Pulsed Power Engineering
Circuit Simulation

January 12-16, 2009

Craig Burkhart, PhD
Power Conversion Department
SLAC National Accelerator Laboratory

Circuit Simulation for Pulsed Power Applications

- Uses of circuit simulation
- Tools
- Limitations
- Typical methodology

Why Simulation of Power Electronic Systems

- In research & development
  - Analyze behavior of new circuit concepts
  - ⇒ Improved understanding of circuit
- In product engineering
  - Study influence of parameters
  - Optimize circuit design and control
  - ⇒ Shorten overall design process
- Simulation results
  - Voltage and current waveforms
  - Dynamic and steady-state system performance
  - Power losses
  - Component ratings

Challenges in Numerical Simulation

- Power semiconductors introduce extreme nonlinearity
  - ⇒ Program must be able to handle switching
- Time constants differ by several orders of magnitude
  - e.g. in electrical drives
  - ⇒ Small simulation time steps
  - ⇒ Long simulation times
- Accurate models not always available
  - e.g. semiconductor devices, magnetic components
  - ⇒ Behavioral models with sufficient accuracy required
- Controller modeled along with electrical circuit
  - e.g. digital control
  - ⇒ Mixed signal simulation

6 plexim

Different Degrees of Simulation Detail

1. Power circuit modeled as linear transfer function
   - Small signal behavior
   - No switching, no harmonics

   ⇒ Controller design
2. Power circuit modeled with ideal components
   - Large signal behavior, voltage and current waveforms
   - Overall system performance

   ⇒ Circuit design and controller verification
3. Power circuit with manufacturer specific components
   - Parasitic effects (magnetic hysteresis)
   - Switching transitions (diode reverse recovery)
   - Component stress (electrical or thermal)

