PSpice Advance Analysis Techsession

Gil Ruiz
May 18, 2017
Why PSpice?

**PSpice Today**

- Most accurate SPICE simulator for mixed signal, SPICE based, circuit simulation
- Largest simulation model library (over 34,000 Models)
- Comprehensive ecosystem – Most IC vendors deliver PSpice® simulation models
- Tight integrations with Virtuoso®, Allegro® and OrCAD® PCB Design flows
- Tight integration with System level solutions like MATLAB/Simulink, C/C++/SystemC, Verilog A-ADMS, etc
- Most comprehensive advanced analyses capabilities to reduce design cost, improve reliability, productivity, and yield
- **NEW** PSpice.com a user community site to share models, design ideas with other engineers, content rich site to explore
Increase Design Manufacturability & Reliability

Which components are most likely going to fail during manufacturing?

Design might work correctly in the "lab," but can we go to production with it?

Will it operate correctly -
- Over temperature?
- Over manufacturing variations?
- Over its full operating range?
- With age?

Are the individual components being over-stressed?
- Will it fail in test or in the field?

Are there critical sensitivities which will cause problems in the future?
Monte Carlo Analysis using Generic Spice Simulator

- Nominal Value of R1
- Tolerance of R1
- Monte Carlo Notation

Main simulation command. It can be a DC-sweep or ac analysis.

Starting Run

Standard syntax

Increment

End Run

\texttt{.tran 10m startup}
\texttt{.step param run 1 100 1}

\texttt{.step param run 1 100 1}

Max: 11.106mA

Min: 9.094mA

©2017 Cadence Design Systems, Inc.
PSpice Advanced Analyses ..... 

- Helps reduce Cost
- Improves Productivity, Predictability, Reliability and Manufacturing Yield
Sensitivity Analysis

• **Easily identify** components impacting key circuit goals and specs

• **Estimate worst case** performance of the circuit given the device tolerances

• Identify components whose tolerance does not matter
  • **Reduce cost** by choosing components with relaxed tolerance

Setup tolerances
• GLOBAL
• Instance specific
• Using TCL

Set up goal(s) (Measurement expression)

Let the tool identify the most sensitive components in design
Circuit Optimization

• One of the **most powerful** analysis tool

• Identify components impacting key circuit goals

• Identify designs goals and **optimize your design** to meet/beat these goals

• Design goals examples: Gain, BW, Overshoot, Power dissipation, etc..

• Design goals can be described as a waveform or Specification

Setup parameters

• Component values
• Global variables
• Model Parameter

Set up goal(s) (Measurement expression)

Let the Tool Optimize your Design… Go grab a coffee!
What an Analog Optimizer Does

- Engineer specifies the circuit topology and desired goals
- Optimizer does the rest. It calculates optimum component values to use to meet the desired goals
First Principal Application of Optimization

- Optimize the performance of a design
  - Re-Use an Old Design to meet new specifications

Achieve Absolute Maximum Performance Possible from a Given set of Circuit Components
• Retarget an Existing Design for New Specifications
Optimizer - Curve Fit

- Define goal as a waveform
- Identify the key components that can be varied
  Go get a coffee
- Let the Optimizer provide you with the real component values that match your goal
- Impress your manager with your Optimized Design

https://www.youtube.com/watch?v=DGEEEdxo4SQ0
Smoke Analysis

• Checks components to see if they are approaching or exceeding their recommended safe operating limits

• Allows users to specify their own derating criteria to ensure components are not stressed, or exceed safe operating limits

• Determines a part’s performance under various stresses and environmental conditions

• Helps in thermal design aspect of electronics circuit

• Calculate Peak, Avg. and RMS power

• Cross-probe to automatically find listed parts in the schematic
Smoke Analysis has Two Types of Derating

1. User defined deration (Static)
   • Maximum Operating condition * Derate factor
     • Example: Capacitor is rated for 100V and it has derate factor = 0.8; Safe operating limit would be 80V

2. Temperature-derated safe operating limit (Dynamic)
   • Certain devices need to be derated for various operating conditions like Temp, Voltage…
   • Power rating of resistance is derated for temperature
   • Power rating of semiconductor device is derated for temperature
   • Rated voltage of capacitor is derated for temperature
Selecting appropriate heat sink is key aspect of any power circuit design

PSpiceAA Smoke analysis allows you to simulate heat sink

Associate appropriate heat sink with semiconductor device

Smoke will calculate junction temperature for that

Define Derate specification

Associate Derate specification with component

Select desired Derate specification

Run SMOKE analysis
Semiconductor Thermal Impedance Network

Modify this parameter as per heat sink data sheet to simulate effect of particular heat sink.

