Australia
Austria
Belgium
Brazil
Bulgaria
Canada
China
Czech Republic
Denmark
Estonia
Finland
France

Germany
Hungary
India
Indonesia
Ireland
Israel
Italy
Japan
South Korea
Latvia
Lithuania

Malaysia
Mexico
Netherlands
New Zealand
Norway
Poland
Philippines
Romania
Russia
Serbia
Singapore

Slovenia
South Africa
Spain
Sweden
Switzerland
Taiwan
Thailand
Turkey
Ukraine
United Kingdom
USA
Việt Nam

ANALOG DEVICES

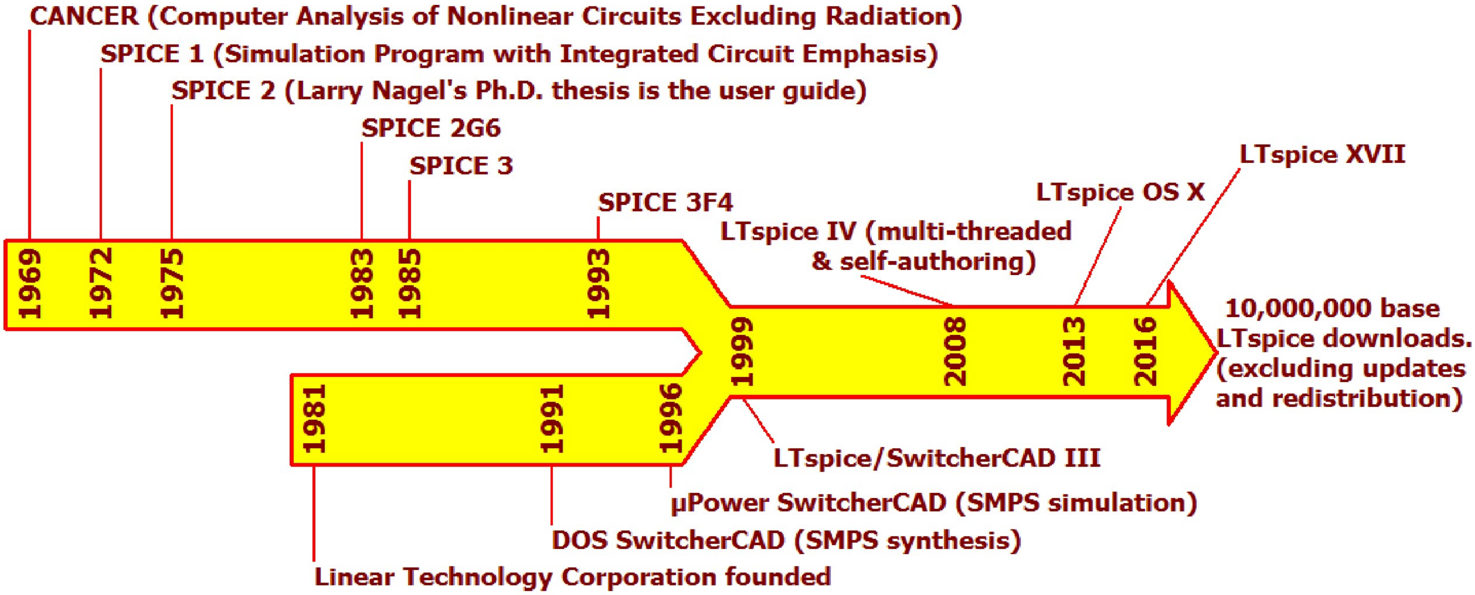
The Best Simulators are Developed by the Concerns that Need them.

- Best charged-particle optic simulator: scanning electron microscope company
- Best MOSFET circuit solver: Intel
- Best radar return solver: Government Intelligence
- Best analog circuit solver: IC company, Linear Technology Corporation

Software companies can't compete because it isn't possible to recoup the NRE with licensing fees. For example, PSpice grosses a few million dollars a year, but LTspice is used in the design and sale of about a billion dollars worth of IC's.

ANALOG DEVICES

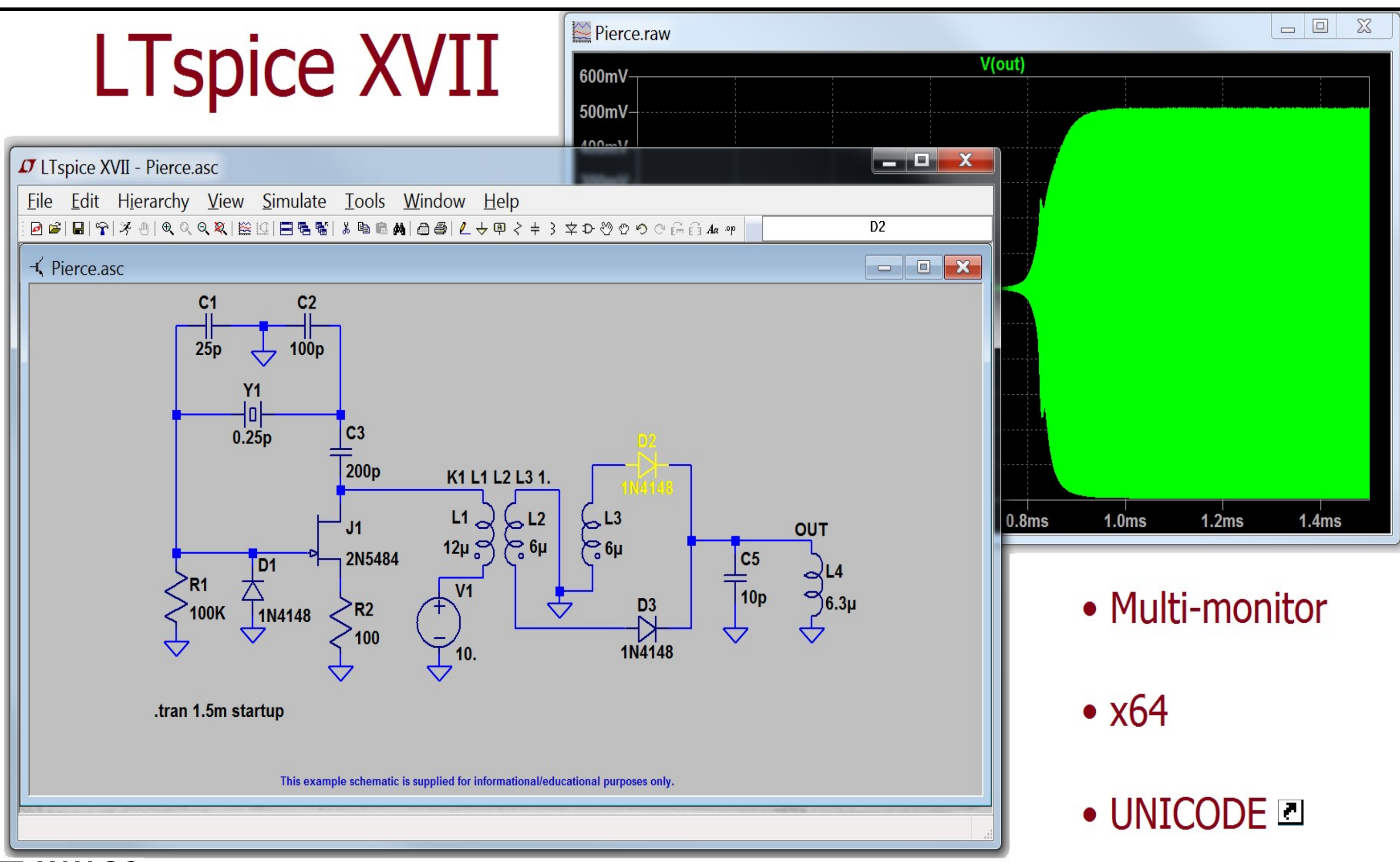
Time Line of SPICE



- LTspice has been the industry de facto standard since mid 2000's
- Downloaded 4 times per minute(excluding updates)
- Distributed 100's of times more than any other SPICE program

ANALOG DEVICES

LTspice XVII



- Multi-monitor
- x64
- UNICODE

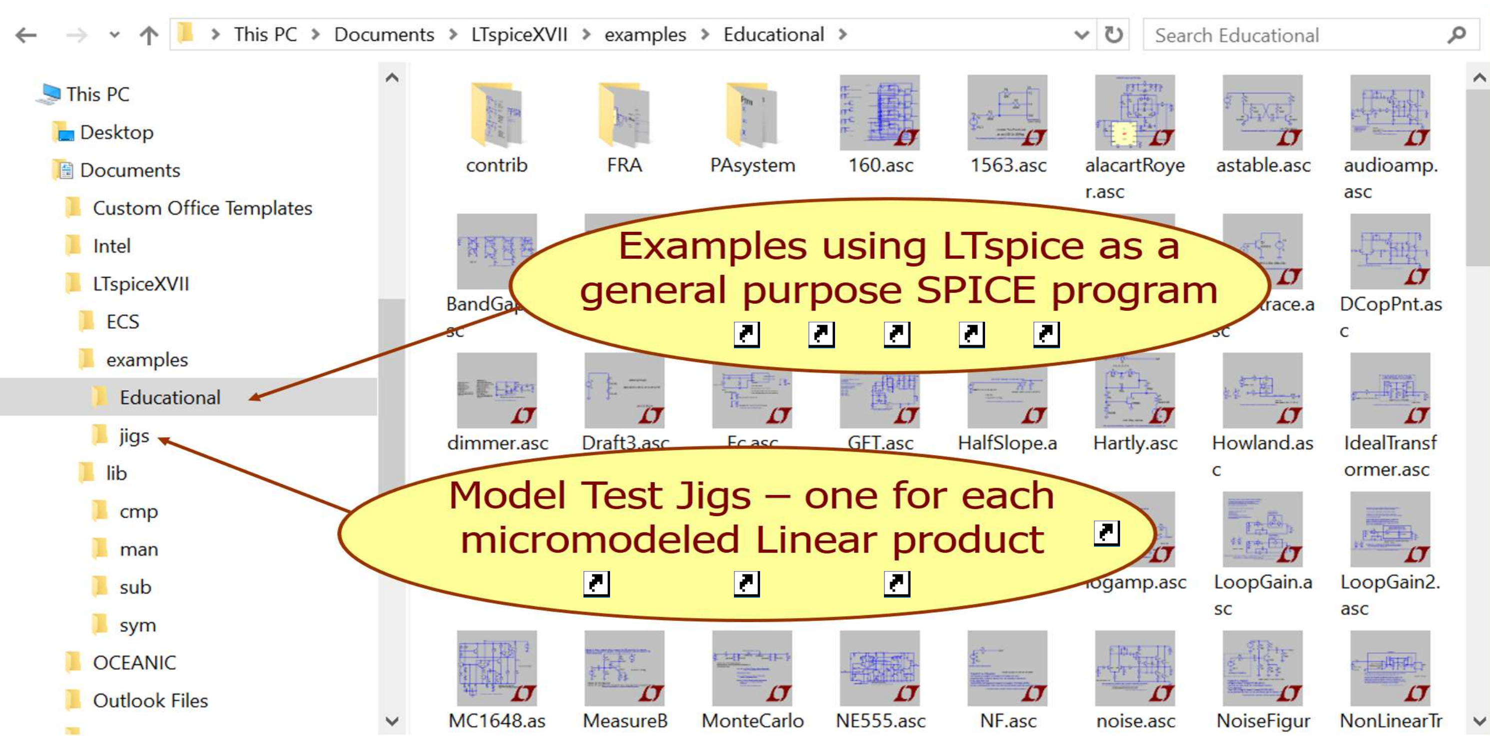
ANALOG DEVICES

Drafting Your Own Circuits

- General Purpose Schematic Capture
  - Unlimited schematic size
  - Unlimited depth of hierarchy
  - Bidirectional cross-probing
  - Graphical Symbol editor
  - Complete documentation
- Macromodels of 2500 Analog Devices Products
- Integration with Industry Superlative SPICE Simulator
  - Unlimited, professional SPICE proven for IC design
  - Unmatched combination of robustness, accuracy, speed and compatibility
  - Advanced analysis/simulation options, parameter sweeps, FFTs, etc.
  - Use 3rd party models
  - Active independent users' group

ANALOG DEVICES

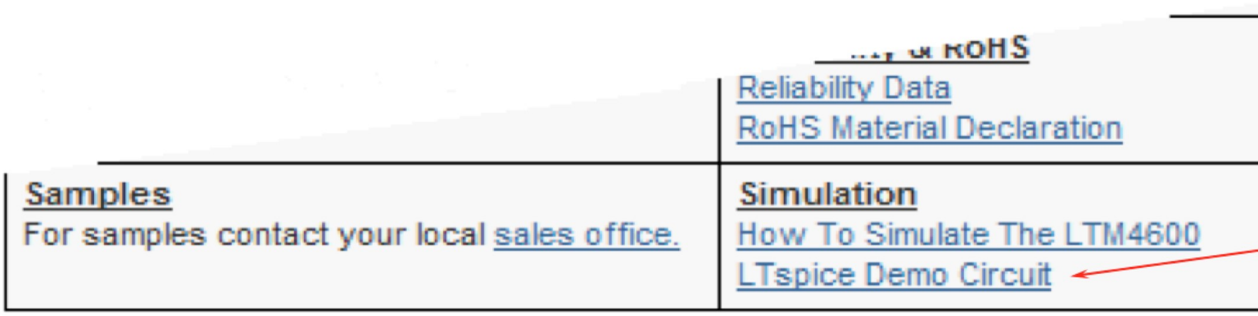
Installation Directory Structure



ANALOG DEVICES

Additional Resources for Example Files

- http://www.analog.com
  - Some products feature an example LTspice schematic



- http://groups.io/g/LTspice (formerly http://groups.yahoo.com/group/LTspice)
- FAE's at Analog
- LTspice@analog.com

ANALOG DEVICES

Waveform Display Features

- Plot expressions of data assisted with cross probing
  - Cross probe voltages, device and port currents
  - Differential crossprobing
  - Dissipation expression composed by the schematic
  - Current in a "wire"
  - Dimensional analysis
  - Horizontal panning with the mouse tilt
- Waveform average and RMS calculator
- Fourier analysis (both .four statements and FFT's)
- Dynamic waveform data compression
- Multiple plot planes
  - Attached cursors ganged across plot panes
- Eye diagrams
- Complex data: Bode, Nyquist, and Cartesian
- Parametric plotting (X-Y plotting)

ANALOG DEVICES

What Usually Is Modeled?

