



 POWER BY  
LINEAR™

 ANALOG  
DEVICES

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

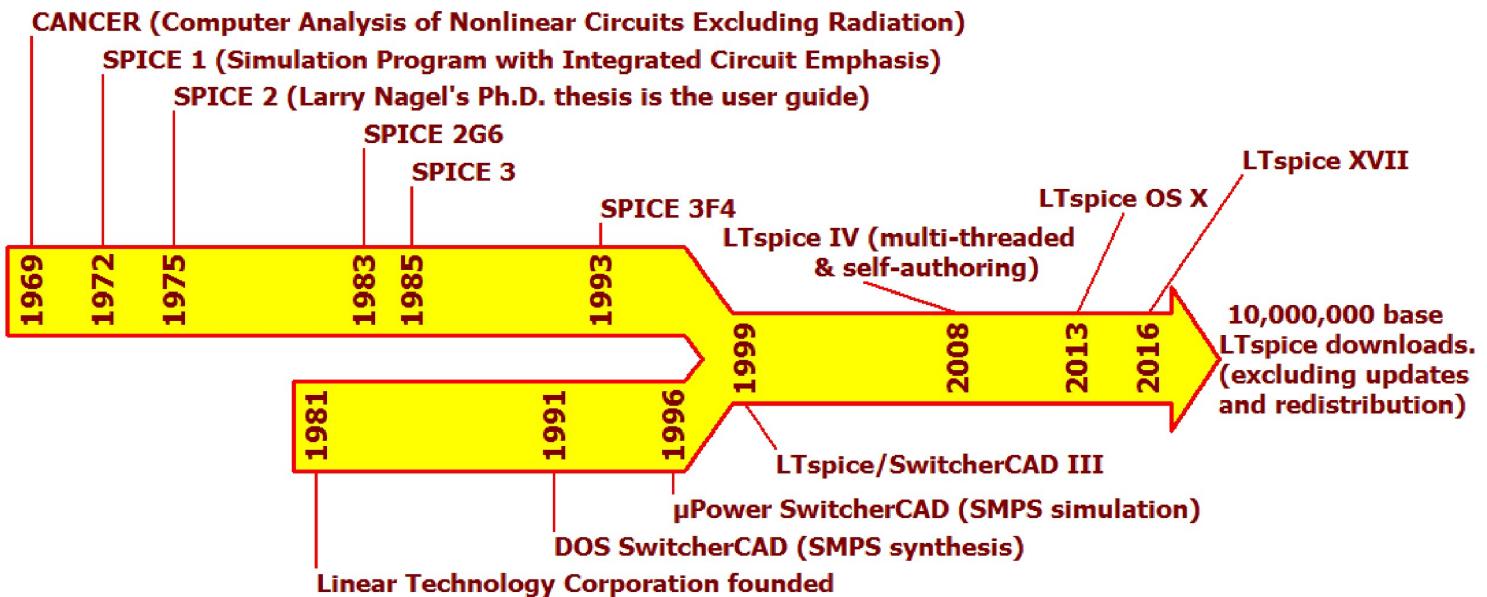
- Best charged-particle optic simulator: scanning electron microscope company
- Best MOSFET circuit solver: Intel
- Best radar return solver: Government Intelligence
- Best analog circuit solver: IC company, Linear Technology Corporation

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.

 POWER BY  
LINEAR™

 ANALOG  
DEVICES

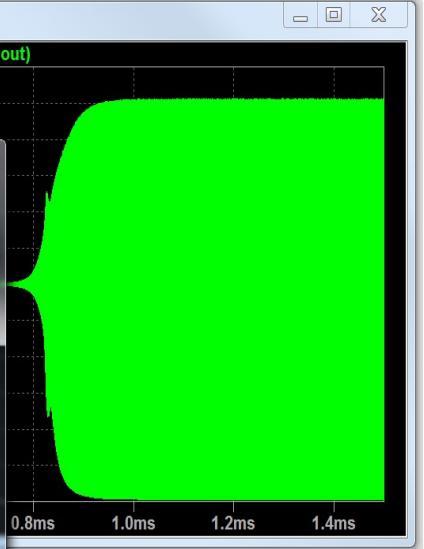
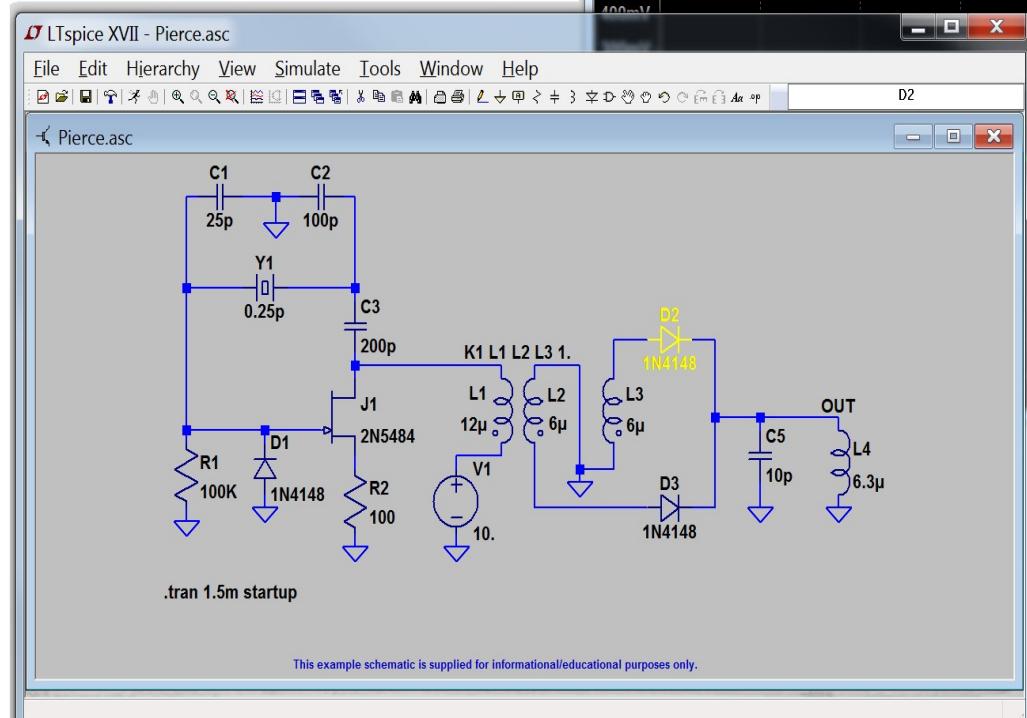
# Time Line of SPICE