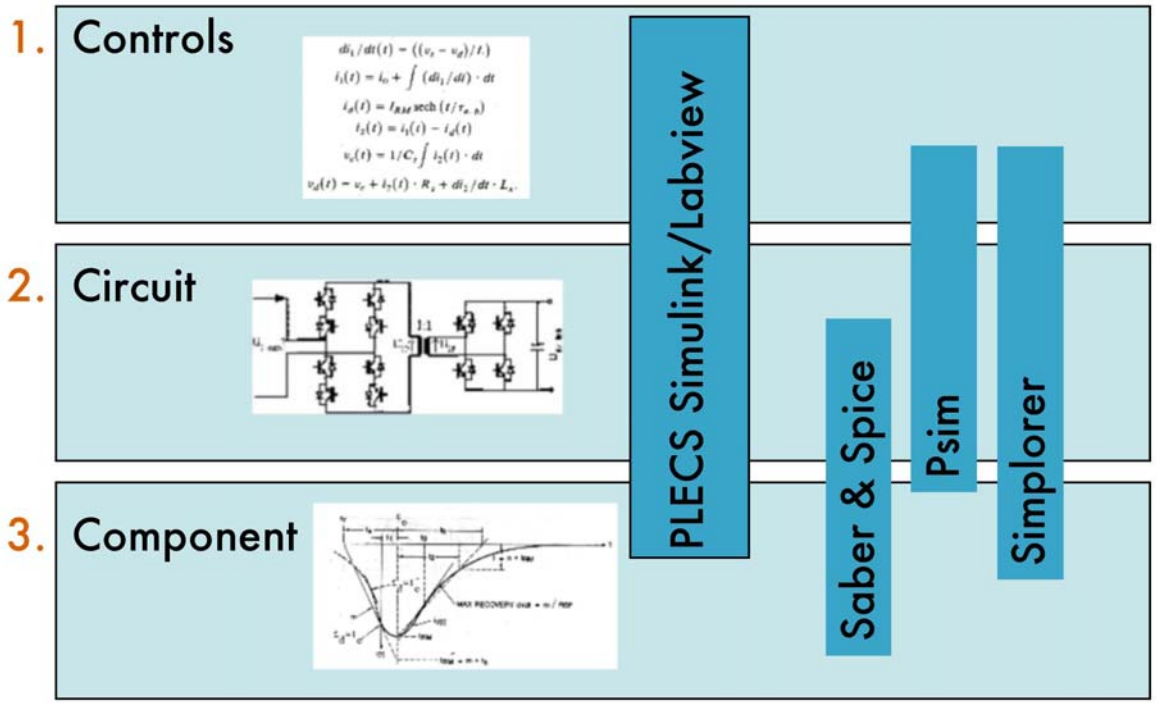
   ⇒ Choice of components

7 plexim

NATIONAL ACCELERATOR LABORATORY

Power Conversion
Solutions for Challenging Problems

Different Degrees of Simulation Detail



9 plexim

Circuit Simulation Tools

- Reduces circuit to N algebraic equations with N unknowns and solve
  - Implicit integration Methods
  - Newton’s method
  - Etc.
- Spice/PSpice (Simulation Program with Integrated Circuit Emphasis)
  - Ubiquitous circuit solver
  - Developed at UC Berkeley, first presented in 1973
  - P in PSpice stands for personal computer
  - Large parts library
  - Analog and digital circuits
- Matlab/Simulink/SimPowerSystems/PLECS
  - Matlab mathematical tool teamed with specialized toolboxes
  - Simulink/SimPowerSystems
  - PLECS
  - Developed primarily for power electronics

Circuit Simulation Tools

- PSIM
  - Developed for power electronics
  - Fast convergence
  - Matlab interface
- Simplorer
  - Developed for power electronics
  - Four modeling languages
    - VHDL-AMS for analog-mixed-signal-design
    - Circuit Simulator for the simulation of power electronic circuits
    - Block diagram simulator for the simulation of controllers and similar tasks
    - State machine simulator for event driven systems
- Saber
  - Developed for a wide range of applications, including power electronics
  - Can handle analog, digital, mixed and event driven devices
  - Can be linked to digital simulations to handle models written in Verilog or VHDL
  - Very powerful/very expensive
- Bertha
  - SNL transmission line-based

Pulsed Power Circuit Simulation

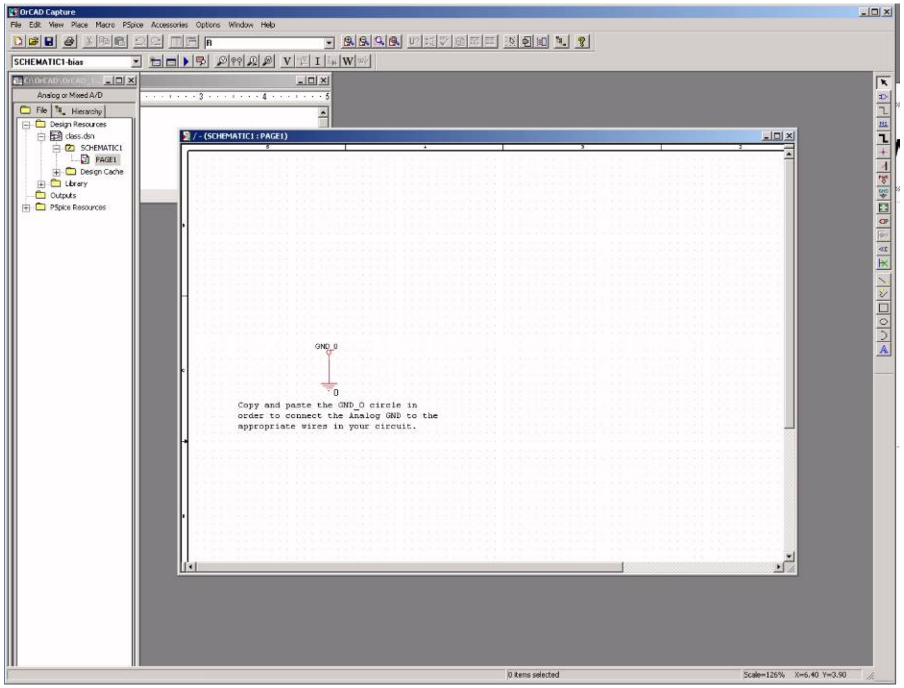
- Almost exclusively transient analysis
- Tend to generate sparse, stiff matrices, challenging for convergence
- Tips to improved convergence
  - Simplify the model as much as possible
    - Replace semiconductors with ideal switches or initial conditions
    - Use generic and/or ideal components
  - Insert resistance to ground from areas with convergence troubles
  - Avoid series elements (2 inductors → a single inductor if nothing else connects to the node between)
  - Relax tolerances
  - Avoid small time steps
  - Slow down transitions (e.g. add RC snubber)

Simulation Software Contents

- Typically included in circuit simulation programs are complementary packages.
  - Extensive part libraries for manufacturers
  - Optimization modules
  - PCB layout software
  - Specialized schematic tools
  - Advanced post-processing