- Typical performance at room temperature
- Error amp
  - Gm
  - Source/Sink Current
- Oscillator
  - Frequency
  - Duty Cycle Limits
- Switch logic
- Switch current limit
- Switch beta
- Peak current vs error voltage
- Slope compensation
- Burst Mode
- Switch minimum on time
- Pulse skipping
- PLL capture & phase lock

ANALOG DEVICES

What Usually Is Not Modeled?

- Production scatter
- Behavior over temperature
- Catastrophic failure modes
- Strategic simplification: Oscillator SYNC pin(unless the device has a PLL)
- Tactical simplifications mentioned on the symbol

ANALOG DEVICES

What Can't Be Counted on to Be Modeled or Not?

- Iq in all modes
- Misc features in shutdown

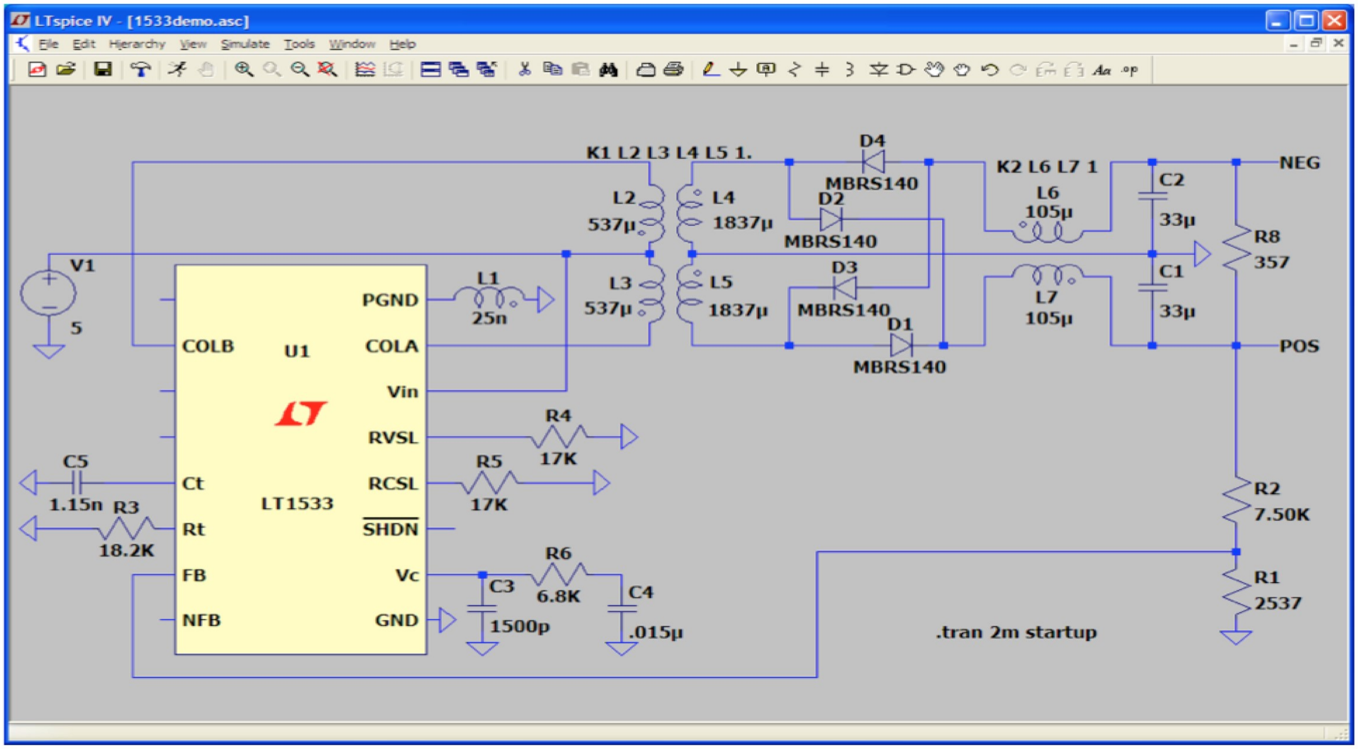
ANALOG DEVICES

LT1533 Case Study

- Voltage and current slew rate limiting
- Complex control logic
- External timing capacitor
- Timing-cap current look-up table
- Demo board available
- Difficult to get efficiency analytically