- 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



## LTspice XVII



- Multi-monitor
- x64
- UNICODE ☒

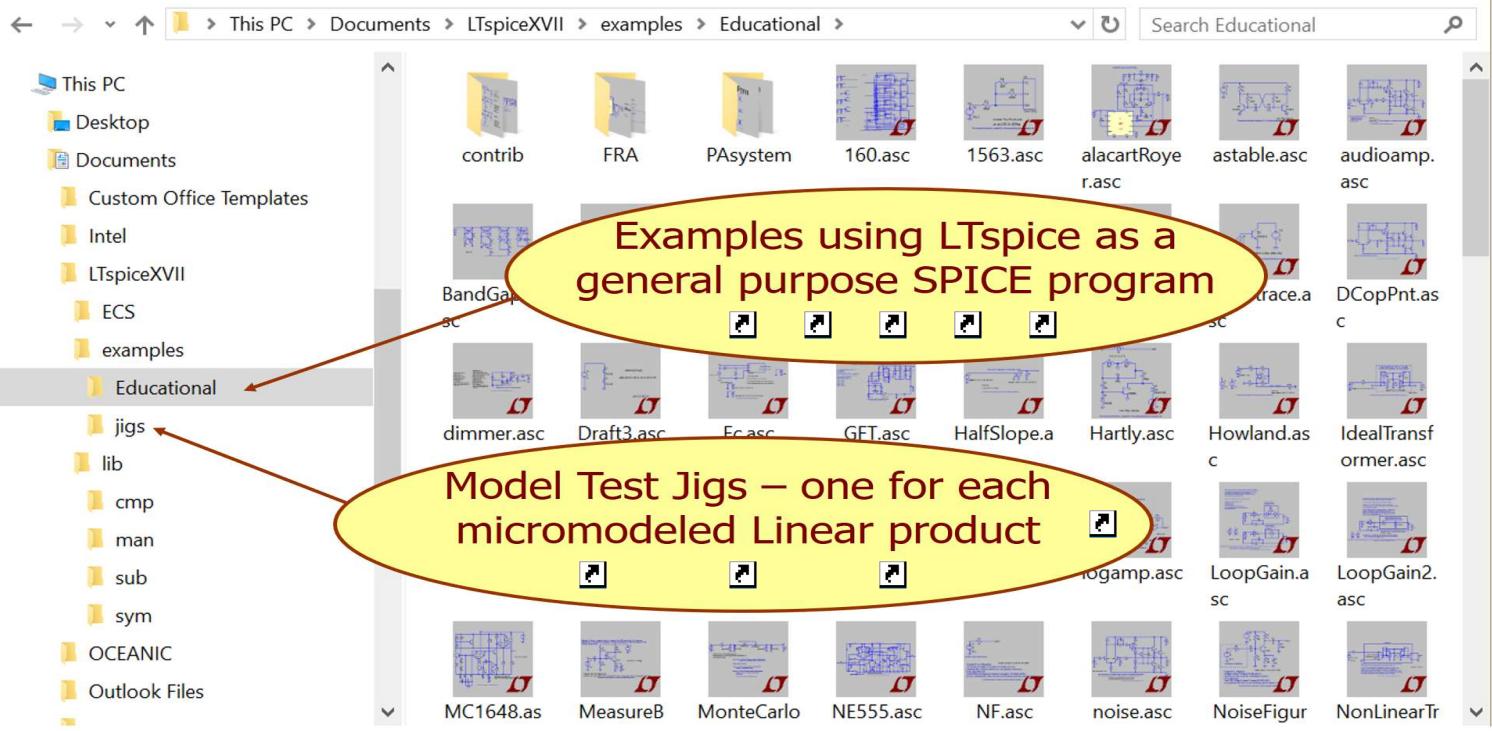


# Drafting Your Own Circuits

- 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 3<sup>rd</sup> party models
  - Active independent users' group



## Installation Directory Structure



# Additional Resources for Example Files

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

The screenshot shows a section of the Analog.com website for the LTM4600. On the right, there's a sidebar with links to 'Reliability Data' and 'RoHS Material Declaration'. Below that is a 'Samples' section with a note to contact local sales offices. To the right of that is a 'Simulation' section with links to 'How To Simulate The LTM4600' and 'LTspice Demo Circuit'. A red arrow points from the text '- Some products feature an example LTspice schematic' in the previous slide to the 'LTspice Demo Circuit' link.

- <http://groups.yahoo.com/group/LTspice>
- FAE's at Analog
- [LTspice@analog.com](mailto:LTspice@analog.com)



