	Low Power Economy BiCMOS Current Mode PWM	uccx813-2
UCC2813-4 *	TI	UCCX813-4 (in Examples\TIUCCX813-X)	Low Power Economy BiCMOS Current Mode PWM	uccx813-4
UCC2813-5 *	TI	UCCX813-5 (in Examples\TIUCCX813-X)	Low Power Economy BiCMOS Current Mode PWM	uccx813-5
UCC2817 *	TI	TI_Power	BiCMOS Power Factor Pre Regulator - Average Model Same model for transient and AC simulations	Startup, Bode
UCC2818 *	TI	TI_Power	BiCMOS Power Factor Pre Regulator - Average Model Same model for transient and AC simulations	Startup, Bode
UCC289x_Average *	TI	UCC2891_AVG (in Examples\TIUCC2891AVG)	Current Mode Active Clamp PWM Controller State-Space Average model	UCC289xAvg
UCC2891 *	TI	TI_Power	Current Mode Active Clamp PWM Controller Generic State-Space Average model simulation provided	UCC289X_App, UCC289xAvg
UCC2892 *	TI	TI_Power	Current Mode Active Clamp PWM Controller Generic State-Space Average model simulation provided	
UCC2893 *	TI	TI_Power	Current Mode Active Clamp PWM Controller Generic State-Space Average model simulation provided	UCC289X_App, UCC289xAvg
UCC2894 *	TI	TI_Power	Current Mode Active Clamp PWM Controller Generic State-Space Average model simulation provided	UCC289X_App, UCC289xAvg
UCC28C40 *	TI	TI_Power	BiCMOS Current Mode PWM	UCC28C40
UCC28C40s *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C41 *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C41s *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C42 *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C42s *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C43 *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C43s *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C44 *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C44s *	TI	TI_Power	BiCMOS Current Mode PWM	UCC28C4X Average
UCC28C45 *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28C45s *	TI	TI_Power	BiCMOS Current Mode PWM	
UCC28600 *	TI	UCC228600 (in Examples\TIUCC22860)	Quasi-Resonant Flyback Green-Mode Controller Low-Power BiCMOS Current-Mode PWM	
UCC3800 *	TI	TI_Power	Low-Power BiCMOS Current-Mode PWM	
UCC3801 *	TI	TI_Power	Low-Power BiCMOS Current-Mode PWM	
UCC3802 *	TI	TI_Power	Low-Power BiCMOS Current-Mode PWM Separate State-Space Average model simulation provided	UCC3802, UCC3802EVM Steady State
UCC3803 *	TI	TI_Power	Low-Power BiCMOS Current-Mode PWM Separate State-Space Average model simulation provided	UCC3803 EVM, UCC3803 EVM average
UCC3804 *	TI	TI_Power	Low-Power BiCMOS Current-Mode PWM	
UCC3805 *	TI	TI_Power	Low-Power BiCMOS Current-Mode PWM	
UCC3809-1	TI	TI_Power	Economy Primary Controller	
UCC3809-2	TI	TI_Power	Economy Primary Controller including state space model Separate State-Space Average model simulation provided	FLYBACKStartup, FLYBACKSteady, FLYBACKLoop
UCC3895_Average *	TI	UCC3895Average (in Examples\TIUCC3895AVG)	BiCMOS Advanced Phase Shift PWM Controller Separate State-Space Average model simulation provided	UCCx895_Average
UCC3895 *	TI	TI_Power	BiCMOS Advanced Phase Shift PWM Controller Separate State-Space Average model simulation provided	UCC3895Test, UCC3895, UCC3895 Transient