Typical Work Flow

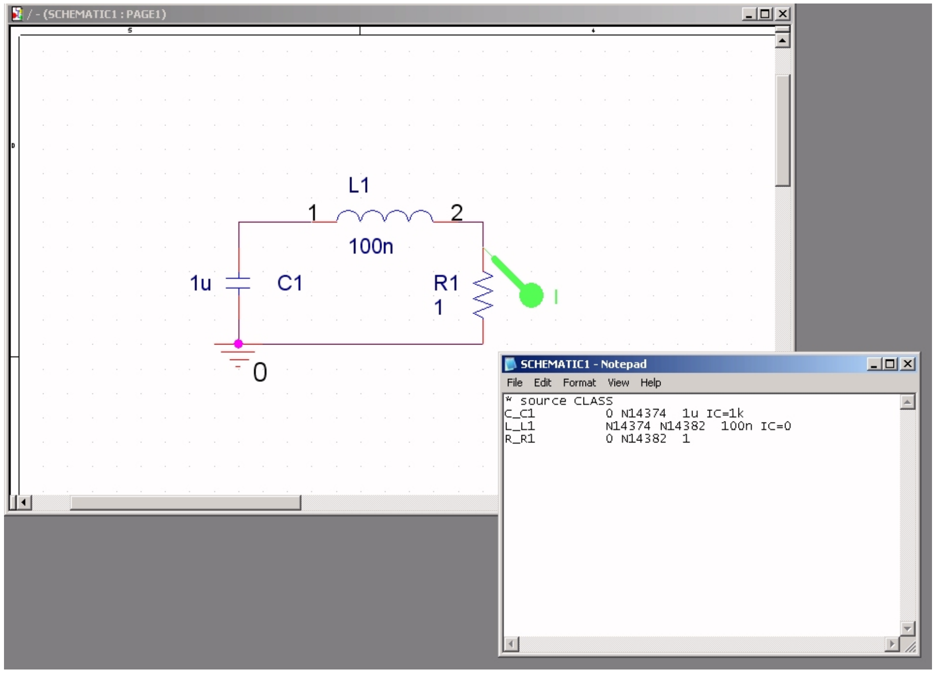
- Circuit simulation programs vary with the specific steps to solve a problem, but many share similar steps



- First, simulation directories are created and a graphical schematic interface is opened.
- For Spice simulators, the graphical interface is simply a tool to easily generate Spice netlists.

Typical Work Flow

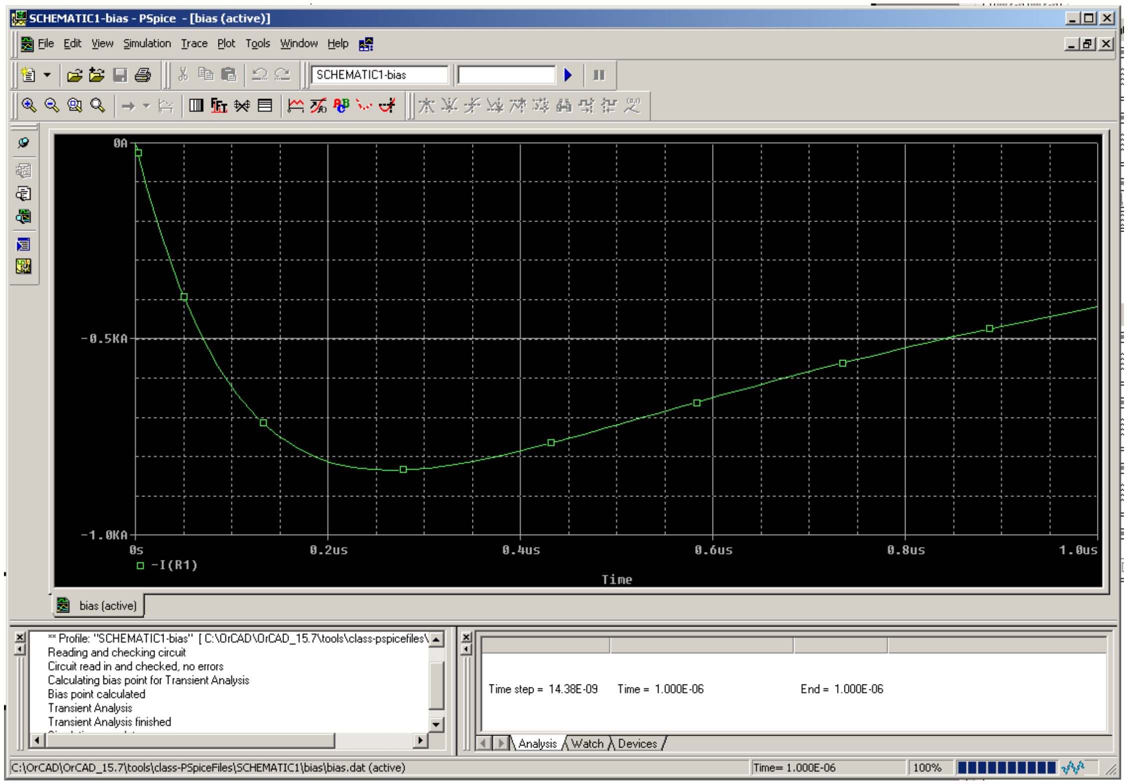
- Shown is a simple pulsed power circuit. Also shown is the Spice netlist that was generated by the software.
- Initial conditions such as voltage on a capacitor and current through an inductor are input.