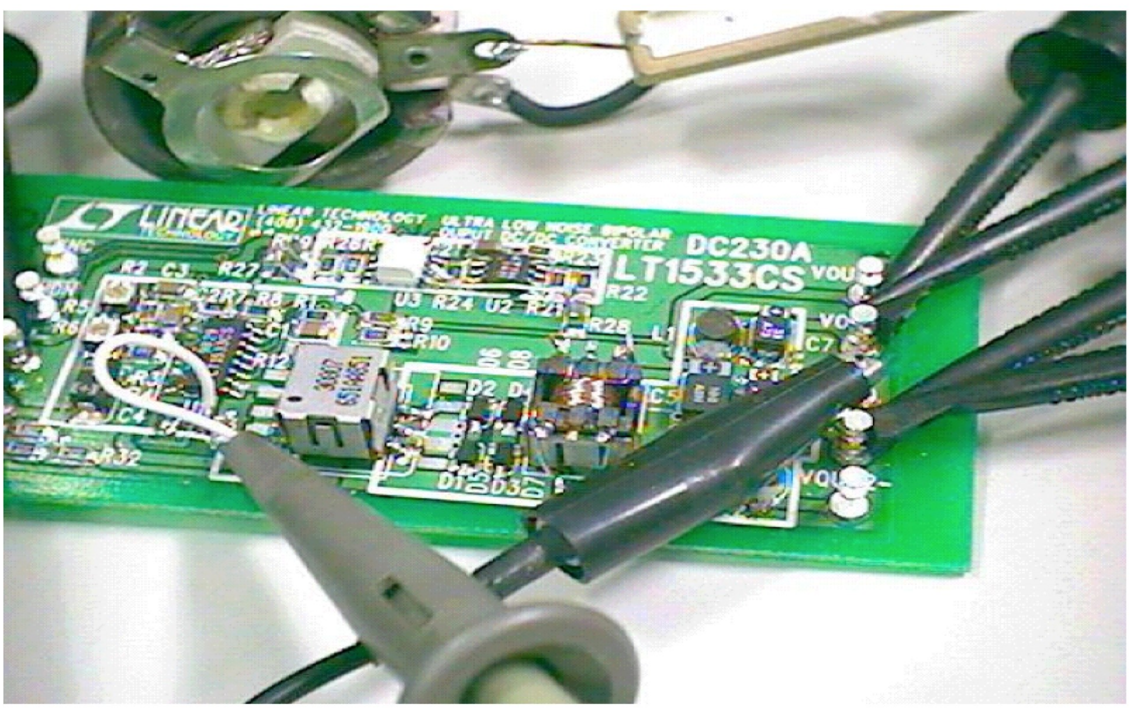
ANALOG DEVICES

Demo Board Schematic

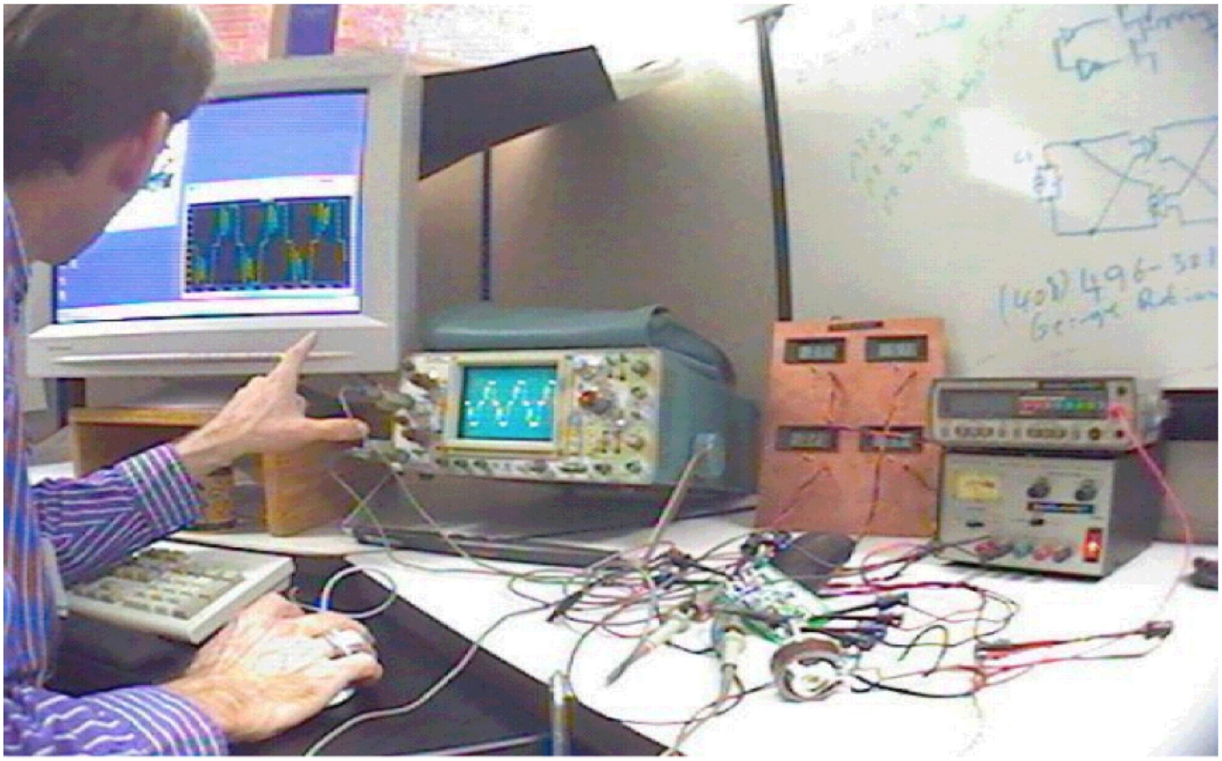


ANALOG DEVICES

Demo Board

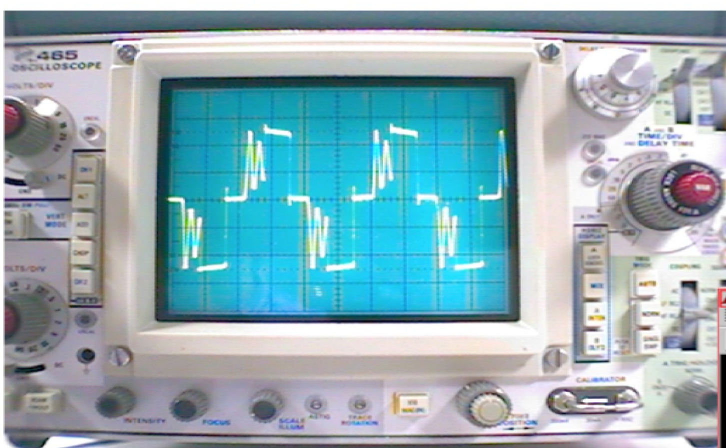


ANALOG DEVICES

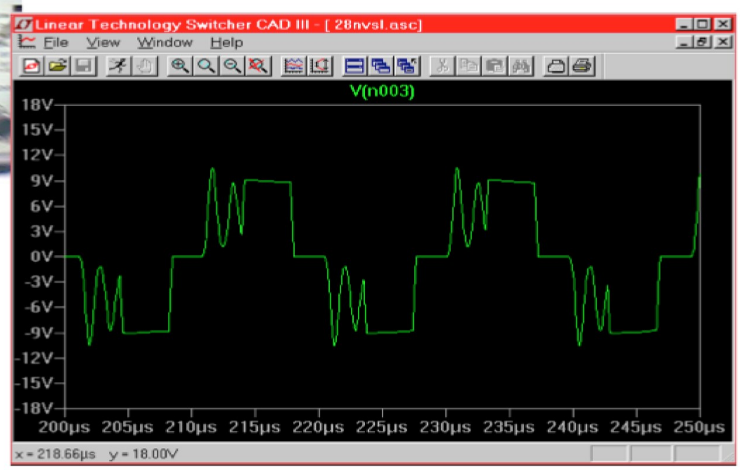


ANALOG DEVICES

Minimum Slew Limits

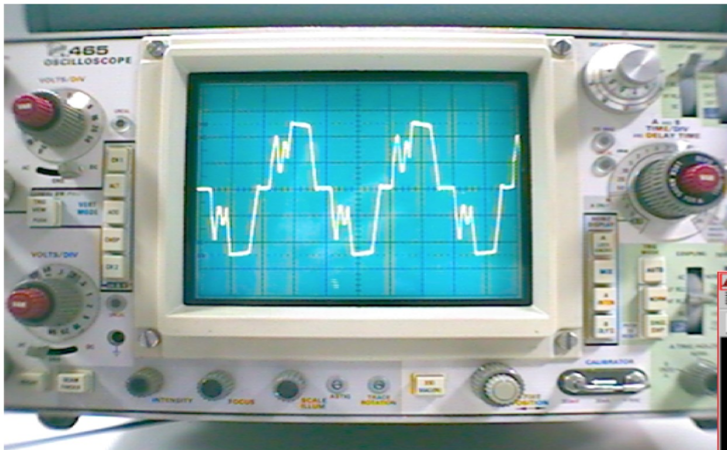


Vin = 5V
Vout = +/-5V
Iout = 28mA

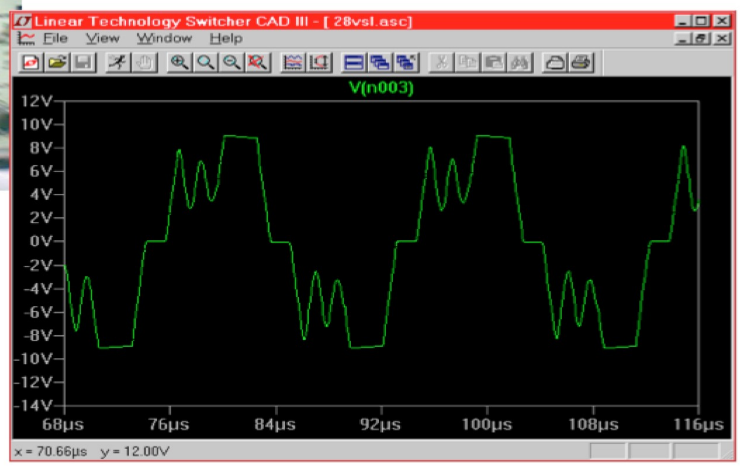


ANALOG DEVICES

With Voltage Slew Limit



Vin = 5V
Vout = ±5V
Iout = 28mA



ANALOG DEVICES

LT1533 Efficiency Comparison

	LTspice	Demo Board
Min. Slew Rate Limit	73.0%	73.0%
With Current Slew Limit	66.0%	65.4%
With Voltage Slew Limit	63.0%	62.0%