Power IC Models	Vendor	Library	Part Description	Application Schematic File Name
CS322	On Semi	ON_Power	High Speed PWM Controller	CS322Test
CS324	On Semi	ON_Power	High Speed PWM Controller	CS324Test
CS51220	On Semi	ON_Power	Feed Forward Voltage Mode PWM Controller with Programmable Synchronization	
CS51411	On Semi	ON_Power	1.5A 260kHz Low Voltage Buck Regulators	CS51411Test
CS5155	On Semi	ON_Power	CPU 5-Bit Synchronous Buck Controller	CS5155Test
CS5156	On Semi	ON_Power	CPU 5-Bit Nonsynchronous Buck Controller	CS5165Test
CS5171	On Semi	ON_Power	1.5 A 280kHz Boost Positive Feedback Regulators	cs517x.dsn
CS5172	On Semi	ON_Power	1.5 A 280kHz Boost Negative Feedback Regulators	
CS5173	On Semi	ON_Power	1.5 A 560kHz Boost Positive Feedback Regulators	
CS5174	On Semi	ON_Power	1.5 A 560kHz Negative Feedback Boost Regulators	
CS5307	On Semi	ON_Power	Four-Phase VRM 9.0 Buck Controller	
CS5308	On Semi	ON_Power	Two-Phase PWM Controller with Integrated Gate Drivers for VRM 8.5	CS5308Test
CS5322	On Semi	ON_Power	Two-Phase Buck Controller with Integrated Gate Drivers and 5-Bit DAC	CS5322TEST
CS5323	On Semi	ON_Power	Three-Phase Buck Controller with 5-Bit DAC	
MC33063	On Semi	ON_Power	1.5A Step-Up/Down/Inverting Switching Regulator	MC33063BOOSTTest, MC33063BUCKTest
MC33064	On Semi	ON_Power	1.5A Step-Up/Down/Inverting Switching Regulator	
MC33161	On Semi	ON_Power	Universal Voltage Monitor	MC33161TEST
MC33201	On Semi	ON_Power	Low Voltage, Rail-to-Rail, Single Operational Amplifier	MC3320XACTEST
MC33202	On Semi	ON_Power	Low Voltage, Rail-to-Rail, Single Operational Amplifier	
MC33204	On Semi	ON_Power	1V, Rail-to-Rail, Single Operational Amplifier	
MC33262	On Semi	ON_Power	Power Factor Controller	MC33262Test
MC33363	On Semi	ON_Power	High Voltage Switching Regulator	AF2
MC33501	On Semi	ON_Power	1V, Rail-to-Rail, Single Operational Amplifier	MC3350XACTEST
MC33502	On Semi	ON_Power	1V, Rail-to-Rail, Single Operational Amplifier	
MC33503	On Semi	ON_Power	1V, Rail-to-Rail, Single Operational Amplifier	
MC34063	On Semi	ON_Power	3.4A Step-Up/Down/Inverting Switching Regulator	MC34063 PSpice, MC34063BUCKBOOSTTest
MC34163	On Semi	ON_Power	3.4A Step-Up/Down/Inverting Switching Regulator	MC34163Test
NCP100	On Semi	ON_Power	Adjustable 0.9-6V \pm 1.7% Output Voltage 0.1-20mA Shunt Regulator	NCP100ACTest, NCP100PULSETest, NCP100SERIESPASS
NCP1000	On Semi	ON_Power	Fixed-100kHz Switching Regulator with 700V / 0.5A Switch	NCP1000Test
NCP1000A	On Semi	ON_Power	Fixed-100kHz Switching Regulator with 700V / 0.5A Switch, average model	NCP1000AVG#1, NCP1000AVG#2, NCP1000AVGTEST
NCP1001	On Semi	ON_Power	Fixed-100kHz Switching Regulator with 700V / 1A Switch	
NCP1002	On Semi	ON_Power	Fixed-100kHz Switching Regulator with 700V / 1.5A Switch	
NCP1203AV	On Semi	ON_Power	PWM Current-Mode Controller average model	
NCP1203P100	On Semi	ON_Power	100kHz PWM Current-Mode Controller for Universal Off-Line Supplies	
NCP1203P40	On Semi	ON_Power	40kHz PWM Current-Mode Controller for Universal Off-Line Supplies	NCP1203
NCP1203P60	On Semi	ON_Power	60kHz PWM Current-Mode Controller for Universal Off-Line Supplies	
NCP1400ASN19T1	On Semi	ON_Power	Up to 100mA, 1.9V, 180kHz Boost PWM Switching Regulator with Enable	NCP1400ASN19Test
NCP1400ASN30T1	On Semi	ON_Power	Up to 100mA, 3.0V, 180kHz Boost PWM Switching Regulator with Enable	NCP1400ASN30Test
NCP1400ASN50T1	On Semi	ON_Power	Up to 100mA, 5.0V, 180kHz Boost PWM Switching Regulator with Enable	NCP1400ASN50Test
NCP1570	On Semi	ON_Power	Low Voltage Synchronous Buck Controller	NCP1570Test
NCP1571	On Semi	ON_Power	Low Voltage Synchronous Buck Controller	NCP1571Test
NCP1653	On Semi	ON_Power	Compact, Fixed-Frequency, Continuous Conduction Mode PFC Controller	Switching
NCP1653Avg	On Semi	ON_Power	Compact, Fixed-Frequency, Continuous Conduction Mode PFC Controller	Average
NCV4269	On Semi	ON_Power	Micropower 150mA LDO Linear Regulator	NCV4269 Line Transient, NCV4269 LoadTransient, NCV4269
NCV8403 *	On Semi	ON_Power	Self-Protected Low Side Driver with Temp and Current Limit, 42V, 14A	NCV8403Test
TL431	On Semi	ON_Power	Adjustable 2.5-36V \pm 1% Tolerance 1-100mA Shunt Regulator	LDOREGULATOR
TLV431A	On Semi	ON_Power	Low Voltage Precision Adjustable Shunt Regulator	LDOREGULATOR, TLV431Test
HA16163	Renesas	Renesas	Synchronous Phase Shift Full-Bridge Control IC, 480 kHz, See Application Circuit.pdf	
LM117	National	Nat_Power	3-terminal adjustable regulator	LM117, LM177_AC
LM78S40	National	Nat_Power	Universal Switching Regulator Subsystem	78S40Test
LP2953	National	National_LDO	Adjustable Micropower Low-Dropout Voltage Regulator, See LP2953A.pdf	LM2953ATest
TNY256	PI	PI_Power	TinySwitch with line under-voltage lockout, auto-restart	TNY256Test