- For transient, AC bias, and DC bias simulations, measurement points are typically placed at areas of interest.
- Simulation parameters such as length of simulation, maximum time step, and convergence criteria are set

Typical Work Flow

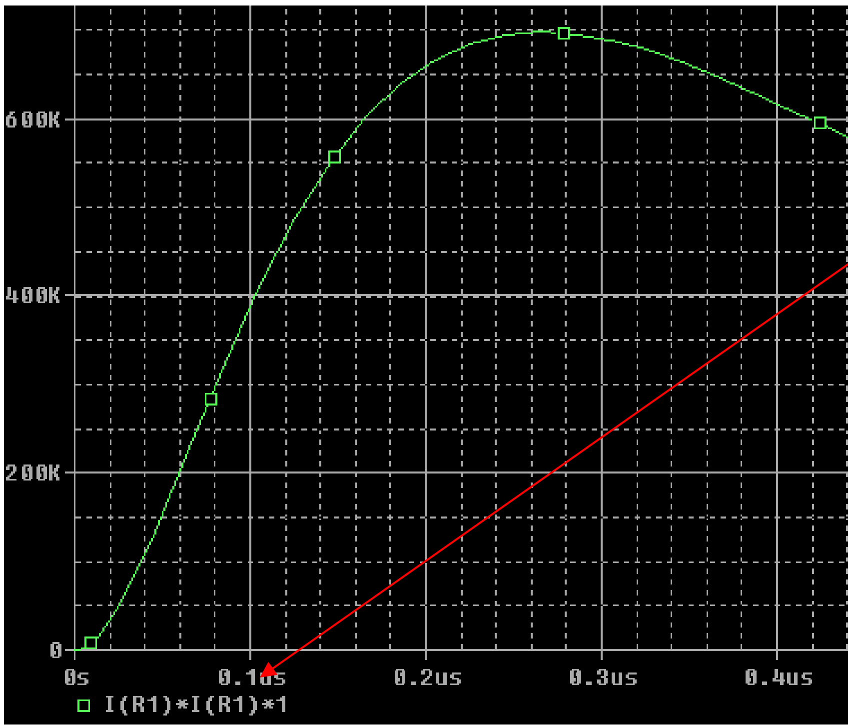
- Shown is the result in the post-processor. This example is the current through a resistor.



- Parameters such as rise-time, maximum, pulse-width, or rms value can be computed.