ANALOG DEVICES

SMPS Stability
or
Open Loop Response From the Closed Loop System

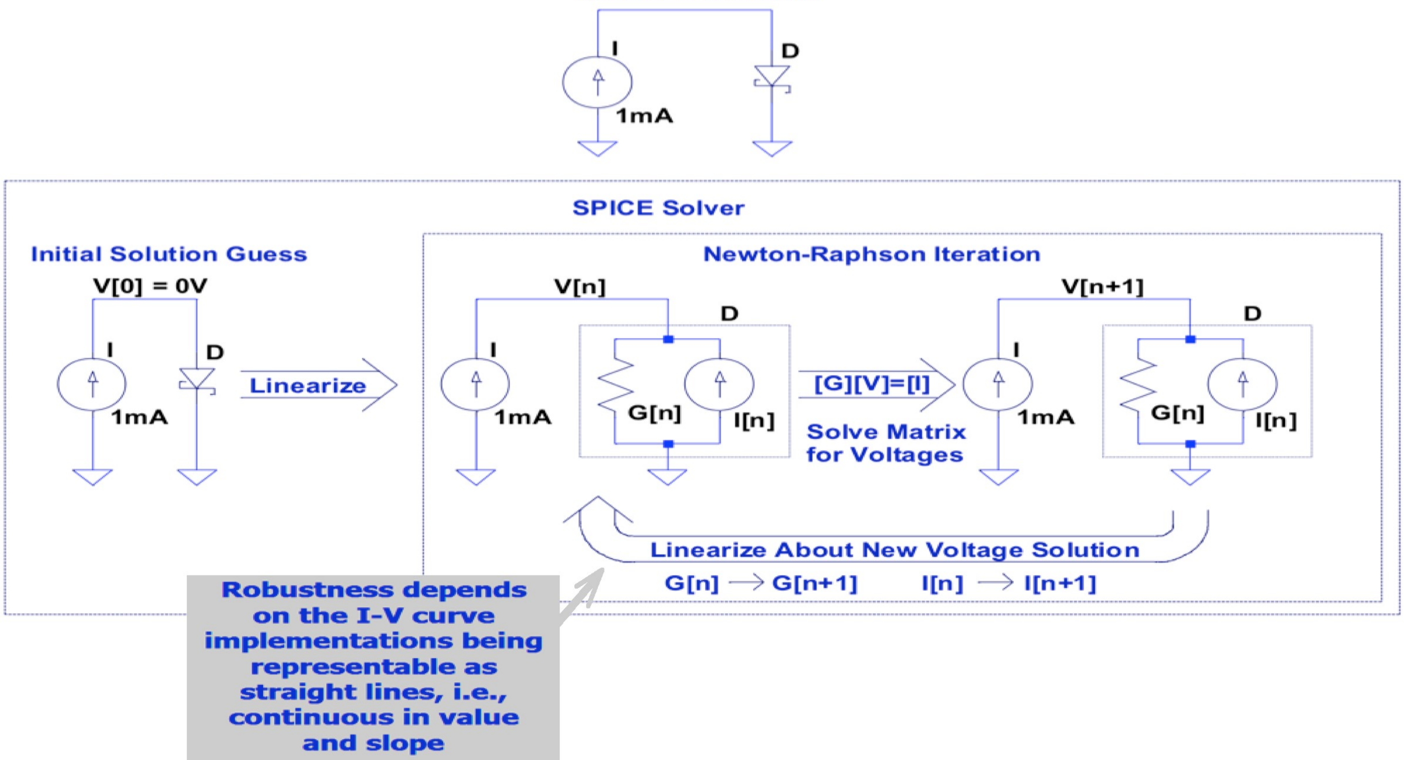
ANALOG DEVICES

Three Numerical Methods Account for the Success of SPICE

- Newton iteration
- Sparse matrix methods
- Implicit integration

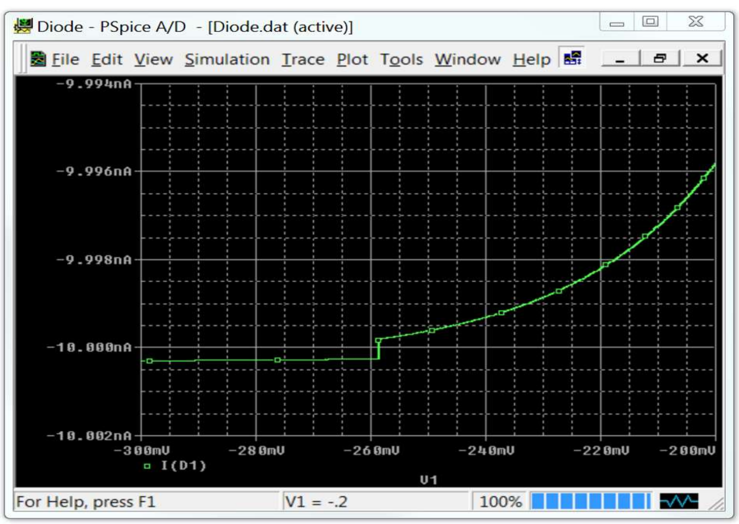
ANALOG DEVICES

Newton Iteration

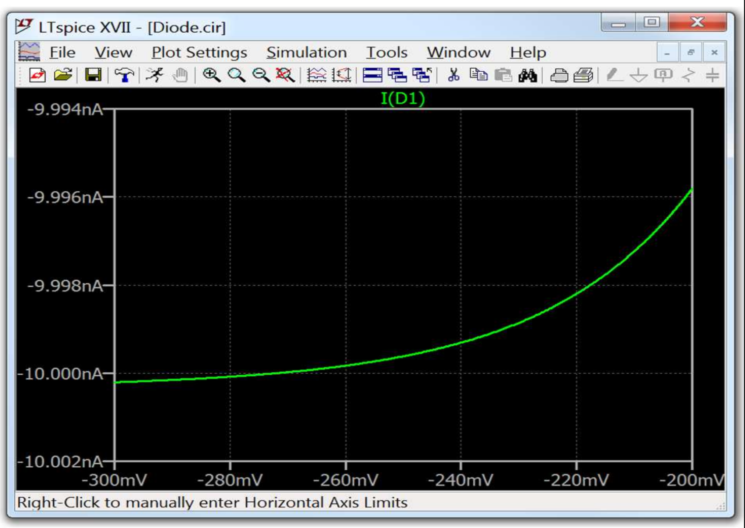


ANALOG DEVICES

Berkeley Diode Discontinuity



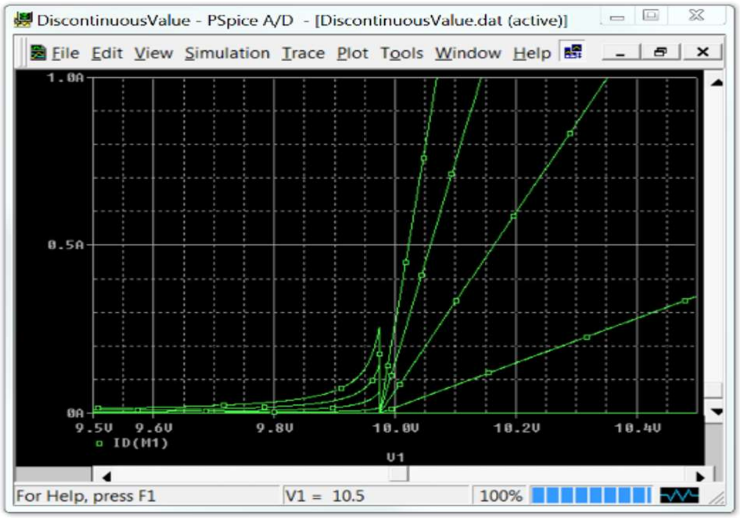
PSpice

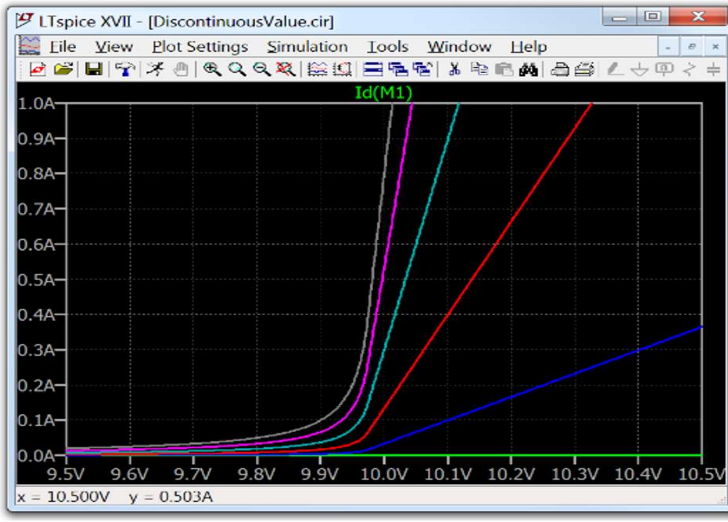


LTspice

ANALOG DEVICES

Berkeley MOS Discontinuous Value



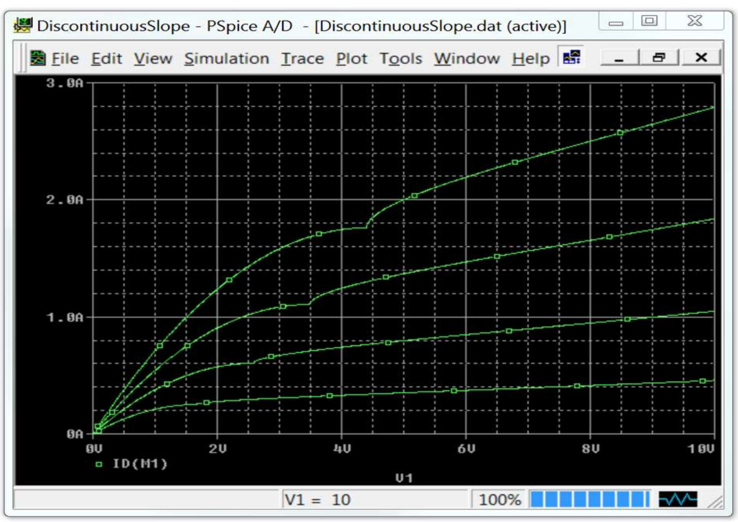


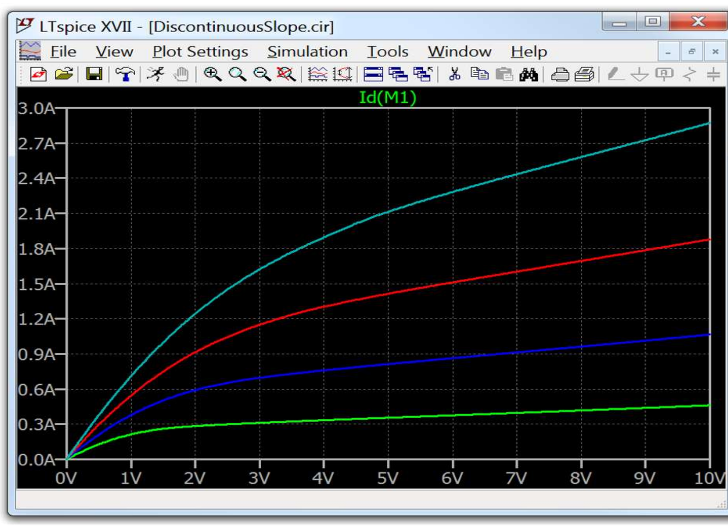
PSpice

LTspice

ANALOG DEVICES

Berkeley MOS Discontinuous Slope



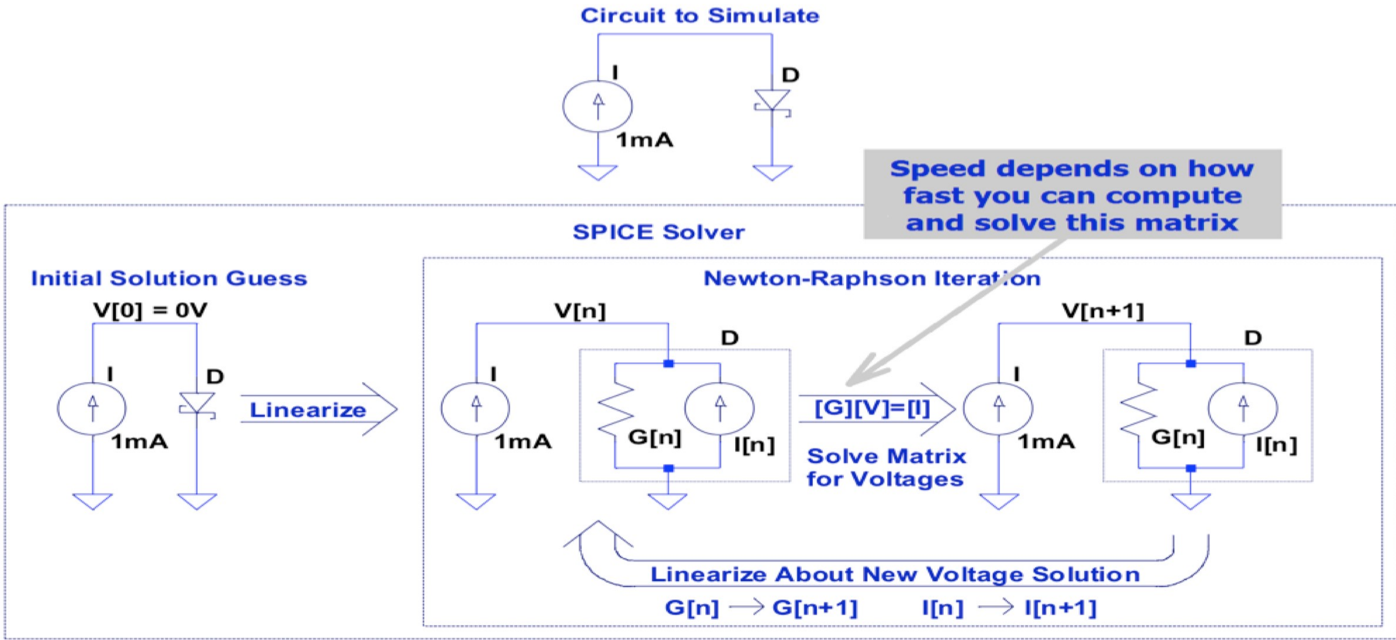


PSpice

LTspice

ANALOG DEVICES

Newton Iteration



ANALOG DEVICES

Simulation Speed Determined by

1. Time to Compute the Matrix
2. Time to Solve the Matrix

ANALOG DEVICES

Simulation Speed Determined by

O(N): **"Compute the matrix" means computing these coefficients**

LTspice's multi-threaded solver computes these coefficients in parallel.

$$\begin{bmatrix} G11 & G12 & \cdots & G1n \\ G21 & G22 & & \\ \vdots & & \ddots & \\ Gn1 & & & Gnn \end{bmatrix} \cdot \begin{bmatrix} V1 \\ V2 \\ \vdots \\ Vn \end{bmatrix} = \begin{bmatrix} I1 \\ I2 \\ \vdots \\ In \end{bmatrix}$$

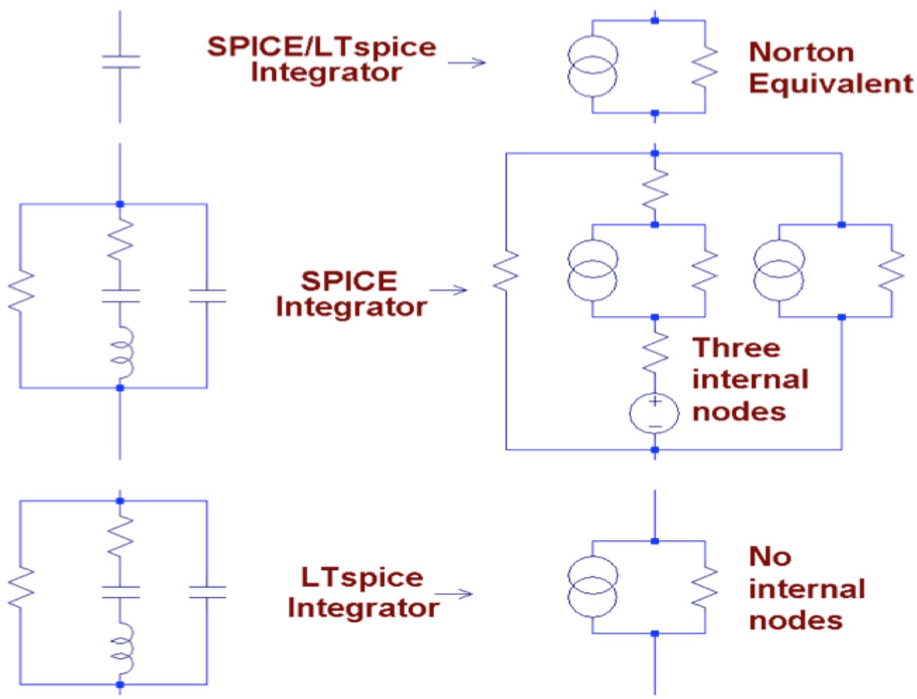
O(N^2): **"Solve the matrix" means solving for this vector**

LTspice's self-authoring solver allows the matrix be solved at the theoretical FLOP limit

So use smaller matrices!

ANALOG DEVICES

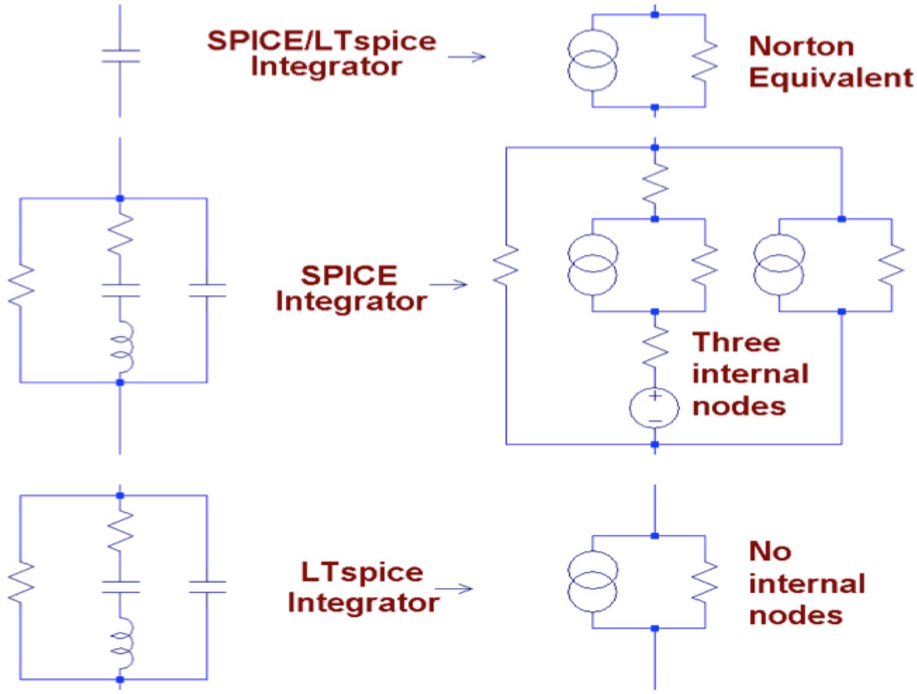
Matrix Node Reduction



But what happens to the accuracy?

ANALOG DEVICES

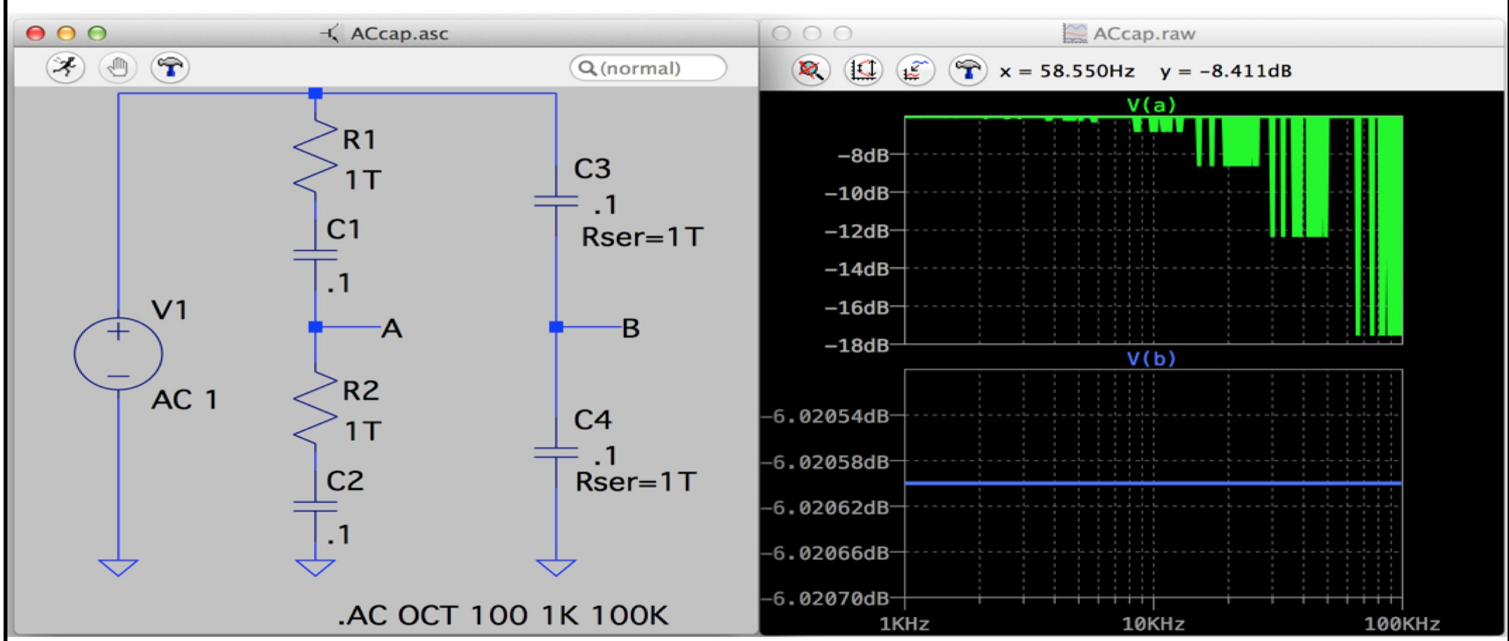
Matrix Node Reduction



Node reduction improves accuracy!

ANALOG DEVICES

Node Reduction Example



Node reduction improves both speed and accuracy

ANALOG DEVICES

Implicit Integration

- Circuit reactances give rise to differential equations that are numerically integrated.
- Because the solution looks like exp(-const*time), if a normal method; e.g., Euler is used; the error would exponentially deviate from the correct solution and diverge to infinity.
- Implicit methods; e.g., backward Euler; are used in circuit simulation to avoid the singularity. Unfortunately backward Euler is extremely slow and inaccurate, so 2nd order methods are used.