Semiconductors

Semiconductors	Vendor	Library	Part Description	Application Schematic File Name
FETs *: 57034, 57130, 57230, IRF7452, IRF7466, IRF6216, IRF8113, IRF7828, IRF7832, IRF7834, IRFRC20,	IR	IR_Semi	Power Mosfets. See Improved Mosfet Model.pdf	
CMPD2004, CMPD3003, CMPD6001, CMDSH2-3, CMDSH-3, CMPD6263, CMHSH5-4, CMHSH5-2L, CMSH1-40M, CMSH1-60M, CMSH5-40, CMSH5-60, CMSH2-40M, CMSH2-60M, CSHD10-45L	CS	CS_Diodes	Diodes, General	
CCL0035, CCL0130, CCL0300, CCL0500, CCL0750, CCL1000, CCL1500, CCL2000, CCL2700, CCL3500, CCL4500, CCL5750, CCLH080, CCLH100, CCLH120, CCLH150	CS	CS_Current_Diodes	JFET Current Regulators	
CMPTA44, CMPTA94, CMPT404A	CS	CS_BJTs	BJTs	
MI1020T	Marlow	Misc	Thermal-Electro Cooler. See TEC.pdf	
53259, 53111, 53124, 53253, 53250	Micropac	Micropac_Relays	Solid-State Relays, Switches. See 53111.pdf & 53250.pdf	53111Test, 53250Test
8CLJQ045, 8CLJQ045_Sub	IR	IR_Semi	Power Schottky. See 8CLJQ045.pdf	
RHRP1540	Fairchild	Misc	Soft Recovery Diode	
SSR8045P	SSDI	Misc	Power Schottky	
SRH615A-1, SRH615A-2, SRH615A-3, SRH615A-4	Vishay	Vishay	Optocoupler, Hi-Rel 5300Vrms	
SRH610A-1, SRH610A-2, SRH610A-3, SRH610A-4	Vishay	Vishay	Optocoupler, Hi-Rel 5300Vrms	
MOC8101, MOC8107, MOC8108	Fairchild	Misc	Optocoupler, Hi-Rel 5300Vrms	
CNY17F-1, CNY17F-2, CNY17F-3, CNY17F-4	Fairchild	Misc	Optocoupler, Hi-Rel 5300Vrms	
Misc BJTs *: MPS750, MMBT2222ALT1, MMBT2907A, PN2222A, 2N2907A, 2N2222A, MMBT3904TT1, 2N4401		Misc	BJTs	
Misc FETs *: MTD1N60E, MTD3022T4, HAT2168H, HAT2167H, HUF75345S3S, SI7415DN, PH6325L, PH2625L, HAT2165, SI7846DP, FDS6898A, SI4866DY, SI4862DY, SI7366DP, SI7866ADP, SI4840DY, SI4848DY, SI4850EY, SI7860ADP, SI7880ADP		Misc	Power Mosfets	
Misc Diodes *: 1N4001, 1N4148, 1N5617, 1N5806, 1N5819, 1N6642US, 1N752a, 1N962B, 30BQ040, B240, BAS16L, BAS19, BAS21, BAT54H, BAT54S, BAT54T1, BZX84C12, BZX84C13, BZX84CV1, BZX84CV2, ES1A-ES1M, KBPC808, MA2ZD18, MBR140p2, MBR1645, MBR2010OCT, MBR320, MBR330, MBR340, MBR350, MBR360, MBRS330T3, MUR1620, MUR1620, MUR260, MURA110I3, MURS120T3, MURS160, MURS260, RB081L20, SSR8045, UPR10, VDT2R12B, BZX84C6V2, BAT54C		Misc	Diodes	