Typical Work Flow

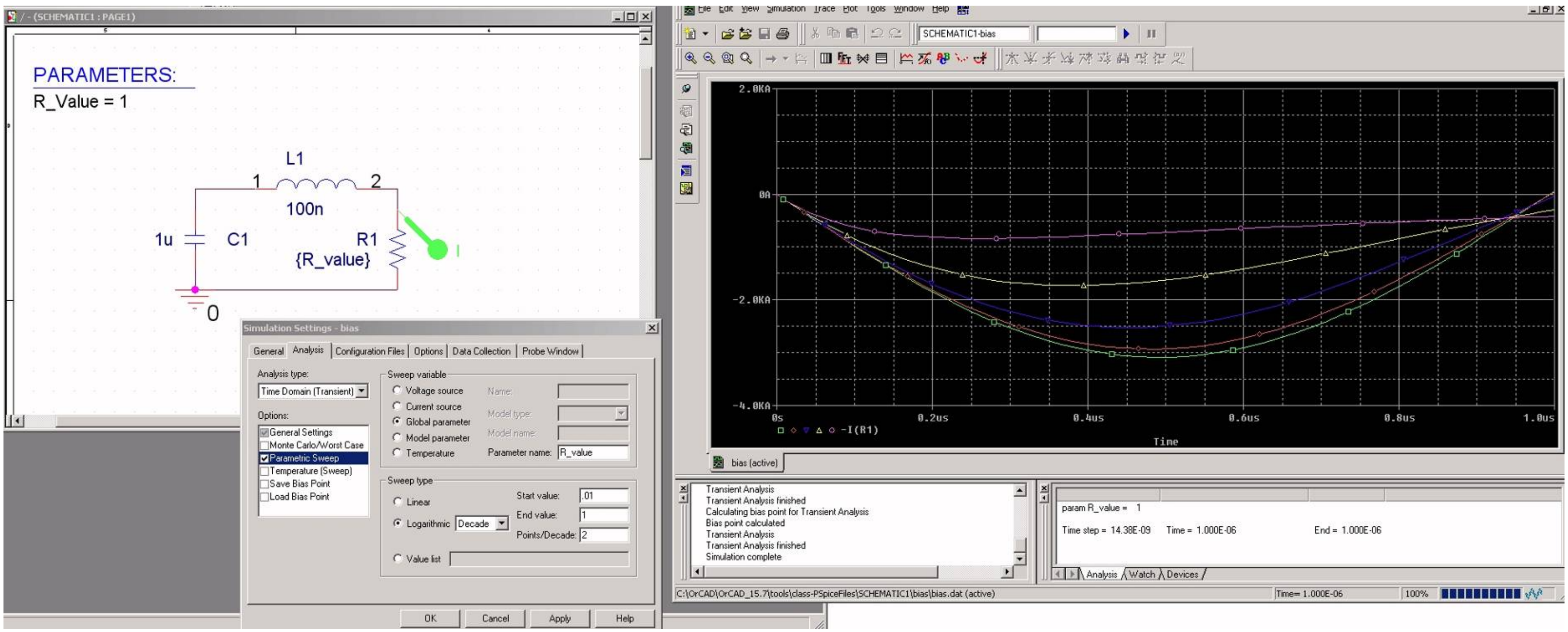
- Algebraic expressions using the simulated traces can be calculated



- In this example, the power through the resistor is calculated

Parametric Study

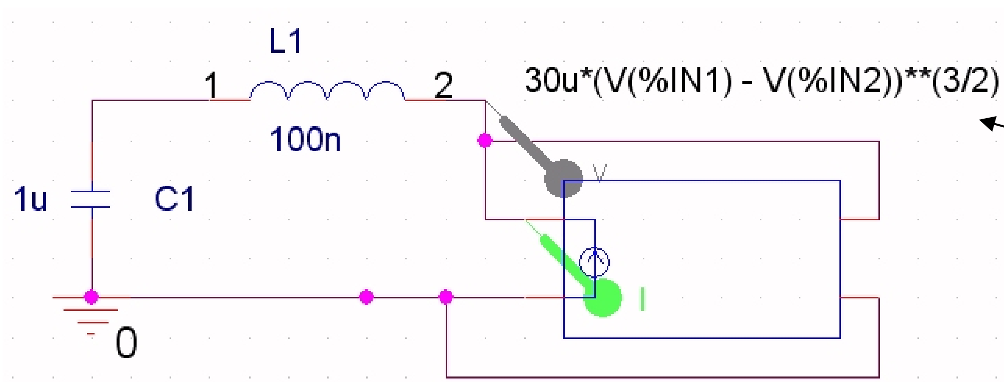
- Parametric sweeps show the circuit response if one or more of the element values is swept through a range of magnitudes.
- In this example, the resistance value is swept.



Analog Behavioral Model

- Complicated models can be represented as circuit elements using the Analog Behavioral Model (ABM) function.

- In the below example, a voltage controlled current source is used to represent a 30µP beam load.



$$I = PV^{3/2}$$

Analog Behavioral Model

- In the below example, a time varying load resistance is modeled using the TIME variable.
- The plot shows the simulated resistance.

$$I = \frac{V}{R}$$

$$= \frac{V}{0.1 + 10e^{\frac{-t}{200n}}}$$