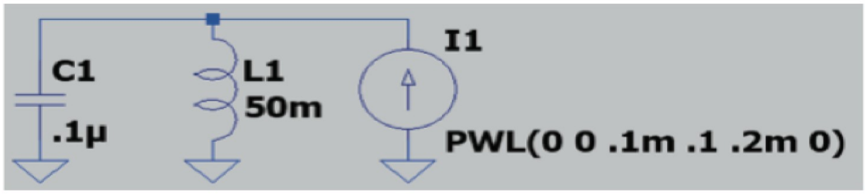
ANALOG DEVICES

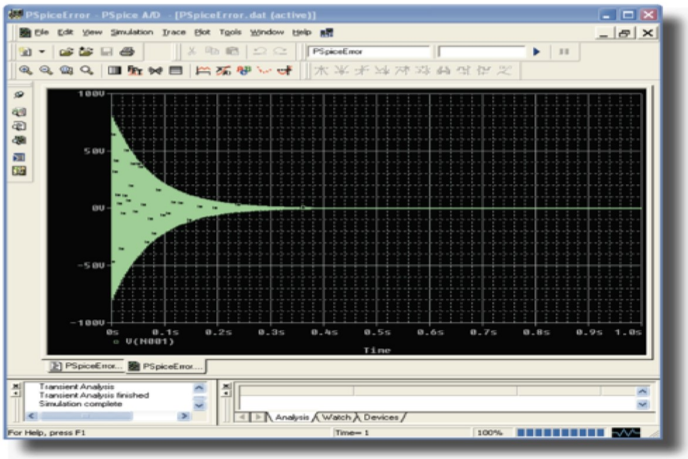
2nd Order Implicit Integration Methods

- Trapezoidal(trap)
  - Potential for Ringing artifact
  - Fast
  - Accurate
- Gear
  - No Ringing artifact
  - Slow
  - Inaccurate
- Modified Trap(proprietary to LTspice)
  - No Ringing artifact
  - Fast
  - Most accurate method known