## 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)



# 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 



# 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



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

- Iq in all modes
- Misc features in shutdown

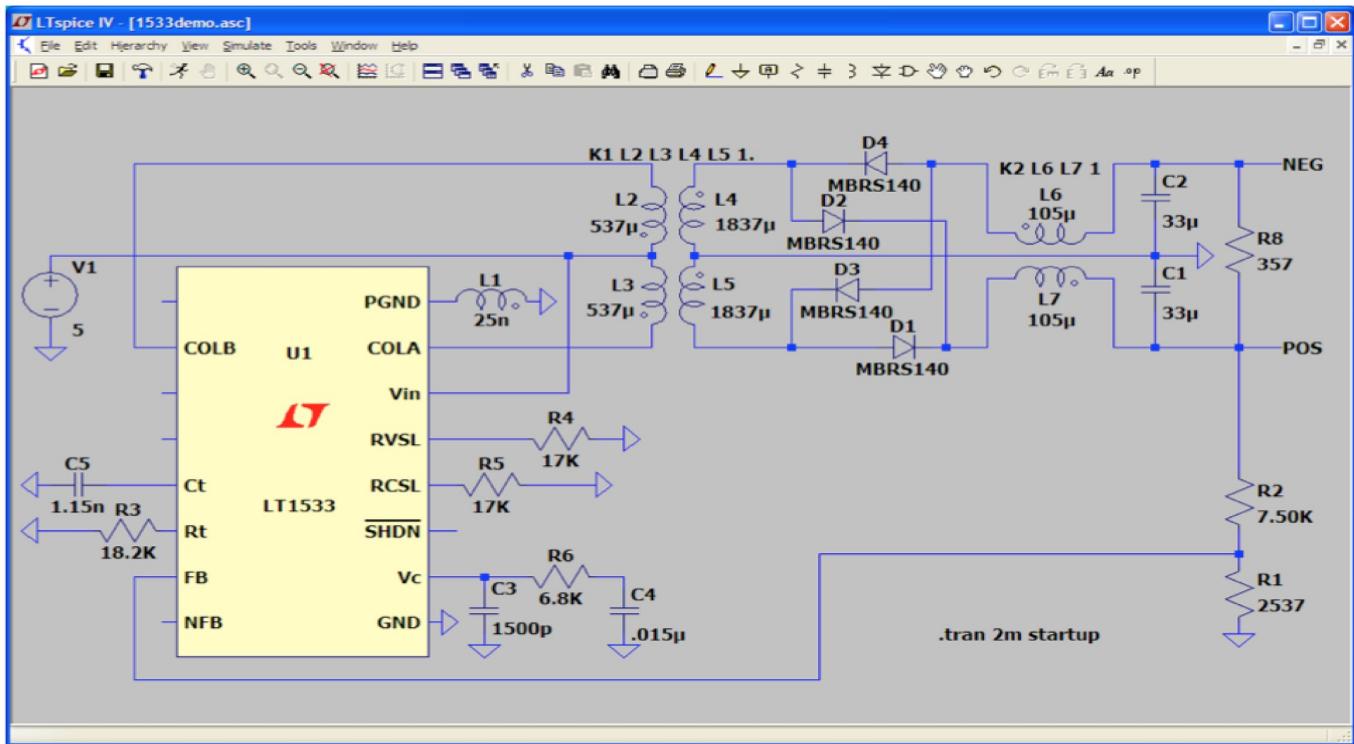


## LT1533 Case Study

- 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



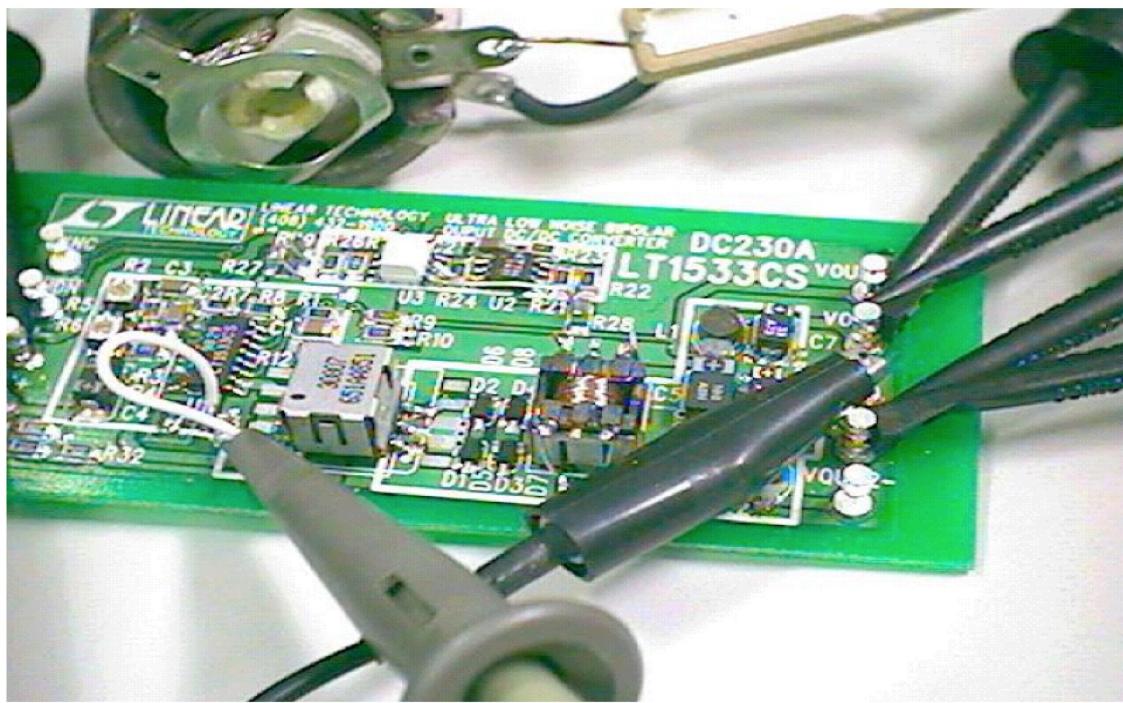
# Demo Board Schematic



POWER BY  