This parameter can be modified on schematic instance, or as part of Model.

Device specific parameter
Controlled by Mounting pattern
Heat sink specification

Combined as RCA

## Devices Covered by SMOKE Analysis

<table>
<thead>
<tr>
<th>RESISTOR</th>
<th>CAPACITOR</th>
<th>INDUCTOR</th>
<th>BJT</th>
<th>MOSFET</th>
</tr>
</thead>
<tbody>
<tr>
<td>SCR</td>
<td>DIODE</td>
<td>DIODE BRIDGE</td>
<td>ZENER</td>
<td>Dual MOS</td>
</tr>
<tr>
<td>JFET</td>
<td>MESFET</td>
<td>OPTOCOUGLER</td>
<td>SWITCH</td>
<td>TRANSFORMER</td>
</tr>
<tr>
<td>OP-AMP</td>
<td>OptoCoupler</td>
<td>Varistor</td>
<td>LED</td>
<td>Linear Voltage Regulator</td>
</tr>
</tbody>
</table>
Parametric Sweep

- **Sweep** multiple (nested) parameters
- Quickly view results and create families of curves
- **Ensure** there is no unusual circuit behavior while sweeping the component values
Monte Carlo to Predict Overall Yield

- Calculate Yield before going into manufacturing
- Produce circuit performance statistics due to device variations
- Set specification minimum and maximum, and estimate production yield before going to production
- View graphical results as probability density histogram, or as cumulative distribution function
PSpice AA Enhancements – 17.2 QIR 2

**Customer Requests**

- Be able to run PSpice® Advance Analysis on existing designs or designs created using SPICE models
- Update/recapture design (partially or fully) using AA enabled components supplied with PSpice installation

**Enhancements**

- Run PSpice Advance Analysis on existing designs without updating any parts/models
- Apply tolerances and distributions to any design using SPICE component(s)
- Download any vendor model(s) and add tolerance and distributions for device parameters
- Tolerances can be added directly to components or through the new Add Tolerance form

Go beyond a traditional SPICE analysis and use PSpice Advanced Analysis tools to fine tune designs, determine yield, perform stress analysis and optimize designs to meet new requirements
Simple 5 Step Process to Improve Reliability

1. Take existing SPICE circuit/designs
2. Assign GLOBAL or Instance specific tolerance
3. Setup goal(s)
4. Run PSpice Advance Analyses
5. Identify any components that may fail in the field

Improve Reliability
Traditional SPICE simulator would let you run:

- Monte-Carlo analysis on one goal
- Supports limited Goal expressions
- DC Sensitivity analysis only
- WorstCase Analysis
  - Only one analysis at a time
  - Only one goal
  - Only one Temperature

PSpice Advance Analysis enables you:

- Monte-Carlo analysis on any number of goals
- All types of Goals supported
- All Analysis types
- WorstCase Analysis
  - Evaluate multiple analysis in one run
  - Multiple goals
  - Evaluate your circuit on multiple Temperatures
Quickly Analyze Results in AA

- Sort columns
- Filter based on reference designator
- View tolerances or values of components
- View Absolute or Relative sensitivity
- View results on Log or Linear scale
- Cross-Probe “critical” component in schematic
Globally Assign Tolerance to Discrete/sources and Run Monte-Carlo, Worst Case analysis

Take any design

Assign Global Tolerance

Run sensitivity analysis to optimize cost and performance

Calculate yield
Take model from Web and use in design for Advance Analyses

Download model from Web

Set your design goal

Assign tolerances to model parameter

Analyze the design
Assign tolerance to External (Subcircuit) Model Parameters easily

Take a model from other source

Assign tolerance to “exposed” model parameters

Analyze the design

No need to modify original models
PSpice AA allows the ability to assign tolerances on:

- Device/model parameters
- Global variables
- Voltage and Current sources
- Sub circuit parameter

Models and designs downloaded from web can be readily used in Advanced Analysis
• PSpice – Most accurate SPICE simulator for mixed signal, SPICE based, circuit simulation
• Comprehensive ecosystem – Most IC vendors deliver PSpice simulation models
• Enables customers improve their Productivity, Reliability, Predictability, and Cost
• Combine PSpice with enhanced Advanced Analyses and we have an unmatched solution
• Significant Performance, Convergence and Model Selection improvements in the past year

Summary – Check out [http://pspice.com](http://pspice.com)