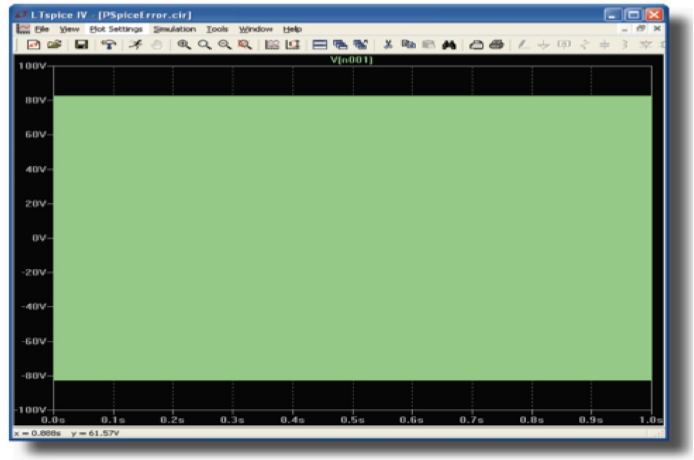
ANALOG DEVICES

Gear Integration Error





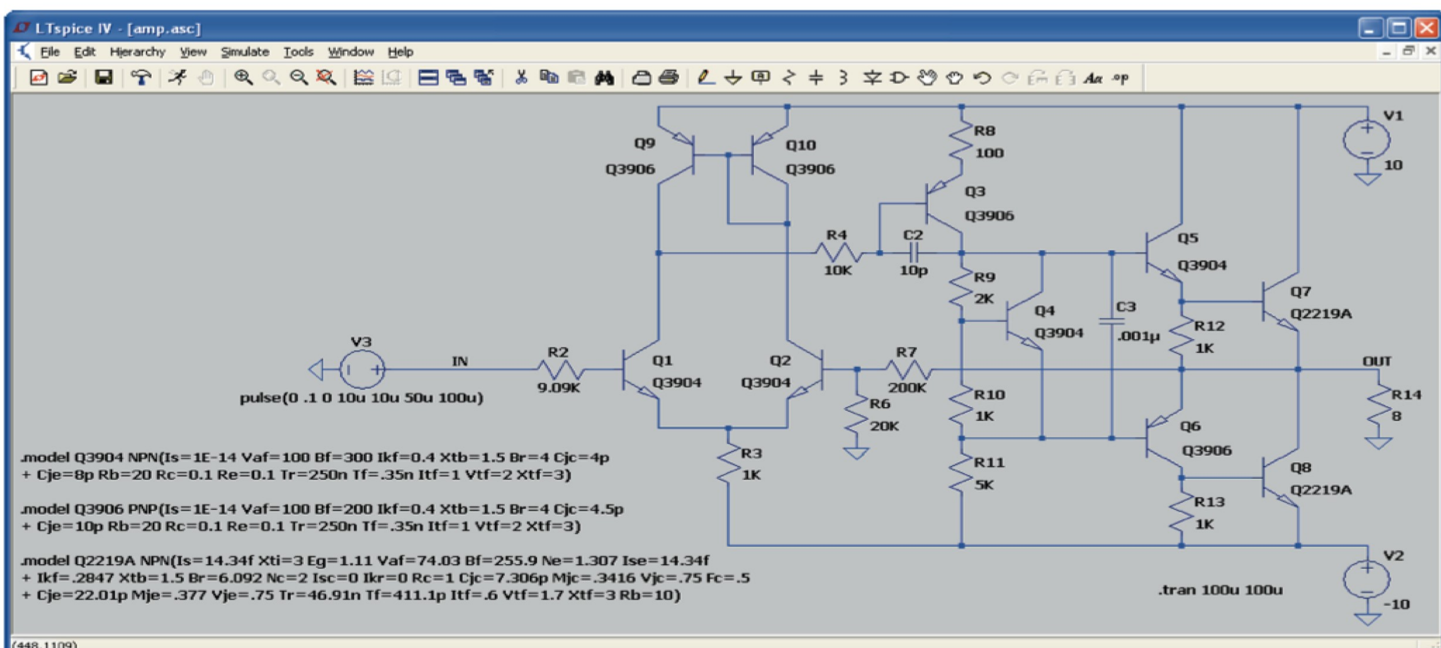
PSpice



LTspice

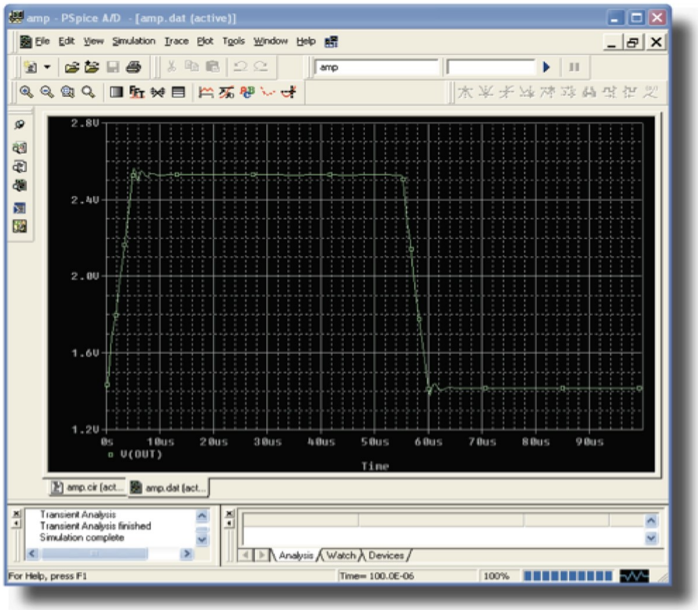
ANALOG DEVICES

Gear Integration Error In Context of a Practical Example

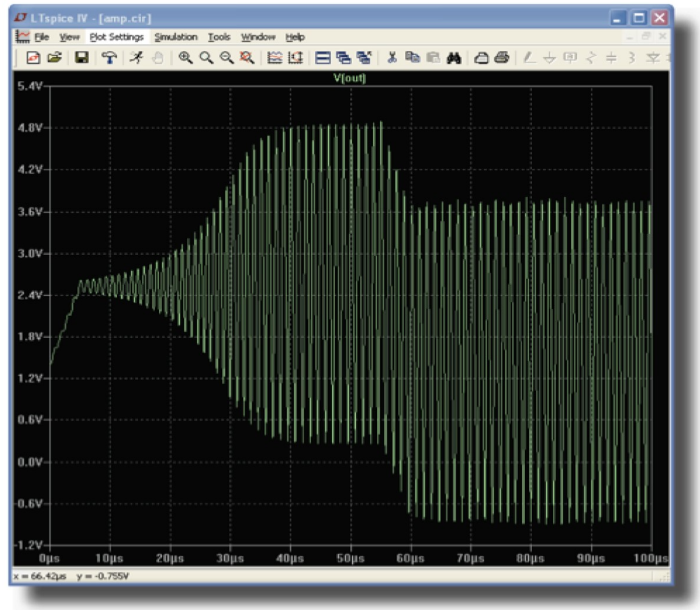


ANALOG DEVICES

Gear Integration Error In Context of a Practical Example



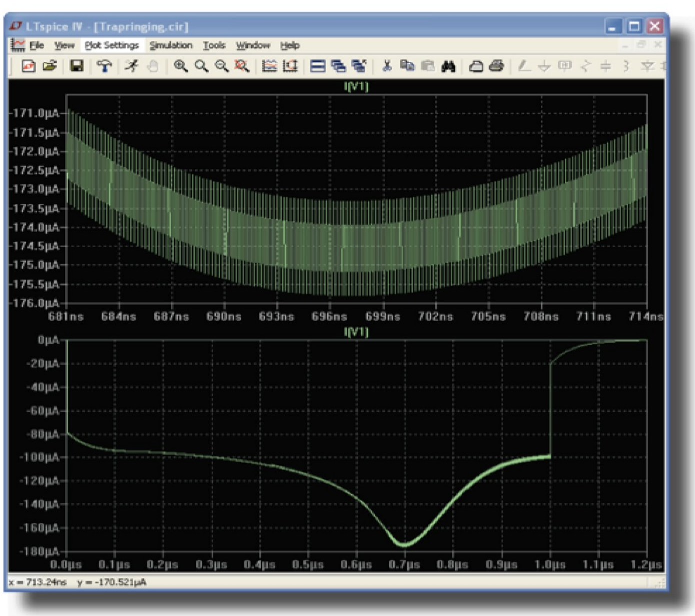
PSpice



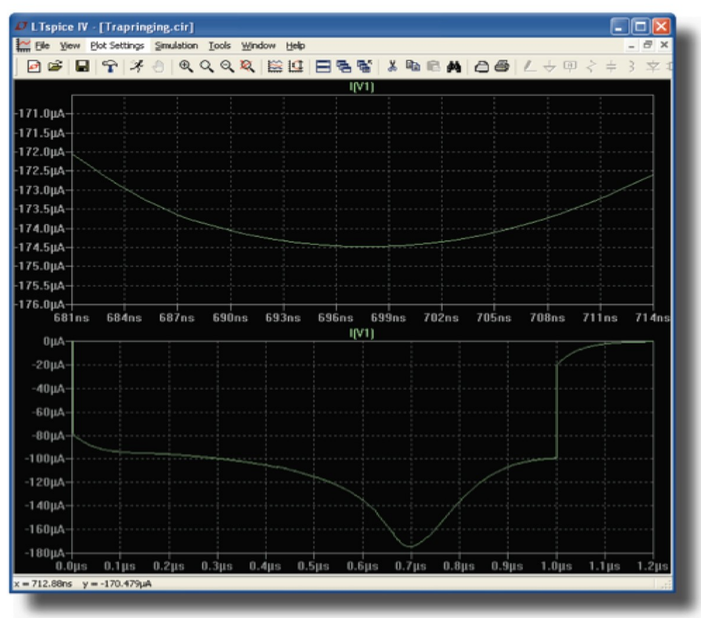
LTspice

ANALOG DEVICES

Trap Integration Artifact



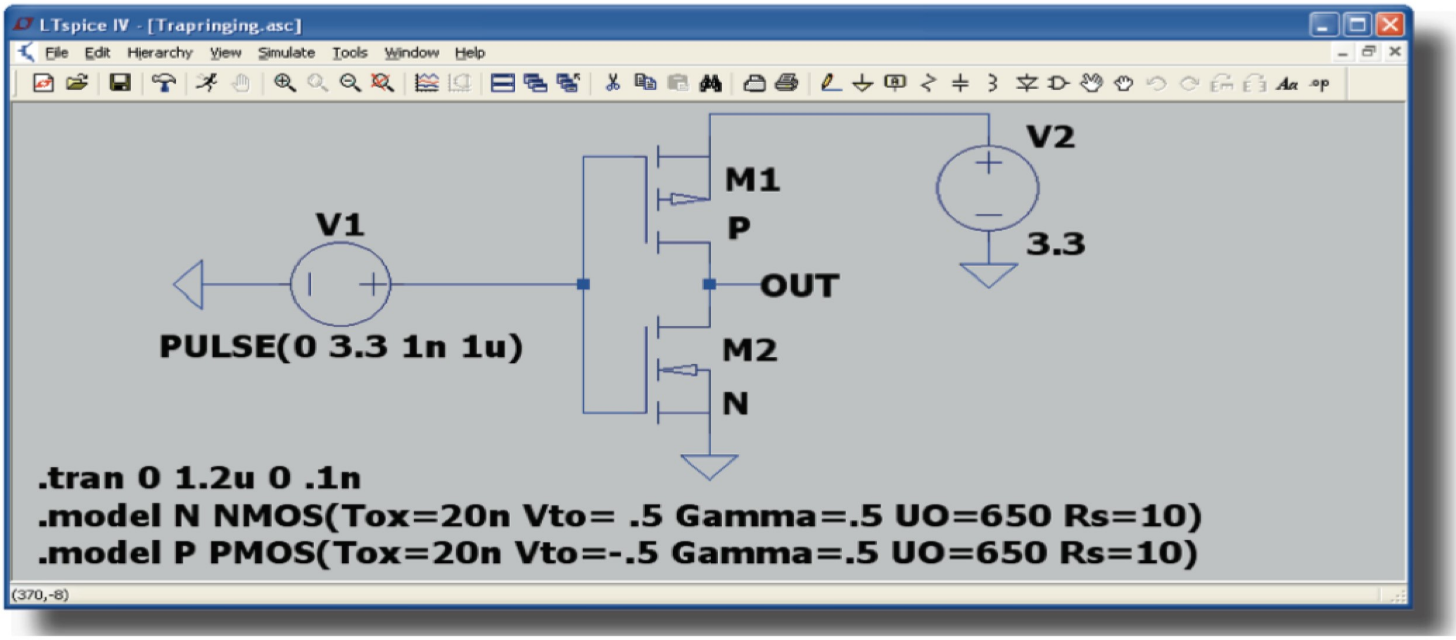
Trap



Modified Trap

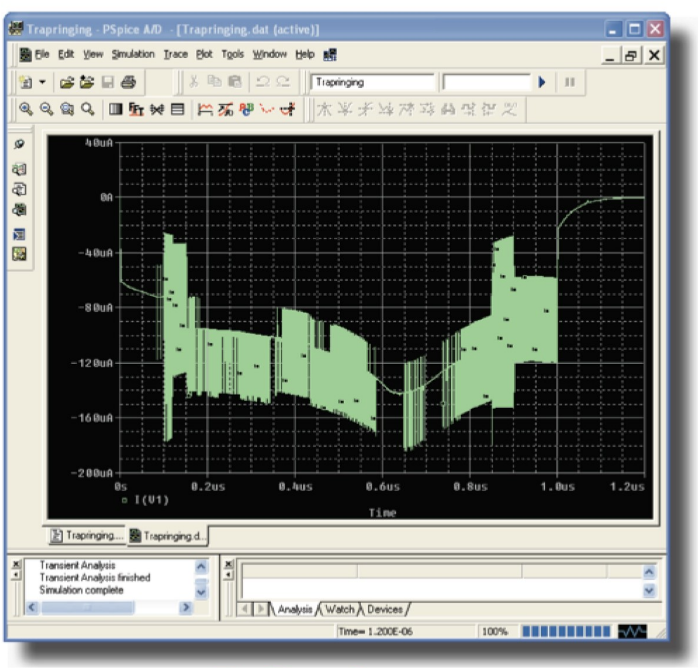
ANALOG DEVICES

Trap Integration Artifact Circuit

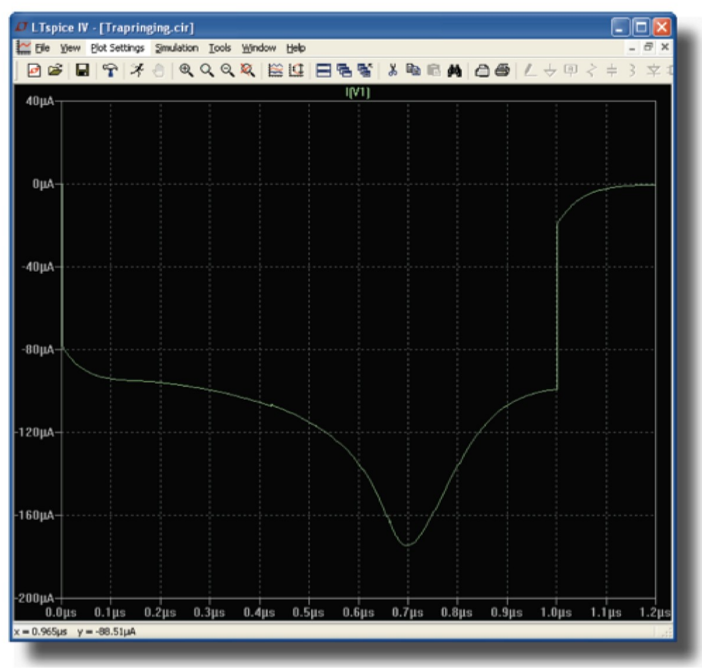


ANALOG DEVICES

Trap Integration Artifact Circuit



PSpice



LTspice

ANALOG DEVICES

Three Numerical Methods Account for the Success of SPICE

- Newton iteration
  - Need I-V curves continuous in value and slope

Robustness

- Sparse matrix methods
  - Node reduction for speed and accuracy
  - Compute matrix coefficients in parallel threads
  - Solve matrix with self-authoring code

Speed

- Implicit integration
  - Proprietary modified trap
  - speed and accuracy of trap
  - No trap ringing

Integrity

ANALOG DEVICES

LTspice was not the first SPICE implementation, nor is it the only free SPICE, but it is the best and most widely used SPICE implementation.

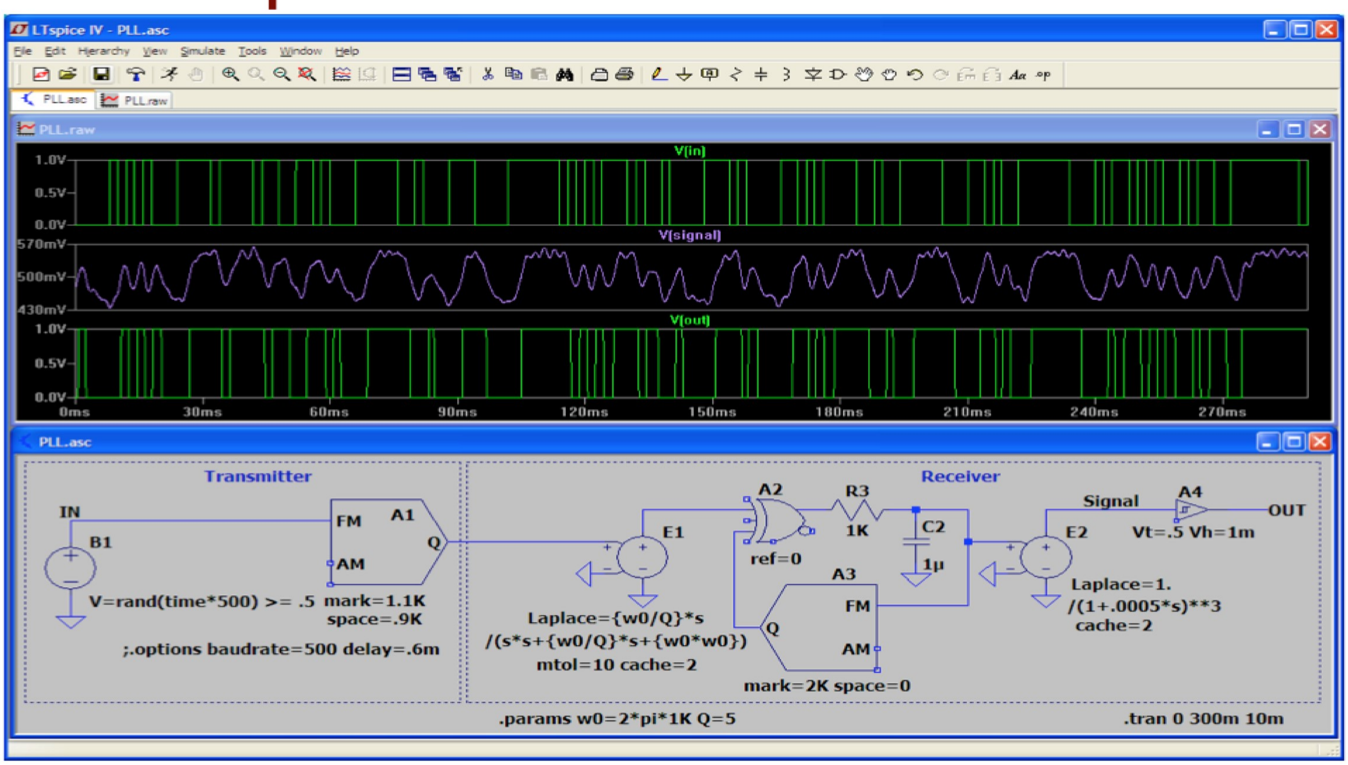
ANALOG DEVICES

LTspice Behavioral Simulator

- PSPICE style behavioral modeling
  - Legacy POLY() statements
  - Arbitrary expressions
  - Laplace
  - Look-up tables.
- Arbitrary capacitance: write an expression for the charge.
- Arbitrary inductor: write an expression for the flux.
- An original mixed-mode simulator -- not xspice based.
- Co-simulation for very complex models

ANALOG DEVICES

Example Mixed-Mode Simulation



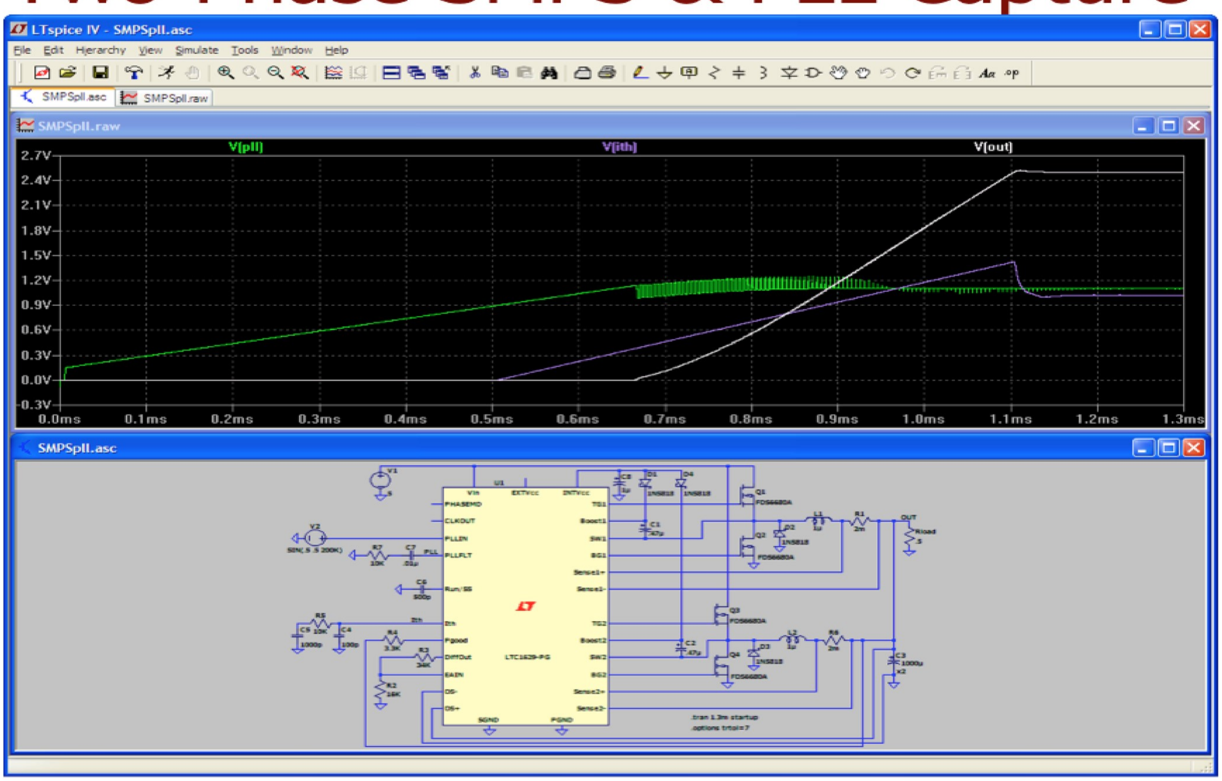
ANALOG DEVICES

Mixed-Mode Simulator

- Computationally lightweight
- Tight feedback between analog and digital circuitry
  - Implemented as a mix of intrinsic SPICE devices and ~30 optimizing HDL compilers.
  - Predictors aid timestep control.
- Easy to program so that models for new products are usually quick to be generated.

ANALOG DEVICES

Two-Phase SMPS & PLL Capture



Total elapsed time: 5.508 seconds.

ANALOG DEVICES

Chan et al. Nonlinear Magnetics

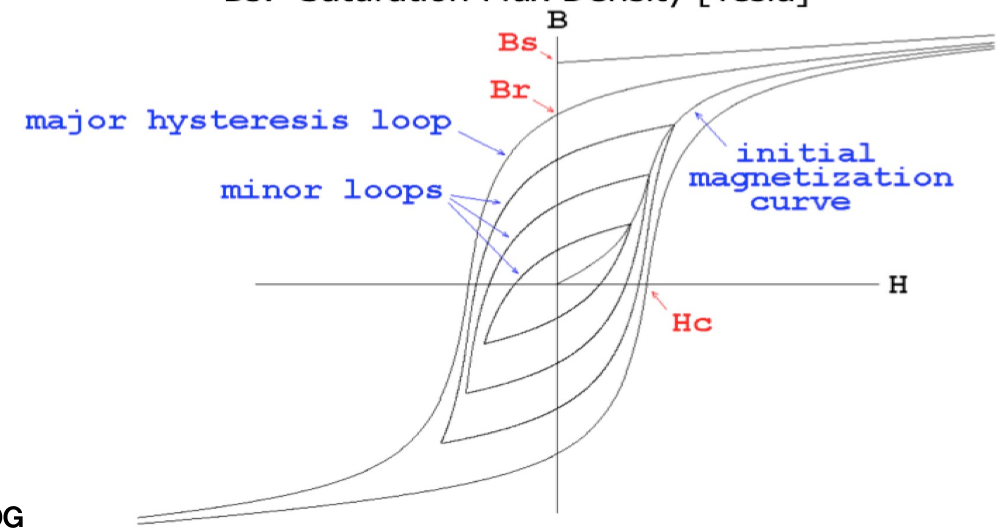
Extended per US Patent 7,502,723

A computationally lightweight model that uses only three parameters to specify the core's major hysteresis loop:

Hc: Coercive force [Amp-turns/meter]

Br: Remnant Flux Density [Tesla]

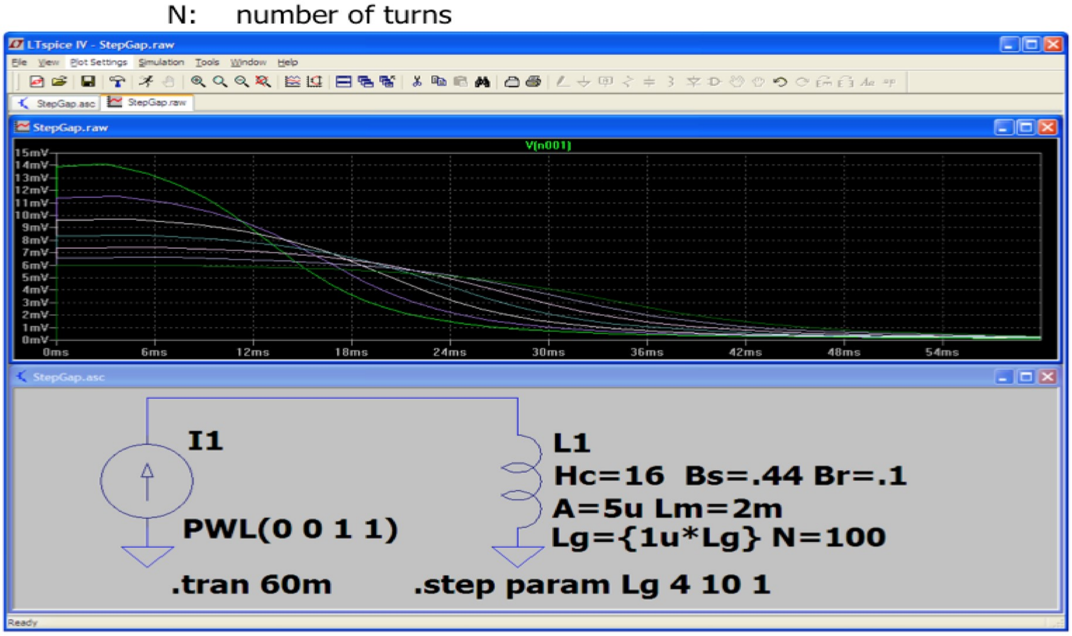
Bs: Saturation Flux Density [Tesla]