LINEAR™

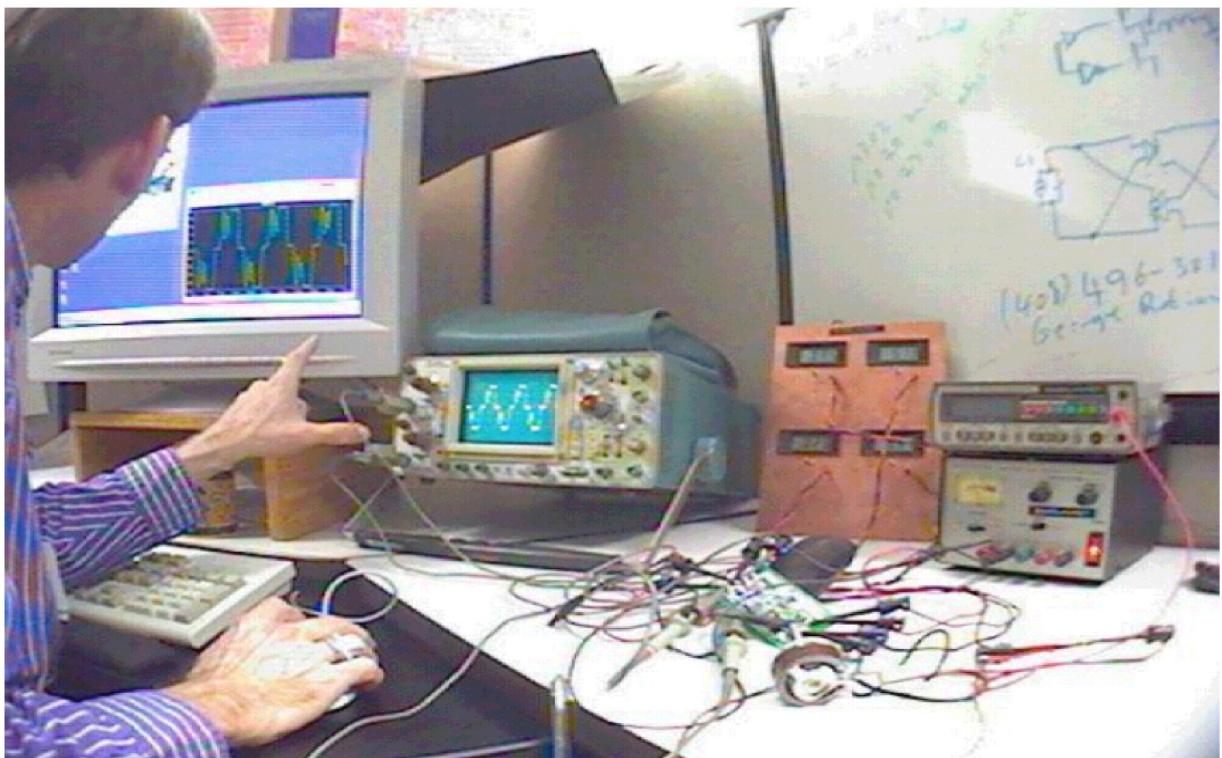
ANALOG  
DEVICES

## Demo Board



POWER BY  
LINEAR™

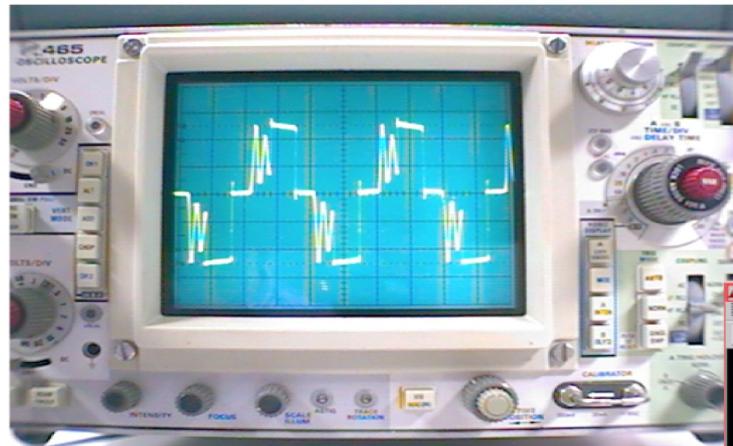
ANALOG  
DEVICES



 POWER BY  
**LINEAR**™

 ANALOG  
**DEVICES** 

## Minimum Slew Limits



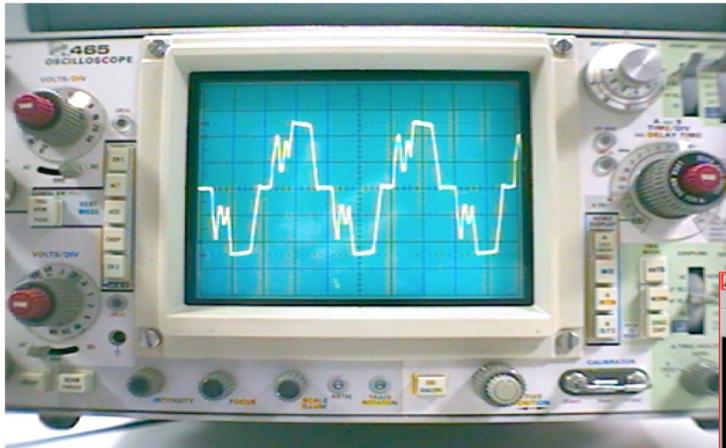
$V_{in} = 5V$   
 $V_{out} = +/-5V$   
 $I_{out} = 28mA$



 POWER BY  
**LINEAR**™

 ANALOG  
**DEVICES** 

# With Voltage Slew Limit



$V_{in} = 5V$   
 $V_{out} = \pm 5V$   
