

Designer Series XIV

Simulation Approaches for Power Supplies

by Dr. Ray Ridley

For as long as I've been in this industry, designers have struggled with simulating power supplies. Attempts to take full schematics and simulate them with Spice programs from CAD providers fail badly. They either cannot converge, take too long to simulate, provide inaccurate waveforms, provide little useful design information, or, in most cases, all of these things.

There are many unique issues in power supplies that make simulation difficult. The first issue is the wide separation of time constants. Modern semiconductor devices can turn on and off in a few nanoseconds, requiring very small simulation time steps to observe switching transitions. High speed switching is used to modulate waveforms at much lower frequencies. For a power factor correction circuit, for example, currents are controlled to emulate 50 or 60 Hz waveforms, and the output voltage is controlled very slowly. Loop bandwidth can be 1 Hz or less, and it requires many seconds of simulation to see long-term transient responses. A system such as this requires over nine orders of magnitude to observe and debug the system.

Secondly, power converter waveforms operate on two levels - the ideal waveforms that you may see in a textbook description of a converter, and the real-world waveforms that are modified by parasitics in the converter. The performance of a power supply is usually determined by parasitic components that do not even appear on the schematic. These include trace inductance, package inductance, packaging capacitance, transformer leakage inductance and capacitance. Most of these parasitic components are impossible to predict before the power supply is built, making simulation before a prototype difficult.

Thirdly, voltage and current excursions in modern power supplies are extreme. It is not uncommon to find semiconductors with 100 V/ns and 10 A/ns transitions in medium-technology systems. Most circuit simulation programs are not designed to handle these excursions.

Finally, while dealing with these three issues, power supplies have highly nonlinear control mechanisms, combining analog and digital signals. Pulse-width modulation is best done with analog comparison of a ramp and moving reference, derived from a very high-gain feedback circuit. The switching time is determined by the control, which then initiates the very high dv/dt and di/dt transitions.

No simulator exists that can handle this. Experienced designers of modern power supplies do not even attempt a full-scale simulation. It is deemed wasted development time. Bypassing simulation is fine for experienced designers working familiar topologies, but difficult for novices to the field who really want to know what waveforms will look like before building circuits.

To address these problems, the power electronics industry has tried many techniques over the last few decades that provide a wealth of design tools. There are several different approaches, yet no single approach yields a complete solution. However, used judiciously, they can accelerate designs and help you achieve a final product with greater confidence and lower cost.

1. Device Level Simulation

There are many circuit simulator packages available to the power electronics industry. Most of them are based on Spice, and software companies have taken the core code of Spice and put their own proprietary interface on it to enhance the circuit entry and waveform presentation process.

Spice is designed for simulation of semiconductor circuits, usually on a large scale. It can do a very nice job on digital logic designs, where semiconductor geometries are small, and where voltage and current transitions are limited to logic levels.

Attempts have been made to model power semiconductors for decades, with varying degrees of success. The relatively simple power MOSFET provides many challenges to Spice circuit modeling. It has nonlinear gain equations, nonlinear capacitances across junctions, and large device geometry. Nonetheless, circuit models are available, and some designers try to use them to simulate power supplies.

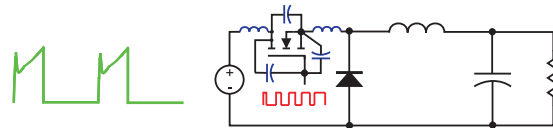


Figure 1: Complex circuit models are used for semiconductor devices, and parasitic components are included in the simulation to try to reproduce exact circuit waveforms.

The advantage of using true device models is that switching transitions can be observed and predicted. Reasonable correlation can be obtained between simulated and measured waveforms if enough time is spent refining models and parasitic values. If your main objective is characterizing these switching characteristics, this can be a useful exercise. One example that comes to mind is the application of synchronous rectifiers where device timing is crucial for efficient operation. If too much time delay is left between switching of the main power switch and the synchronous rectifier, slow internal diodes can conduct, resulting in large switching losses. If the timing is too fast, switch conduction can overlap, and a short circuit results.

A word of caution— don't trust simulation results without comparing constantly with measured waveforms. A small error in a circuit model or parasitic value can result in large changes in switching losses. Constant refinement of a model versus a prototype is needed to provide confidence in modeling. And this, of course, defeats part of the purpose in simulating as a tool to eliminate circuit development testing.