ANALOG DEVICES

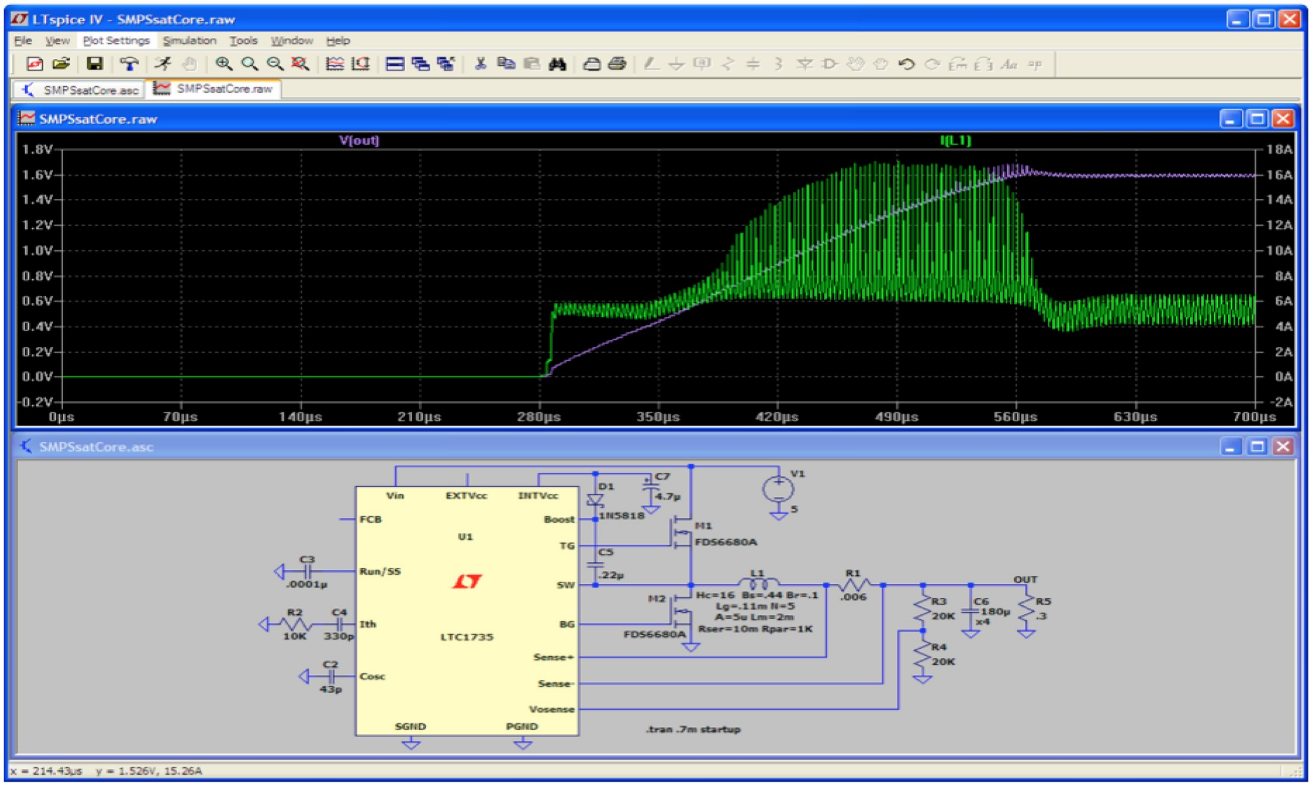
Gapped Core Magnetic Solver

- Core physical dimensions specified with four parameters:
  - Lm: Magnetic Length(excl. gap)[meter]
  - Lg : Length of gap [meter]
  - A: cross sectional area [meter**2]
  - N: number of turns



ANALOG DEVICES

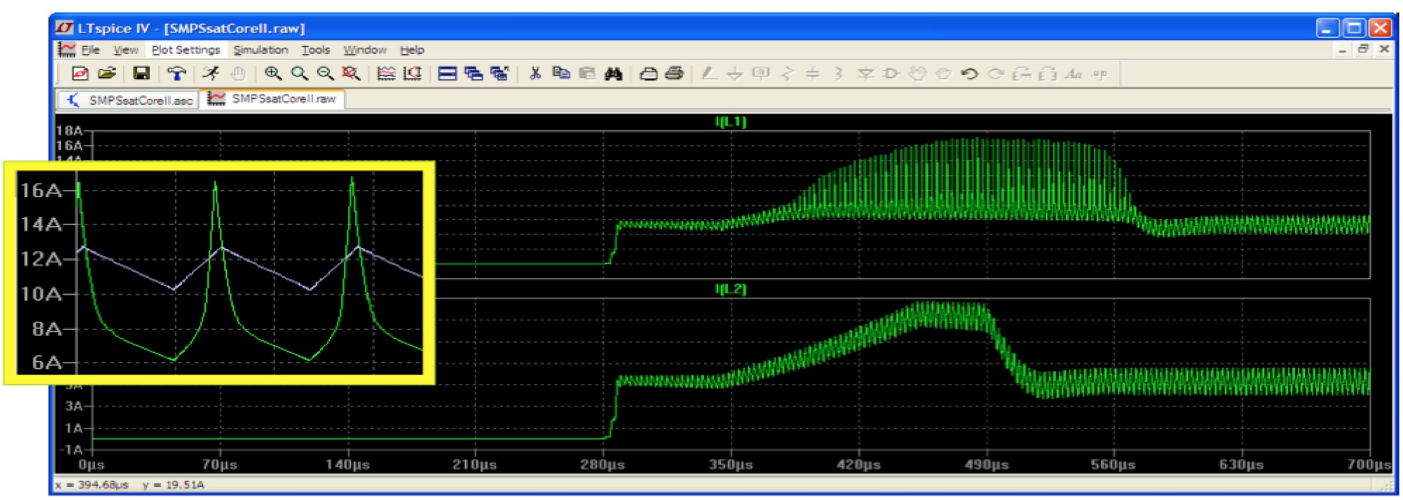
SMPS Inductor Saturation



ANALOG DEVICES

Core Saturation Considerations

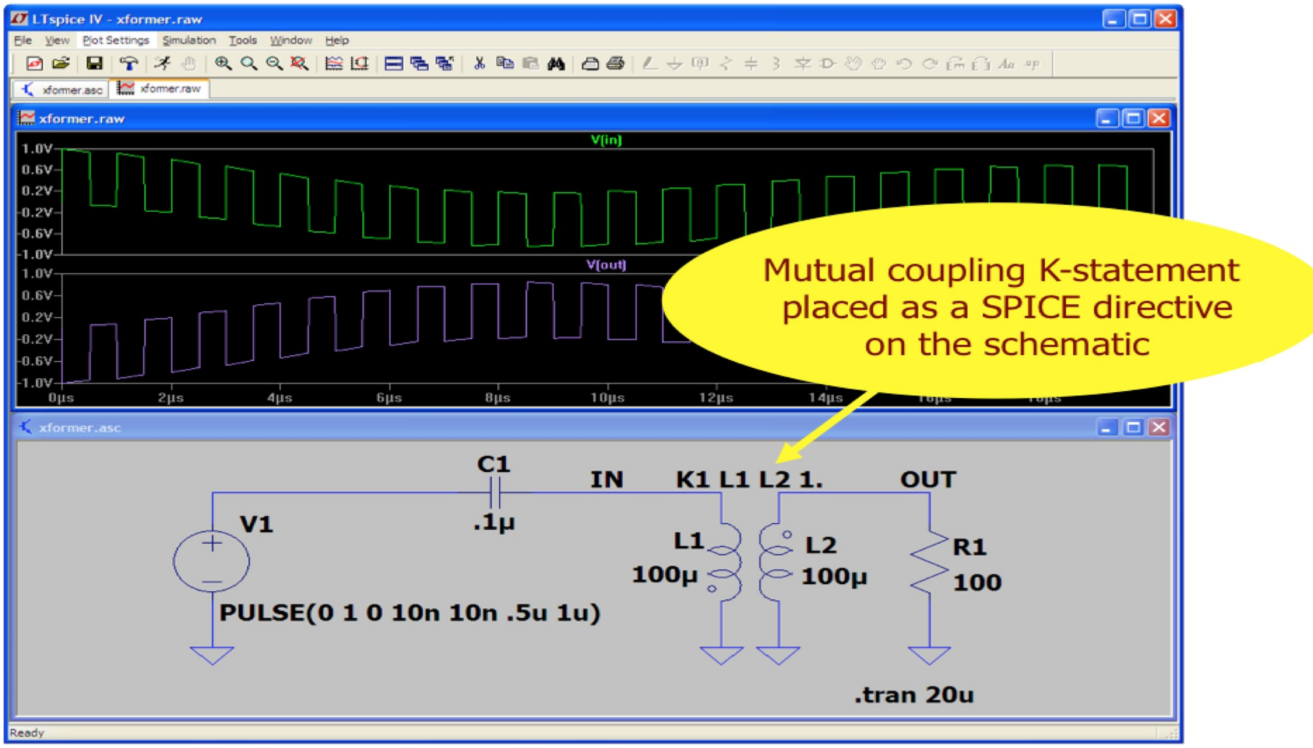
- Saturation flux density goes down monotonically with temperature
- Maximum service temperature plus self-heating
- Controller peak current production scatter
- Startup/transient/short circuit conditions



If you use the worst inductor that works in simulation, you will have failures over service temperature and production scatter.