 $I_{out} = 28mA$



## 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%

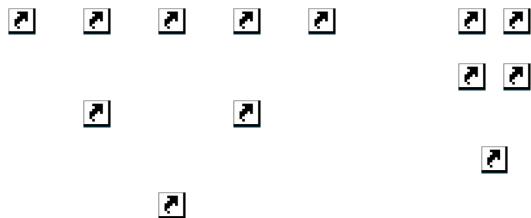




## SMPS Stability

*or*

## Open Loop Response From the Closed Loop System

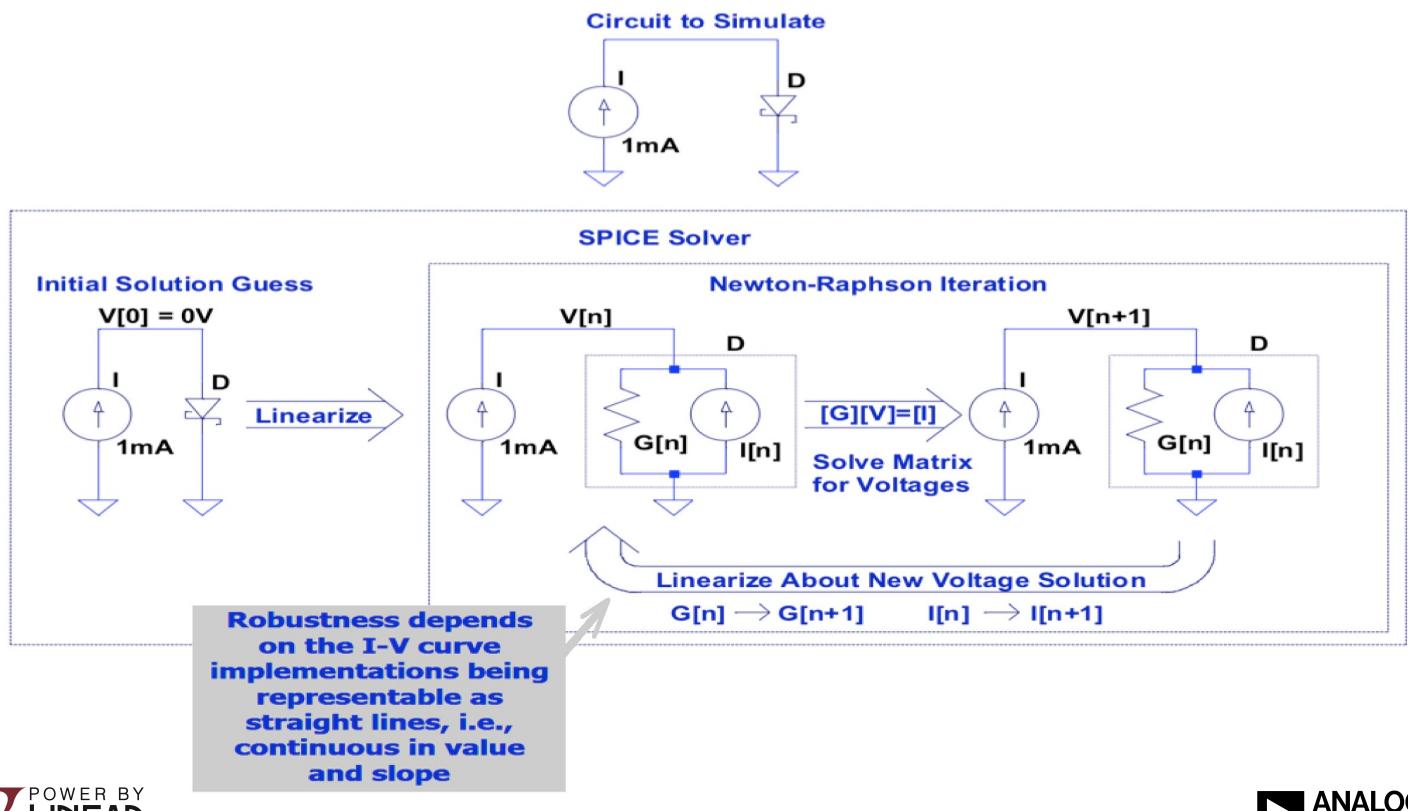


## Three Numerical Methods Account for the Success of SPICE

- Newton iteration
- Sparse matrix methods
- Implicit integration



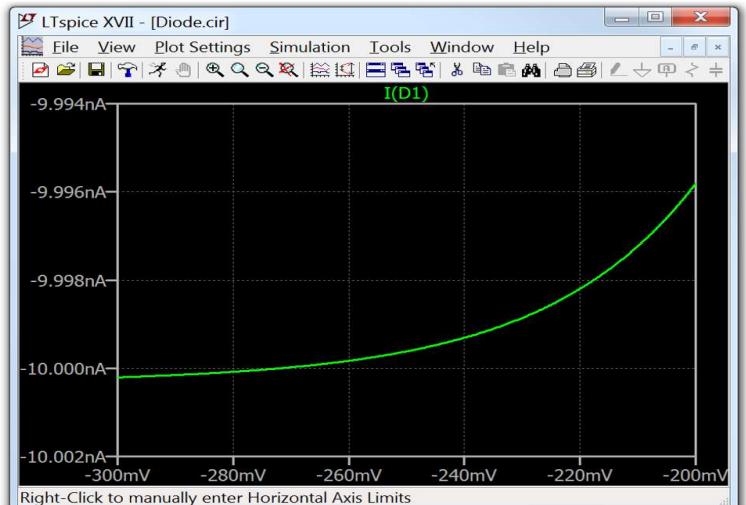
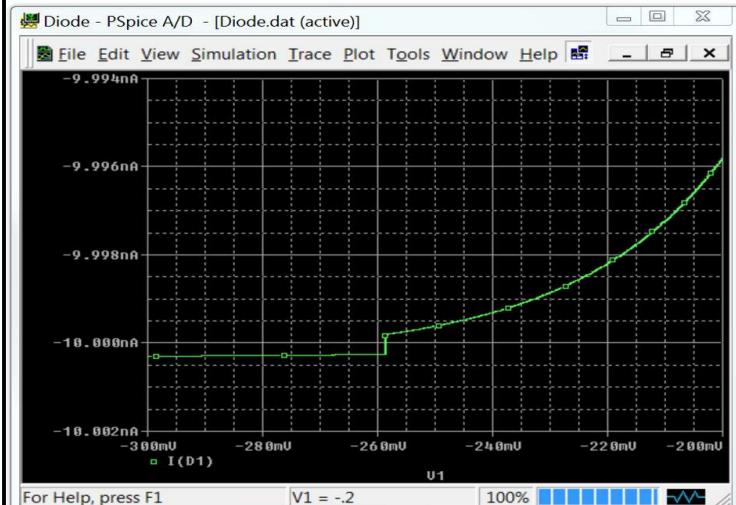
# Newton Iteration