Despite many decades of model refinement, not one satisfactory model exists that explains (to all users) all phenomena observed in semiconductor operation. My favorite example is of a company that was using a MOSFET as a circuit breaker, and during normal opera-

Frequency Response Measurement



USB port compatibility.

Designed specifically for switching power supplies, the AP200 makes swept frequency response measurements that give magnitude and phase data plotted versus frequency.

Features

Control Loops

- Avoid expensive product instability
- Control loops change with line, load, and temperature
- Optimize control loops to reduce cost and size

Magnetics

- Design and specify more reliable magnetics
- Measure critical parasitic components
- Detect winding and material changes
- Characterize component resonances up to 15 MHz

Power Line Harmonics

- Check IEC compliance for AC input systems
- Measure line harmonics to 10 kHz
- Avoid expensive redesign, and minimize test facility time

Capacitors

- Measure essential data not provided by manufacturers
- Select optimum cost, size, shape, and performance

Filters

- Characterize power systems filter building blocks
- Optimize performance at line and control frequencies
- 15 MHz range shows filter effectiveness for EMI performance

Pricing & Services

Analyzer & Accessories:

Analog source/receiver unit AP200 USB* <i>includes Digital Signal Processing (DSP) unit, Interface cables, and software</i>	\$12,500
Overseas Orders	\$13,100
Differential Isolation Probes	\$650/pair
5 Hz to 15 MHz Injection Isolator	\$595
Power 4-5-6	\$995*
*discounted price available only when purchasing the AP200	

Services:

Rental Units	\$1600/month
Consulting	\$250/hr + travel expense for On-Site \$200/hr Off-Site

Frequency Range	0.01 Hz to 15 MHz
Selectivity Bandwidth	1 Hz to 1 kHz
Output Injection Isolator	5 Hz to 15 MHz 3:1 Step Down
Input Isolation	Optional 1,000 V
Averaging Method	Sweep by Sweep
PC Data Transfer	Automatic

* Free USB upgrade kit for AP200 Parallel users. Contact us for more information.



Distributed exclusively by:

Ridley Engineering

www.ridleyengineering.com
770 640 9024
885 Woodstock Rd.
Suite 430-382
Roswell, GA 30075 USA

tion, the device was kept off with high voltage applied for long periods of time. Over time, device failures were encountered. Lab testing showed that some of the devices gradually failed - after a day leaving the devices off, individual cells of the FET would turn on. The cause was traced to charges appearing on the oxide layer over time. No models have been developed to show this kind of effect. Similarly, modern MOSFET die are very large, and during switching transitions, part of the die is on, and part of it has not yet turned on. This variability across the surface of the die is not captured in circuit models.

If your company insists on this kind of simulation, just be aware that results are only as good as the models that are entered. Also, do not attempt to model an entire power circuit this way. Trying to include nonlinear device models, parasitics, and feedback loops all in one simulation is a recipe for frustration. You can expect severe convergence issues, and many hours or days trying to get a simulation to complete.

2. Ideal Switch Simulation

Many designers recognize that trying to simulate with semiconductor models for their circuit is a futile operation, and consumes far more time than it is worth. As a result, they replace the semiconductor model with an ideal switch model. Each diode or power switch is assigned two values of resistor, representing the conduction resistance when it is on, and the high value of blocking resistance when it is off.

Transitions between these two values are determined by control circuit elements, or by applied voltages to the device terminals. Depending on the simulation software used, choices must be made regarding "how" the device transitions from one value to another. Some custom-designed simulation software programs (POWER 4-5-6 for example) model the transition as an instantaneous event, greatly reducing analysis time. Generic circuit simulators such as Spice do not like instantaneous transitions, and finite rise and fall times must be provided to facilitate convergence.

Ideal switch simulation has the advantage of eliminating many of the nonlinear circuit elements that can cause simulation errors with Spice-based programs. Circuit waveforms still maintain the essential character of the real waveforms in the circuit, but, of course, transition waveforms and circuit ringing during switching are not accurately modeled.

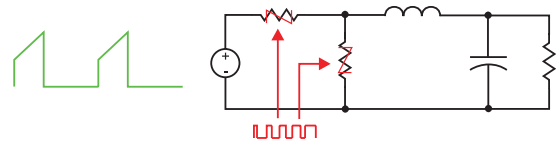


Figure 2: Semiconductors are replaced with ideal switches, which are just two different resistor values, representing on and off resistances. Selected parasitic elements may also be included in the simulation, but switching transition waveforms are no longer accurate.