Magnetics

Magnetics	Vendor	Library	Part Description	Parameters
MP55xxx	Magnetics	AEiMPP55	Molypermalloy Powder core models, Part numbers 55014 - 55933	
MP58xxx	Magnetics	AEiMPP58	High Flux powder core models, Part numbers 58018 - 58933	
MPP	Generic	Mags	Molypermalloy Powder (MPP) core model. See also MP55 Series	N= # of turns U= Permeability AL= Inductance reference of the core mHy $1000T^2$ LM=Magnetic Path Length in cm DCR=Series resistance in ohms IC=Initial Conditions
MPP2	Generic	Mags	High Flux powder core model. See also MP58 Series	N= # of turns U= Permeability AL= Inductance reference of the core mHy $1000T^2$ LM=Magnetic Path Length in cm DCR=Series resistance in Ohms
Core	Generic	Mags	Generic Saturable Core model. See Magnetics Modeling.pdf Ex: VSEC=25U IVSEC=-25U LMAG=10MHY LSAT=20UHY RFEDDY=25KHZ	VSEC=Core Capacity in Volt-Sec IVSEC Initial Condition in Volt-Sec LMAG Magnetizing Inductance in Henries LSAT Saturation Inductance in Henries FEDDY Frequency when LMAG Reactance = Loss Resistance in Hz
CoreX	Generic	Mags	Generic Saturable Core model. See Magnetics Modeling.pdf	ACORE=Magnetic cross section area in cm ² LPATH=Magnetic path length in cm FEDDY=Frequency when Lmag Reactance=Loss resistance UMAX=Maximum Permeability, dB/dH USAT=Saturation Permeability, dB/dH BR=Flux density in gauss at H = 0 for saturated B- H loop BI=Initial Flux density, default = 0 N=Number of Turns
CorewHyst	Generic	Mags	Generic Saturable Core model. See Magnetics Modeling.pdf Ex: SVSEC=25U IHYST=10m IVSEC=1 LMAG=10MHY LSAT=20UHY RFEDDY=25KHZ	SVSEC=Volt-sec at Saturation = BSAT • AE • N IVSEC=Volt-sec Initial Condition = B • AE • N LMAG=Unsaturated Inductance = $\mu O \mu R \cdot N^2 \cdot AE$ /LM LSAT=Saturation Inductance = $\mu O \cdot N^2 \cdot AE$ / LM IHYST=Magnetizing I @ 0 Flux = H • LM / N REDDY=Eddy Current Loss Resistance
Transformers	Generic	Mags	Transformers, Various topologies, 1:1, Center tapped, etc.	Series resistance and turns ratio

MPP Core Model Note: The AEiMPP55.LIB and AEiMPP58.LIB model libraries contain individual models for various MPP55 and MPP58 Cores in the respective series. It is recommended that you use these individual models, as opposed to the generic MPP55 and MPP58 model versions. The parameters for the MPP55xxx and MPP58xxx models include N (the number of turns), DCR (DC resistance), and IC (initial condition).