POWER BY  
LINEAR™

ANALOG  
DEVICES

## Berkeley Diode Discontinuity



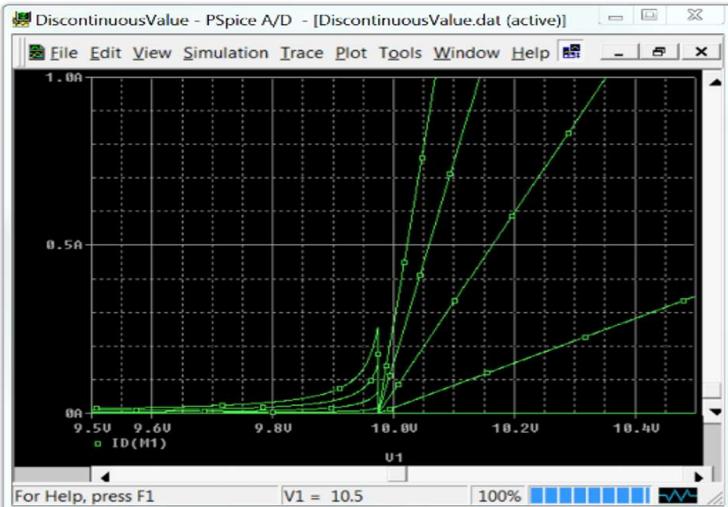
PSpice

LTspice

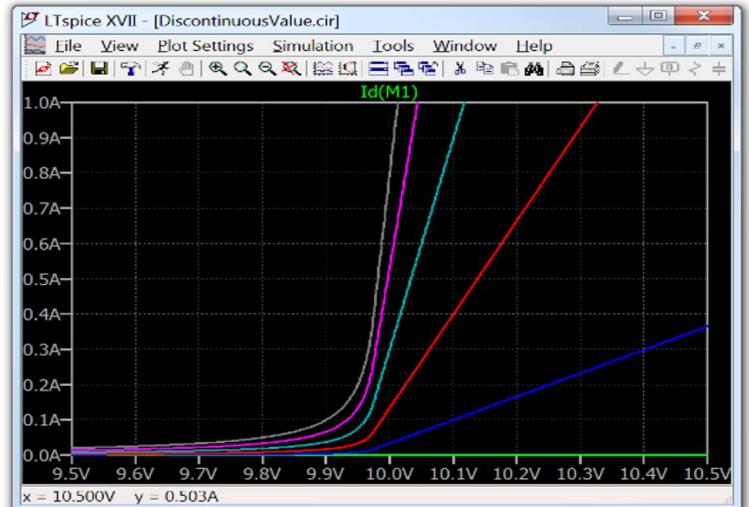
POWER BY  
LINEAR™

ANALOG  
DEVICES

# Berkeley MOS Discontinuous Value



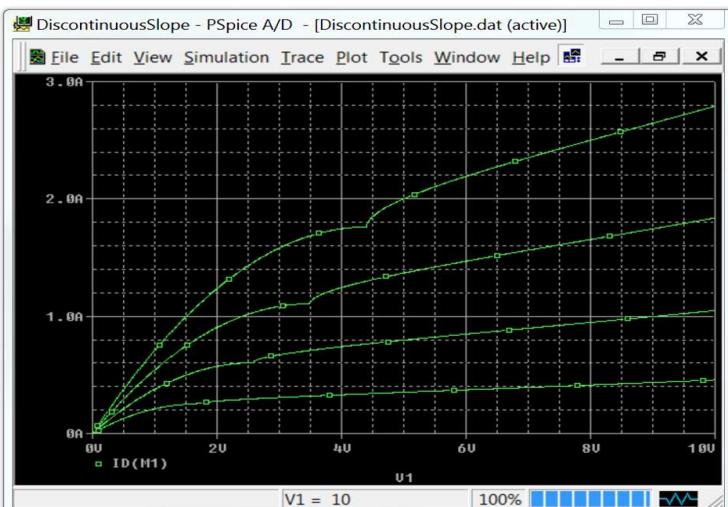
PSpice



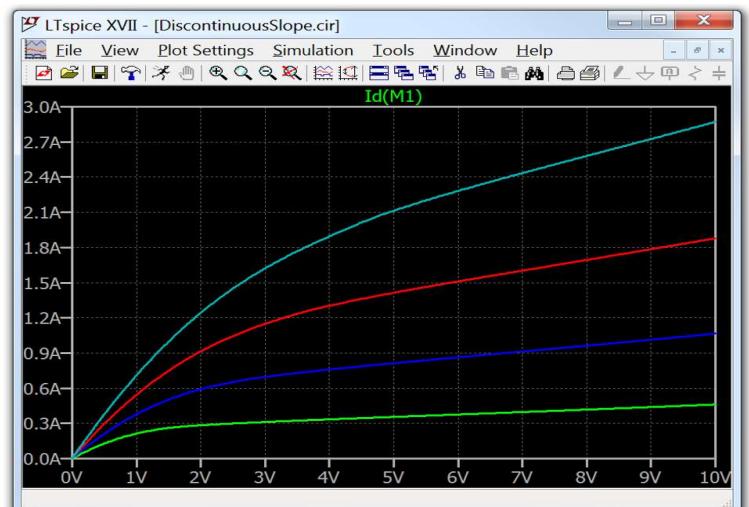
LTspice



# Berkeley MOS Discontinuous Slope



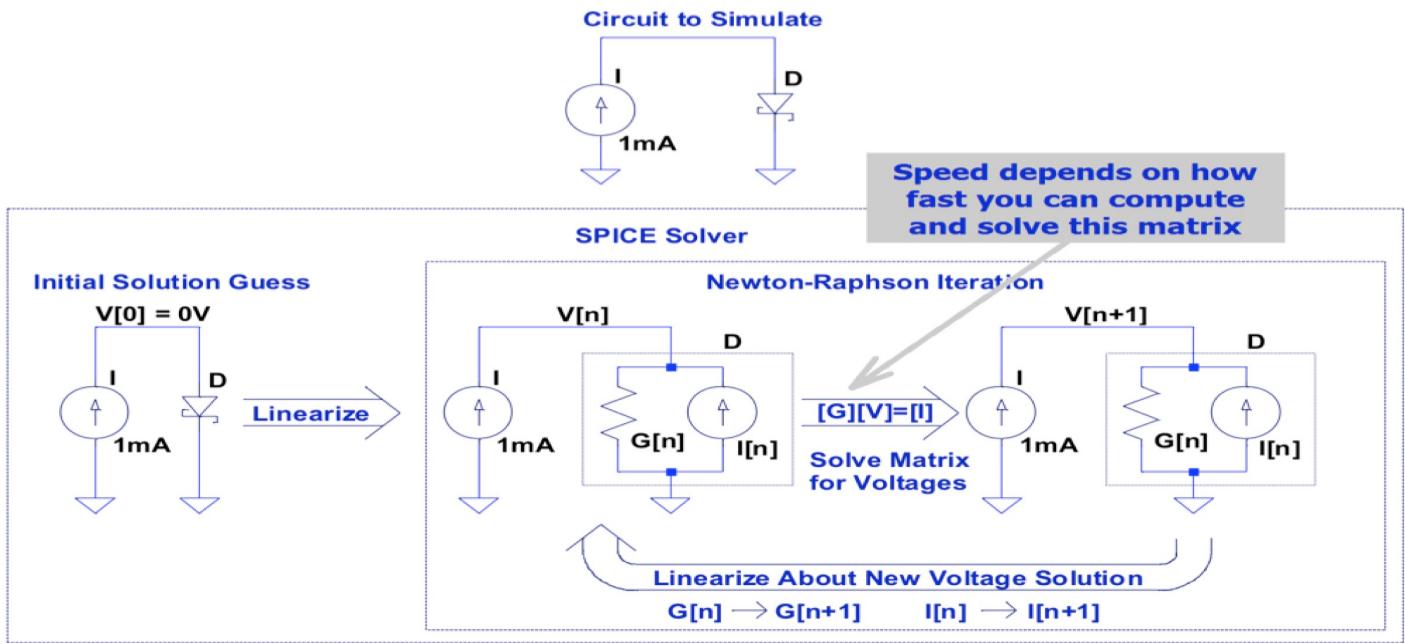
PSpice



LTspice



# Newton Iteration



## Simulation Speed Determined by

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



# Simulation Speed Determined by

$O(N)$ :

"Compute the matrix"  
means computing  
these coefficients

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

$$\begin{bmatrix} G_{11} & G_{12} & \dots & G_{1n} \\ G_{21} & G_{22} & & \\ \vdots & \ddots & & \\ \vdots & & & \\ G_{n1} & & & G_{nn} \end{bmatrix} \cdot \begin{bmatrix} V_1 \\ V_2 \\ \vdots \\ V_n \end{bmatrix} = \begin{bmatrix} I_1 \\ I_2 \\ \vdots \\ I_n \end{bmatrix}$$

$O(N^2)$ :

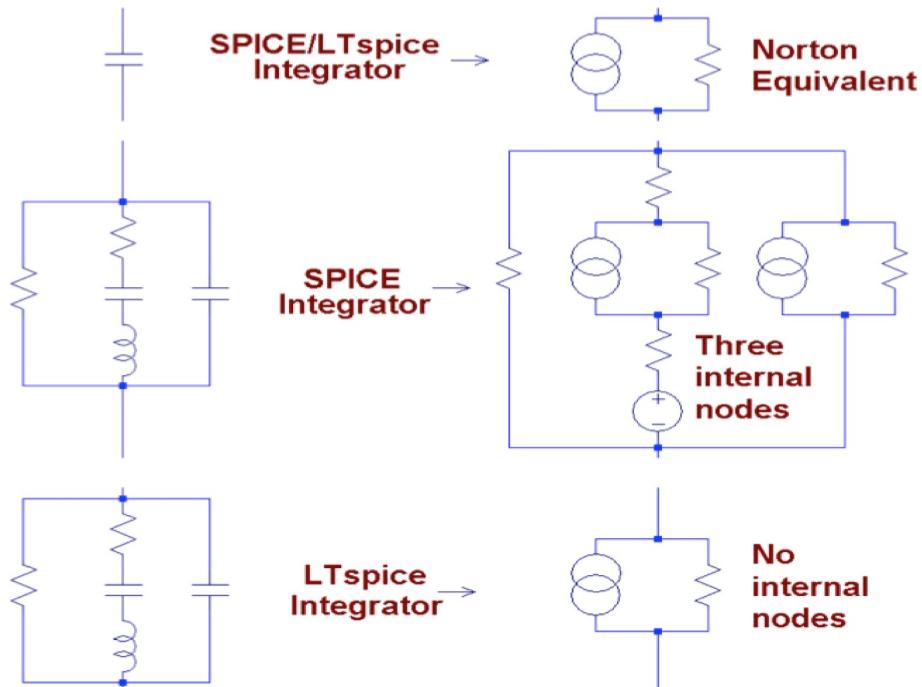
"Solve the matrix"  
means solving for  
this vector