Specialized circuit simulators like POWER 4-5-6 operate extremely efficiently with ideal switch modeling. They can handle feedback loops, control nonlinearities, and long-term simulation with ease. However, it is not possible to model a generic power electronics circuit, only pre-configured ones.

Once you transition to software that can handle generic schematics, the issues of convergence always return.

3. Nonlinear Average Model Simulation

The problem in switching transition convergence can be eliminated in the obvious way—by eliminating the switching from the simulation circuit. This can be done in two stages. In the first approach, the switch and diode of a buck are replaced with a special nonlinear transformer, which has a turns ratio determined by the average duty cycle of each simulation cycle.

This circuit model eliminates the switching information in the converter, but it does preserve the average value of all the waveforms. This nonlinear average model can be used to simulate converters with large disturbances, and will show nonlinear effects. It has been used effectively in the past to model audio amplifiers to simulate distortion effects.

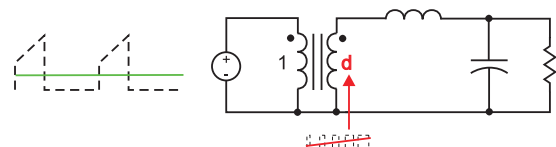


Figure 3: Average circuit model eliminates circuit switching but preserves average quantities and nonlinearities.

While this nonlinear large-signal average model is useful for control simulations, amplifier operation, and other events, it suffers from the fact that the waveforms given by the simulation are continuous, and no longer directly represent those seen in the circuit. The switch waveform, for example, is an average continuous current, and the turn-on, turn-off, and peak current information is lost. This can be a problem if using simulation to provide a backup for a worst-case design, as required in some aerospace designs.

The nonlinear average model can also suffer from convergence issues when combined with high-gain feedback systems. In my experience, 80% of circuits simulated in this manner will converge on the first attempt using the nonlinear average model. The other 20% can result in nonlinear convergence issues, and difficulties in finding steady-state operating points.

4. Linearized Average Model Simulation

A final level of model reduction is also possible. The nonlinear transformer of the previous approach can be perturbed and linearized for small-signal operation. There are several reasons we might want to do this. Firstly, the linearized models are guaranteed to converge with any Spice program. This saves a tremendous amount of time. The linearized model can produce any desired system bode plot very rapidly, and assess system stability. It can also be used for time-domain simulation to see the effect of step loads or step inputs to the system.

The linearized model can also be used for hand analytical work. When you need to do worst-case analysis for a design, analytical expressions may be needed to assess extremes of operation. Only linear circuit models can be used for this work.

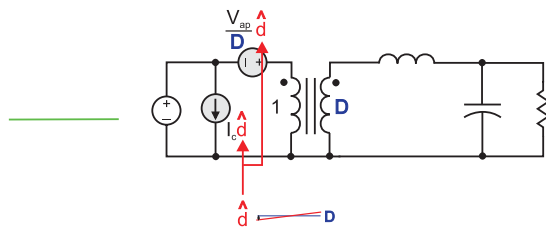


Figure 4: Averaged linearized PWM switch model. Switching ripple and nonlinear events are no longer available.

While the linearized circuit model is the fastest and easiest to simulate, it shows the least amount of waveform detail. Switching currents and voltages are obviously averaged, as with the nonlinear model, but they are also linearized around an operating point. The model can no longer be used to assess large-signal distortion, for example, of a switching amplifier. It cannot predict response across nonlinear boundaries of events such as current limiting, transition from continuous conduction mode to discontinuous conduction mode, or duty cycle limiting.

5. Which Model to Use?

Figure 5 shows a summary of the capabilities of each of the different modeling approaches. Your choice of model or models will depend upon which of these features you really need to see. The more complex and detailed the model, the lower the probability of waveform accuracy on a first simulation attempt. Likewise, the probability of convergence inside a circuit simulator will be lower.

As we mentioned previously, most experienced designers spend very little time, if any, on simulation during power supply development. Time spent will typically be used on a combination of these models, chosen for speed and usefulness in solving a particular problem. Regardless of which models you use, there is one thing that is always true: some aspect of the modeling will not match reality perfectly, and you will *always* need to build a prototype before proceeding to production. This is still a hard concept for system managers to grasp since the power supply is often viewed as a simple component of a system that, surely, does not need much effort to make it work properly.