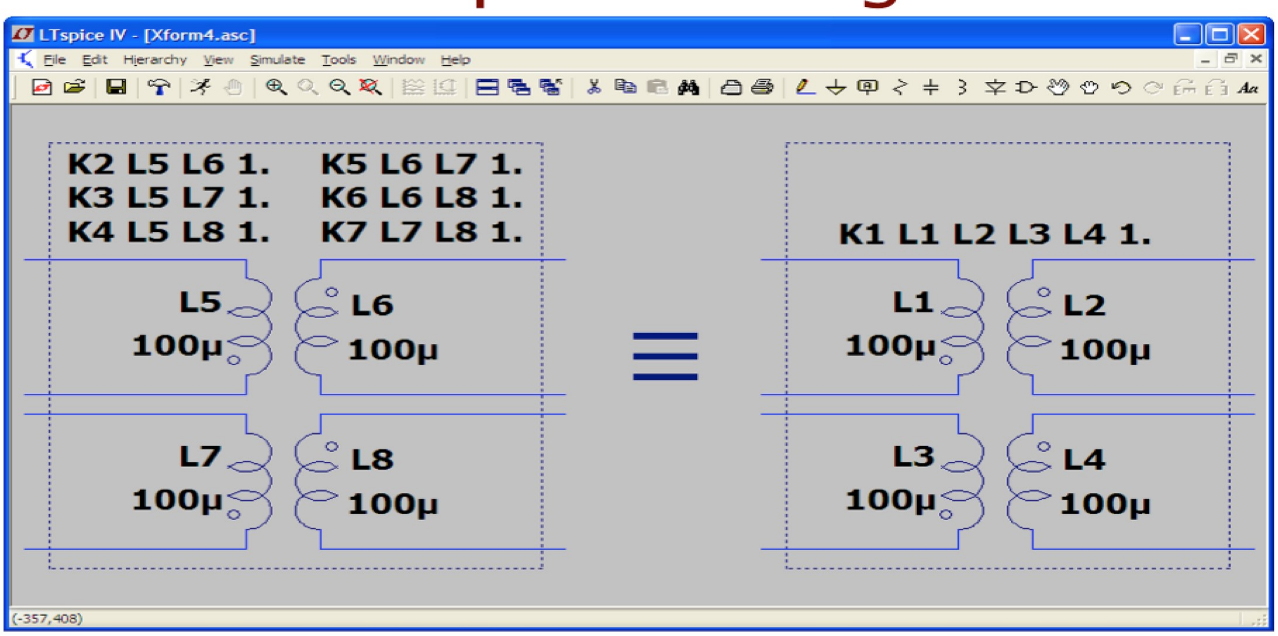
ANALOG DEVICES

Simulating Transformers



ANALOG DEVICES

Multiple Windings



For N windings, the number of mutual couplings is $\frac{N(N-1)}{2}$

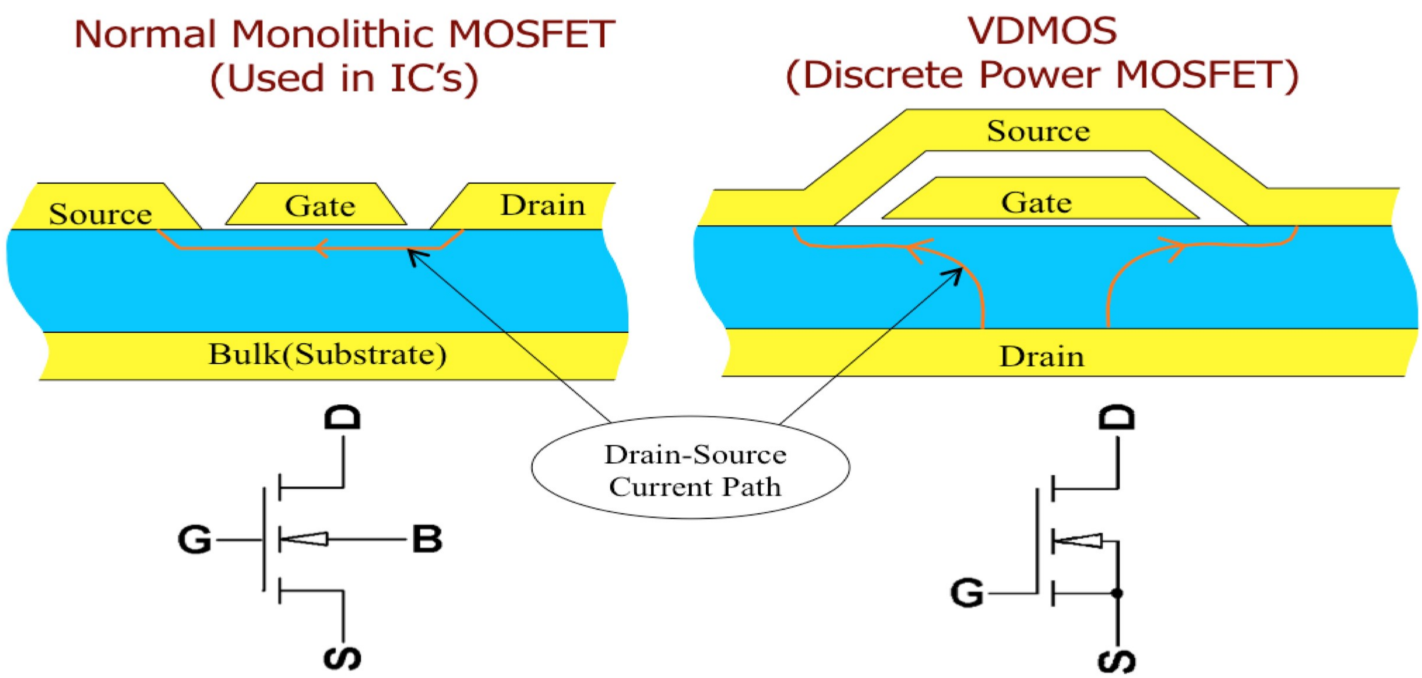
ANALOG DEVICES

LTspice's Special Enhancements for SMPS Simulation

- Automatic Steady State Detection and Efficiency Computation
- VDMOS MOSFET Model
- Node Reduction
- Mixed-Mode Simulator with intrinsic SMPS controller functions
- Nonlinear magnetics with gapped magnetic circuit solver(US Patent 7,502,723)

ANALOG DEVICES

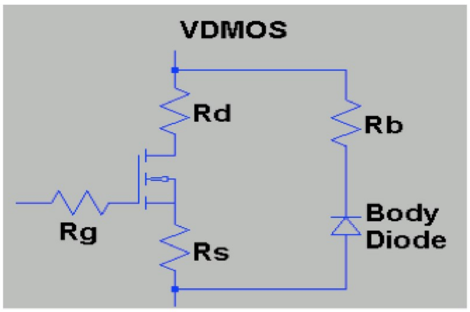
VDMOS MOSFET

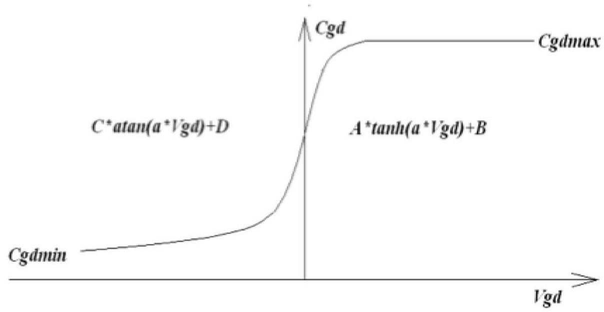


ANALOG DEVICES

LTC Proprietary VDMOS Model

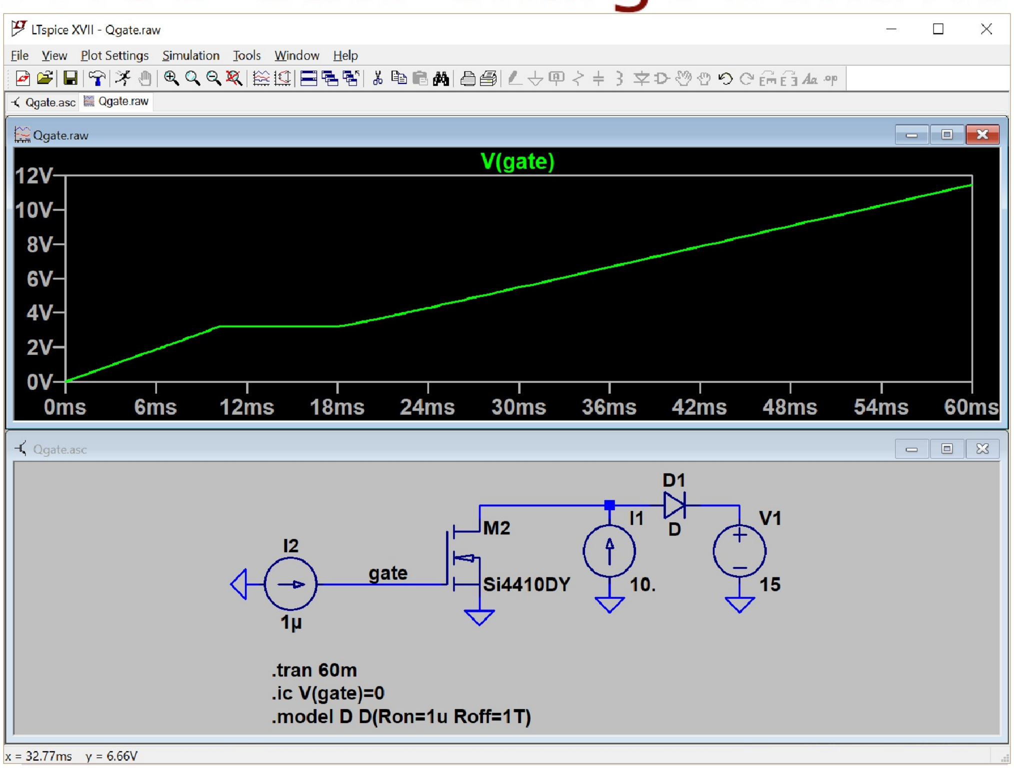
Replace a problematic subcircuit with a single new intrinsic SPICE device





ANALOG DEVICES

VDMOS Gate Charge Behavior



ANALOG DEVICES

And That's Often Not Even the Worst of It!

To get the charge correct:

The I-V curve got botched:

ANALOG DEVICES

How Do I Add 3rd Party Models?

- .model Statements
  - supply parameters for the built-in device equations of native internal devices.
  - Common for diodes, bipolar's, and JFET's
- .subckt Statements: Random Libraried Circuitry
  - Automatic symbol generation!

ANALOG DEVICES

Beware of OpAmp Models

- Boyle Model
- Noise

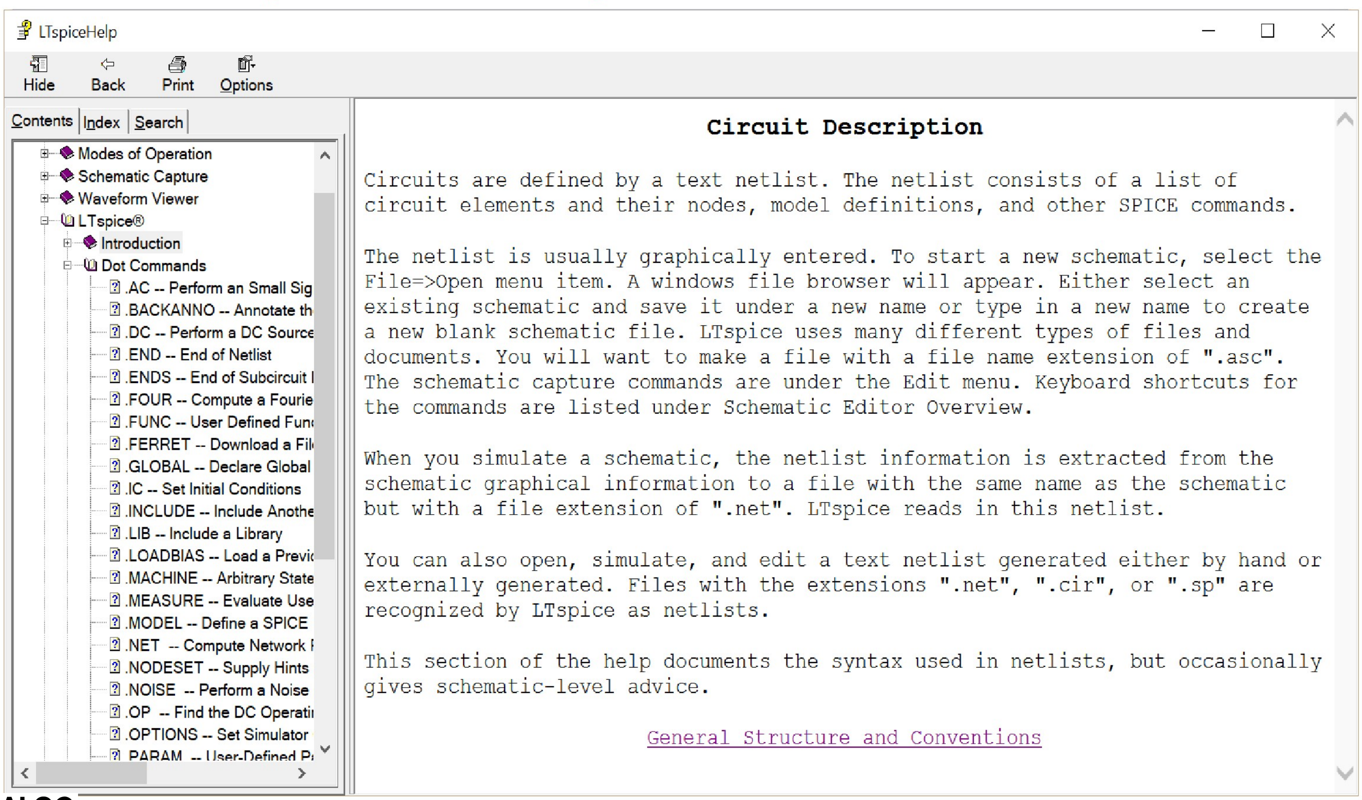
Misc. Advanced Features

- Waveform plot annotations
- Hierarchy
  - automatic symbol generation
- BUS's
- Fast Access file format
- .measure statements
- Optional double precision data files
- Read/Write .wav files
- URL's in a .lib and .inc statements
- Color Preference Editor

Misc. Advanced Techniques

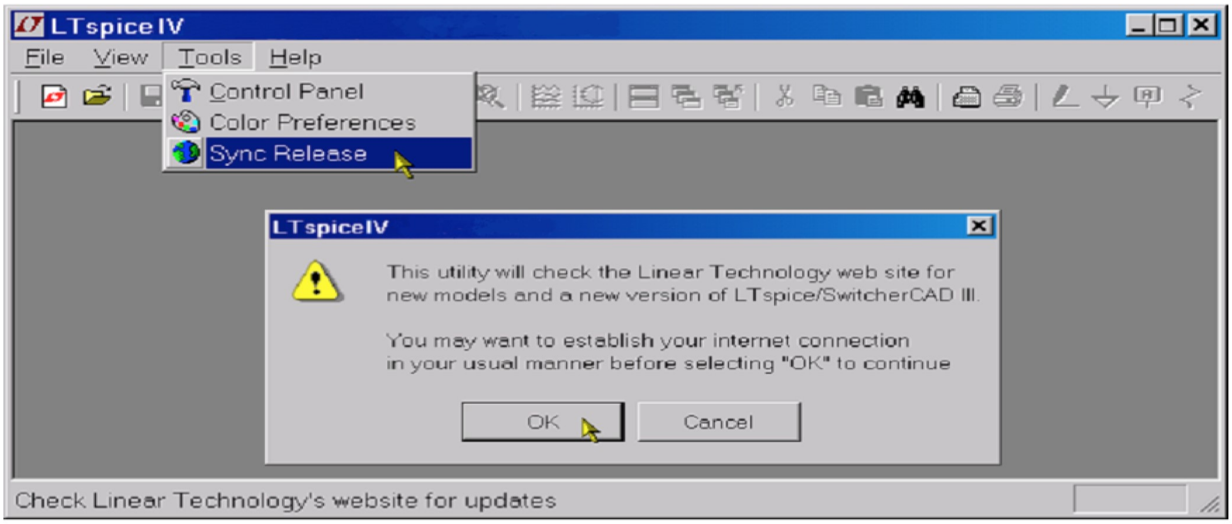
- User-defined parameters & functions
- .step'ing a user-defined parameter
  - Overlay simulation runs
  - Parameter sweeps
  - Monte Carlo
  - Optimization
  - .step'ed .meas data can be plotted
- Place .op data on the schematic
- Using the Universal Opamp Model

Complete Help Documentation



Updates With Field Sync

- Incrementally updates your installation off the web
- Automatically merges databases of devices
- Free Lifetime Updates



ANALOG DEVICES

Thanks for Listening!

POWER BY LINEAR™

ANALOG DEVICES