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

## So use smaller matrices!



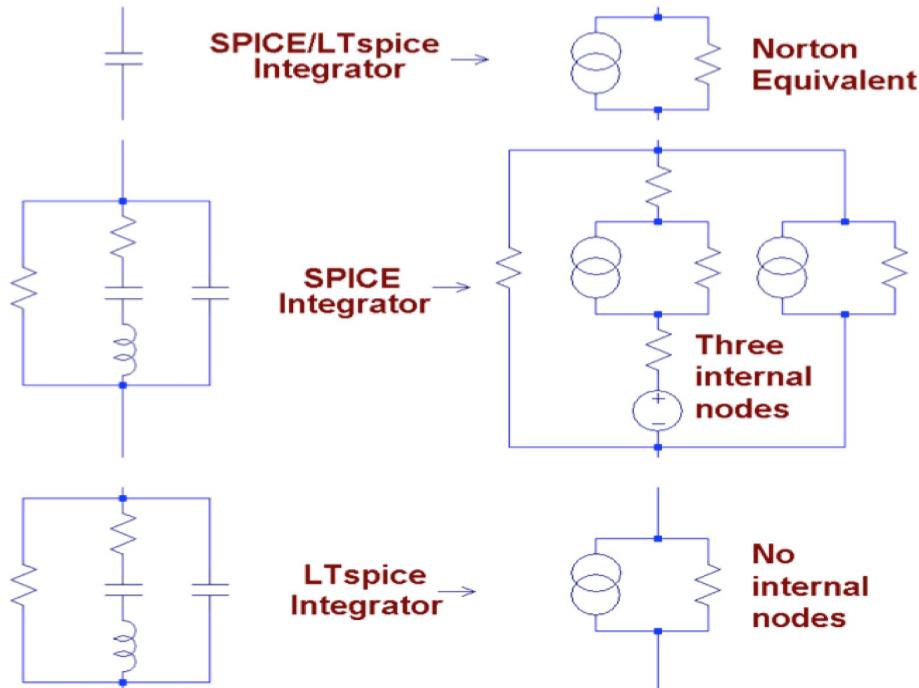
## Matrix Node Reduction



## But what happens to the accuracy?



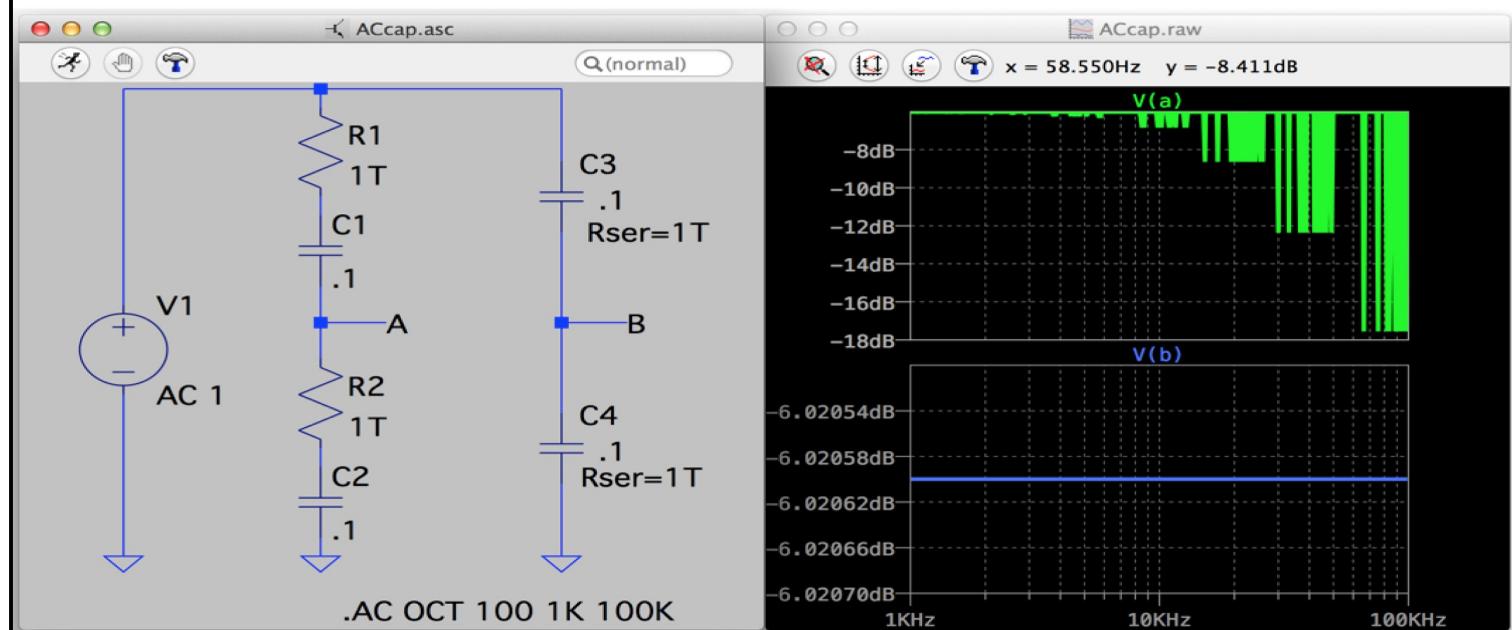
# Matrix Node Reduction



*Node reduction improves accuracy!*



## Node Reduction Example



Node reduction improves both speed and accuracy



# Implicit Integration

- Circuit reactances give rise to differential equations that are numerically integrated.
- Because the solution looks like  $\exp(-\text{const} \cdot \text{time})$ , 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 2<sup>nd</sup> order methods are used.

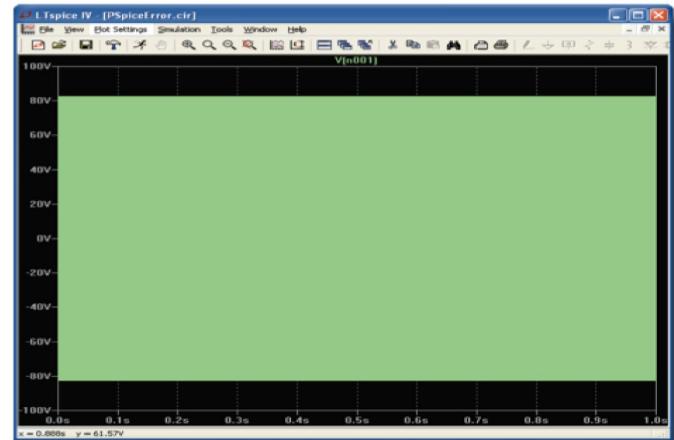
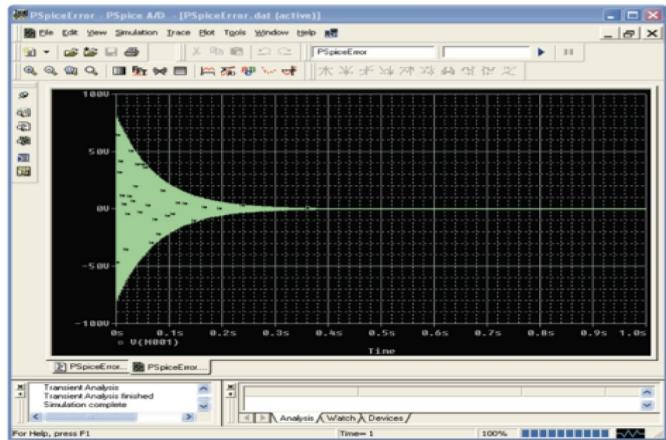
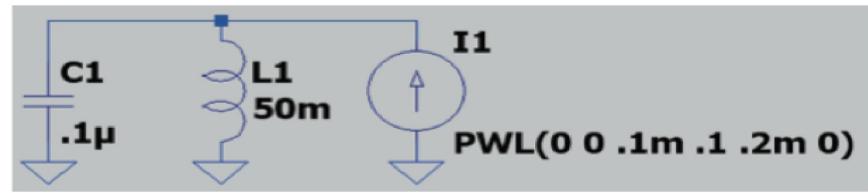