6. Extended Small-Signal Model with Injected Ripple

At the APEC 2004 conference in California, an important new development was announced in a seminar by Dr. Vatche Vorperian. He showed that it is possible to reconstruct ripple currents and voltages in the converter, using the small-signal model only, and driving it with the ac portion of the steady-state duty cycle waveform.

This very simple idea provides a wealth of new information from the small-signal model. Peak currents and voltages are reconstructed, input and output filter per-

Only Need One Topology?

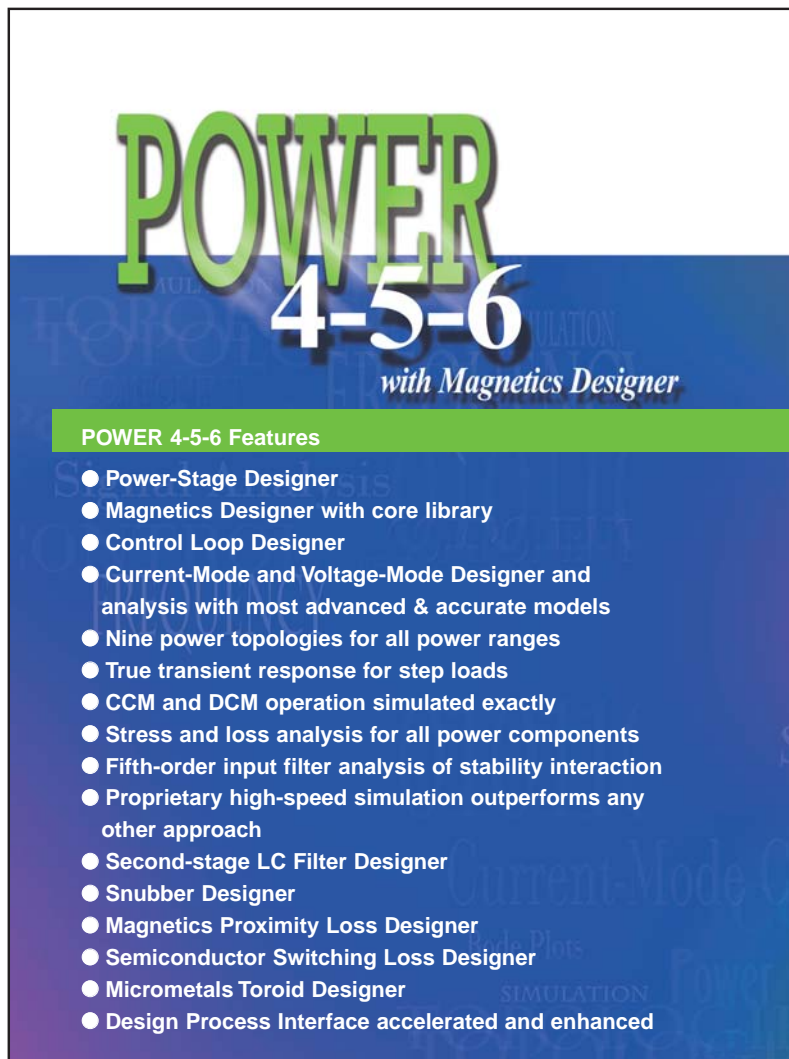
Buy a module at a time . . .

Modules

A	Buck Converter	\$295
B	Boost Converter	\$295
C	Buck-Boost Converter	\$295
D	Flyback Converter	\$595
E	Isolated Forward, Half Bridge, Full-Bridge, Push-Pull	\$595

Bundles

Bundle A-B-C	\$595
All Modules A-B-C-D-E	\$1295



POWER 4-5-6
with Magnetics Designer

POWER 4-5-6 Features

- Power-Stage Designer
- Magnetics Designer with core library
- Control Loop Designer
- Current-Mode and Voltage-Mode Designer and analysis with most advanced & accurate models
- Nine power topologies for all power ranges
- True transient response for step loads
- CCM and DCM operation simulated exactly
- Stress and loss analysis for all power components
- Fifth-order input filter analysis of stability interaction
- Proprietary high-speed simulation outperforms any other approach
- Second-stage LC Filter Designer
- Snubber Designer
- Magnetics Proximity Loss Designer
- Semiconductor Switching Loss Designer
- Micrometals Toroid Designer
- Design Process Interface accelerated and enhanced



Ridley Engineering
www.ridleyengineering.com
770 640 9024
885 Woodstock Rd.
Suite 430-382
Roswell, GA 30075 USA