Generic Model Templates

Generic Models				
Sandler State Space Average Models				
	Vendor	Library	Description	Parameters
Flyback	Generic	PowerSS	State Space average model for Flyback converters.	L=Primary inductance in Henries NC=Current transformer turns ratio NP=Power transformer turns ratio F=Switching frequency in Hz EFF=Efficiency RB=Current transformer burden resistor in ohms TS=Propagation delay time in the current loop in secs
Forward	Generic	PowerSS	State Space average model for Forward converters.	L=Primary inductance in Henries NC=Current transformer turns ratio NP=Power transformer turns ratio F=Switching frequency in Hz EFF=Efficiency RB=Current transformer burden resistor in ohms TS=Propagation delay time in the current loop in secs
Boost	Generic	PowerSS	State Space average model for Boost converters.	L=Primary inductance in Henries F=Switching frequency in Hz NC=Current transformer turns ratio NP=Power transformer turns ratio EFF=Efficiency RB=Current transformer burden resistor in ohms TS=Propagation delay time in the current loop in secs
Basso PWM Switching Models				
	Vendor	Library	Description	Parameters
PWMCCMVM	Generic	Basso	PWM switching model	RE=Parasitic resistance in Ohms
PWMDCMVM	Generic	Basso	PWM switching model	L=Primary inductance in Henries FS=Switching frequency in Hz
PWMVM	Generic	Basso	PWM switching model	L=Primary inductance in Henries FS=Switching frequency in Hz RE=Parasitic resistance in Ohms
PWMC	Generic	Basso	PWM switching model	L=Primary inductance in Henries FS=Switching frequency in Hz RI=Current Sense Element SE=External ramp in Vs
PWMBMVM	Generic	Basso	PWM switching model	L=Primary inductance in Henries
PWMBMCM	Generic	Basso	PWM switching model	L=Primary inductance in Henries RI=Current Sense Element
PWMBMCM2	Generic	Basso	PWM switching model	L=Primary inductance in Henries RI=Current Sense Element
Other Generic Models				
	Vendor	Library	Part Description	Parameters
CPWR	Generic	Misc	Constant Power Load	VKnee=Load is resistive below knee and then constant power for all voltages above that Power=Constant Power
SWhyste	Generic	Misc	Switch with hysteresis	Ron=On Resistance Roff=Off resistance VT=Threshold voltage (On/Off @ VT+VH, VT-VH) VH=Hysteresis voltage
CATS	Generic	Misc	Category 5 Cable	L=Length in meters
DBEHAV	Generic	Misc	Soft Recovery Diode, See subcircuit netlist for more information	IS1, TM, TAU, RMO, VTA, CAP, ISE
Tant	Generic	TantCap	Tantalum Capacitor Model with and w/o Initial Conditions, See Capacitor.pdf	C= capacitance ESR1K= ESR at 1kHz ESL=Series Inductance RLEAK=Leakage Resistance IC=Initial Conditions
DeadDrv	Generic	Dead	Dead Time for Synchronous Rectification, variable output voltage	DT = Dead time in seconds
DeadSync	Generic	Dead	Dead Time for Synchronous Rectification	DT = Dead time in seconds RS = GateUpper to SourceUpper resistance
DeadTime	Generic	Dead	Dead Time Signal Generator, Floating	DT = Dead time in seconds VHIGH = High voltage output in Volts VLOW = Low voltage output in Volts RS = GateUpper to SourceUpper resistance
NewDT	Generic	Dead	Dead Time Signal Generator similar to Deadtime but not floating	DT = Dead time in seconds VHIGH = High voltage output in Volts VLOW = Low voltage output in Volts RS = GateUpper to SourceUpper resistance
Sparkgap2	Generic	Misc	Highly nonlinear device whose function is to stop transient surges on DC or AC power supply lines.	V_GLOW = Glow discharge voltage VARC = Arc voltage ISUS = Minimum sustaining current V_BREAKDN = Break down voltage I_ARC = Minimum arc current
Rtube	Generic	Misc	Fluorescent Tube	VTHRES = Cold voltage at which the lamp strikes in Volts VARC = Voltage corresponding to the lamp arc voltage in Volts ISUS = Current under which the arc is stopped in Amps

Chapter 5 - References

General

1. “SMPS Simulation with SPICE 3”, by Steven M. Sandler, McGraw-Hill Professional; 1 edition (December 1, 1996), ISBN: 0079132278
2. “Switch-Mode Power Supply SPICE Cookbook”, by Christophe P. Basso, McGraw-Hill Professional; 1 edition (March 19, 2001), **ISBN:** 0071375090
3. “Power Specialist's App Note Book, Papers on Simulation, Modeling and More”, Edited by Charles Hymowitz, <http://www.intusoft.com/lit/psbook.zip>
4. “Inline equations offer hysteresis switch in PSpice”, Christophe Basso, On Semiconductor, EDN, August 16, 2001
5. “SPICE Circuit Handbook”, by Steven M. Sandler and Charles E. Hymowitz, McGraw-Hill Professional; 1 edition (2006), ISBN: 0071468579