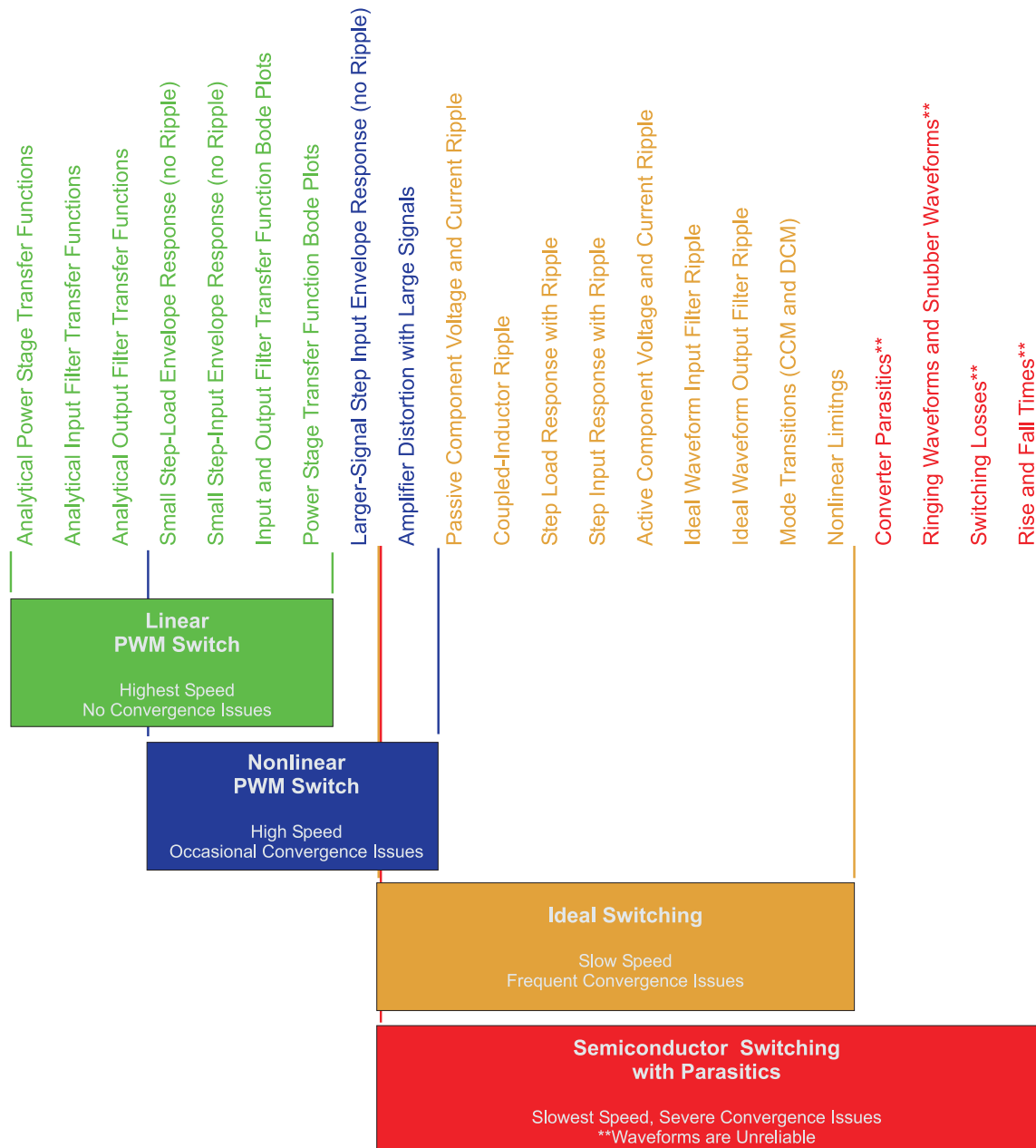


Figure 5: Four different ways to simulate switching power supplies, and the features that can be seen with each approach.

formance is predicted, and complex waveforms for converters with coupled inductors are accurately predicted. Furthermore, analytical results for ripple currents in coupled-inductor converters can be predicted. Full results and details of this model will be presented at the Power Electronics Specialists' Conference in June 2004, and we will cover this in a future issue of *Switching Power Magazine*.

I've only just started experimenting with this new model, and found it to be a powerful addition to the library of different approaches to power converter simulation.

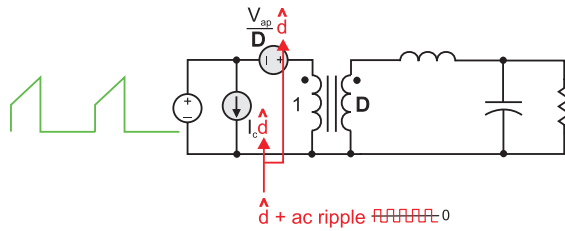


Figure 6: Linearized average PWM switch model with ripple injection

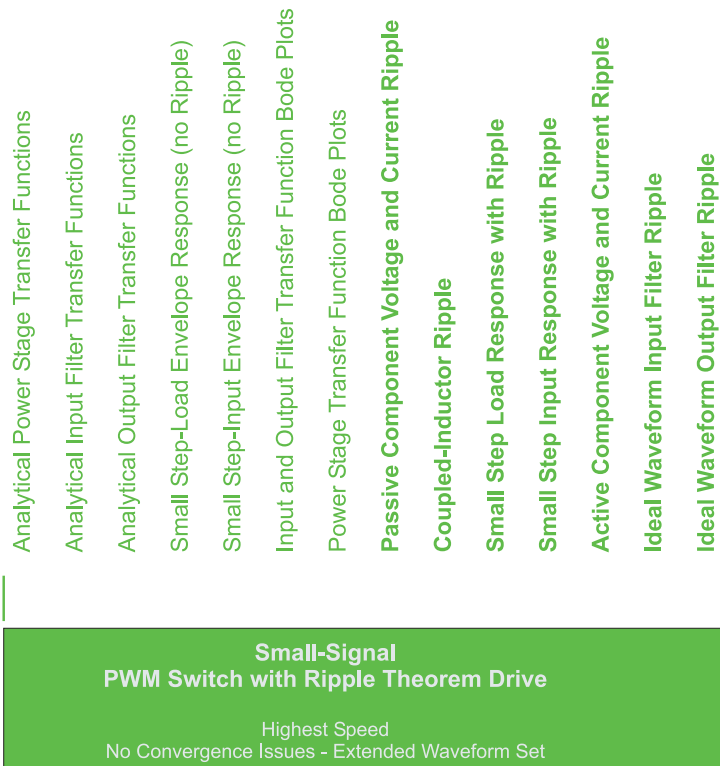


Figure 7: Extended feature set of the PWM switch model with ripple injection

Power Supply Design Workshop

Gain a lifetime
of design experience
... in four days.

Workshop Agenda

Day 1

Morning Theory

- Converter Topologies
- Inductor Design
- Transformer Design
- Leakage Inductance
- Design with Power 4-5-6

Afternoon Lab

- Design and Build Flyback Transformer
- Design and Build Forward Transformer
- Design and Build Forward Inductor
- Magnetics Characterization
- Snubber Design
- Flyback and Forward Circuit Testing

Day 2

Morning Theory

- Small Signal Analysis of Power Stages
- CCM and DCM Operation
- Converter Characteristics
- Voltage-Mode Control
- Closed-Loop Design with Power 4-5-6

Afternoon Lab

- Measuring Power Stage Transfer Functions
- Compensation Design
- Loop Gain Measurement
- Closed Loop Performance

Day 3

Morning Theory

- Current-Mode Control
- Circuit Implementation
- Modeling of Current Mode
- Problems with Current Mode
- Closed-Loop Design for Current Mode w/Power 4-5-6

Afternoon Lab

- Closing the Current Loop
- New Power Stage Transfer Functions
- Closing the Voltage Compensation Loop
- Loop Gain Design and Measurement

Day 4

Morning Theory

- Multiple Output Converters
- Magnetics Proximity Loss
- Magnetics Winding Layout
- Second Stage Filter Design

Afternoon Lab

- Design and Build Multiple Output Flyback Transformers
- Testing of Cross Regulation for Different Transformers
- Second Stage Filter Design and Measurement
- Loop Gain with Multiple Outputs and Second Stage Filters

Only 24 reservations are accepted. \$2495 tuition includes POWER 4-5-6 Full Version, lab manuals, breakfast and lunch daily. Payment is due 30 days prior to workshop to maintain reservation.

For dates and reservations, visit www.ridleyengineering.com



Ridley Engineering

www.ridleyengineering.com
770 640 9024
885 Woodstock Rd.
Suite 430-382
Roswell, GA 30075 USA