## 2<sup>nd</sup> Order Implicit Integration Methods

- 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



# Gear Integration Error



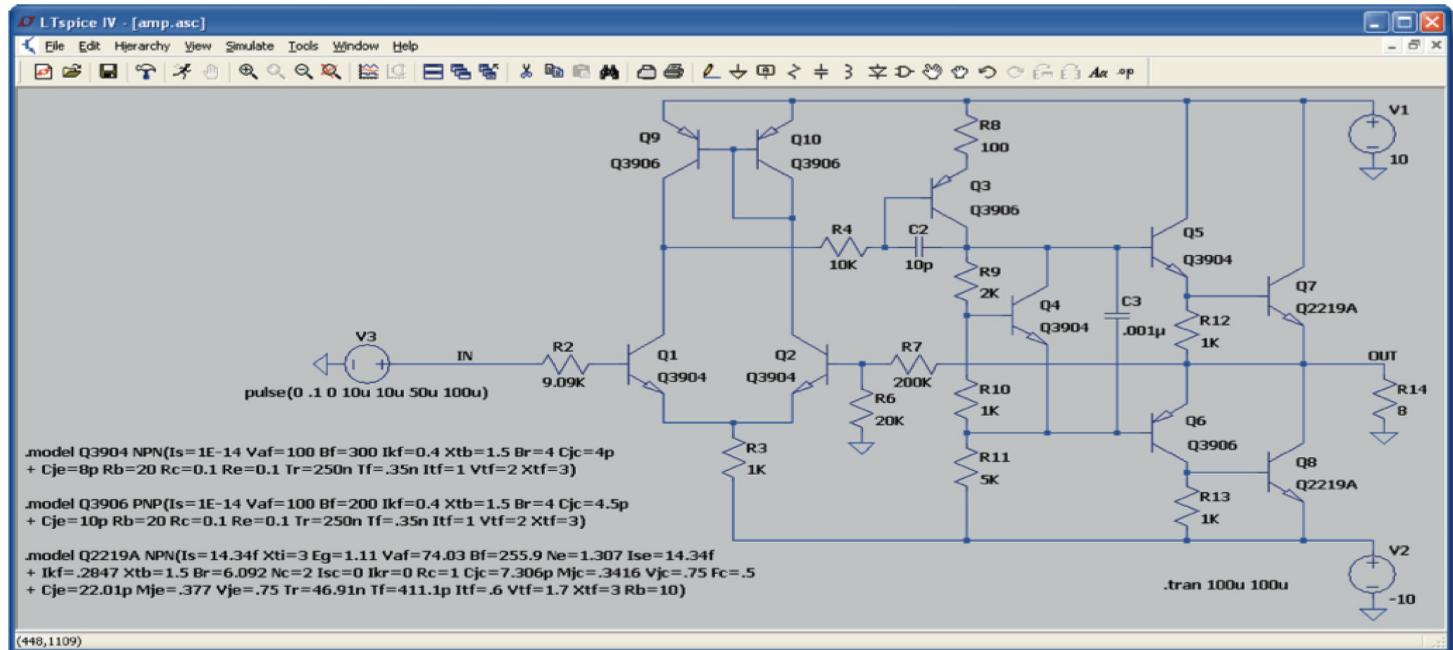
PSpice



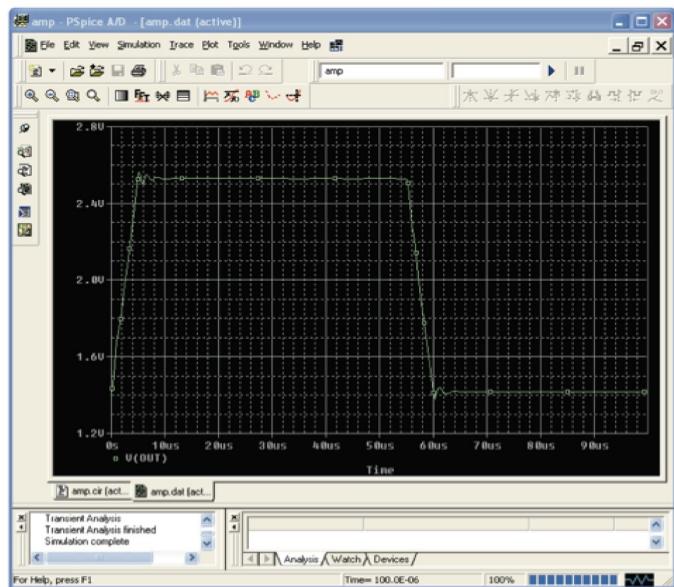
LTspice



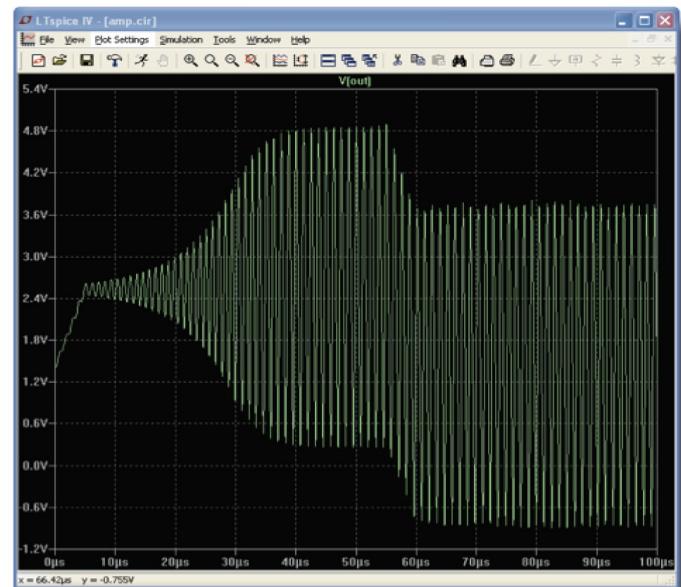
## Gear Integration Error In Context of a Practical Example



# Gear Integration Error In Context of a Practical Example



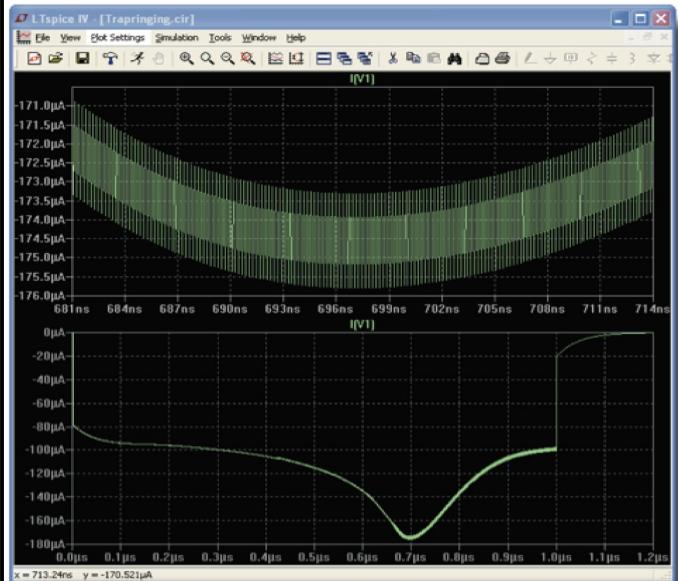
PSpice



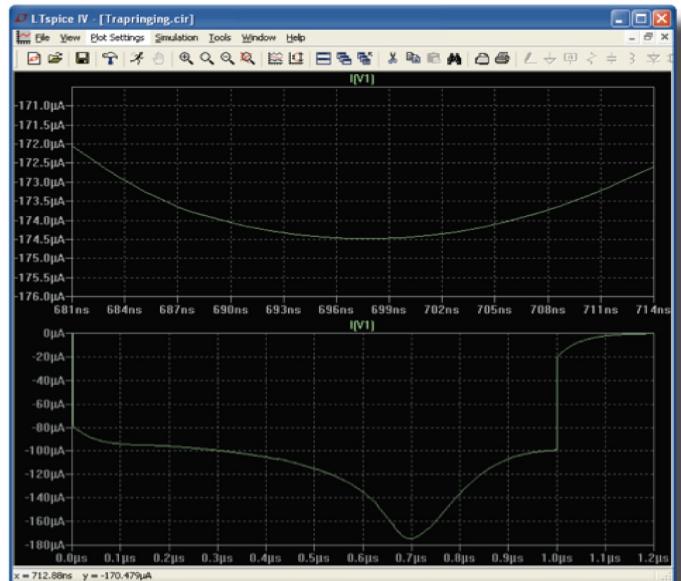
LTspice



## Trap Integration Artifact



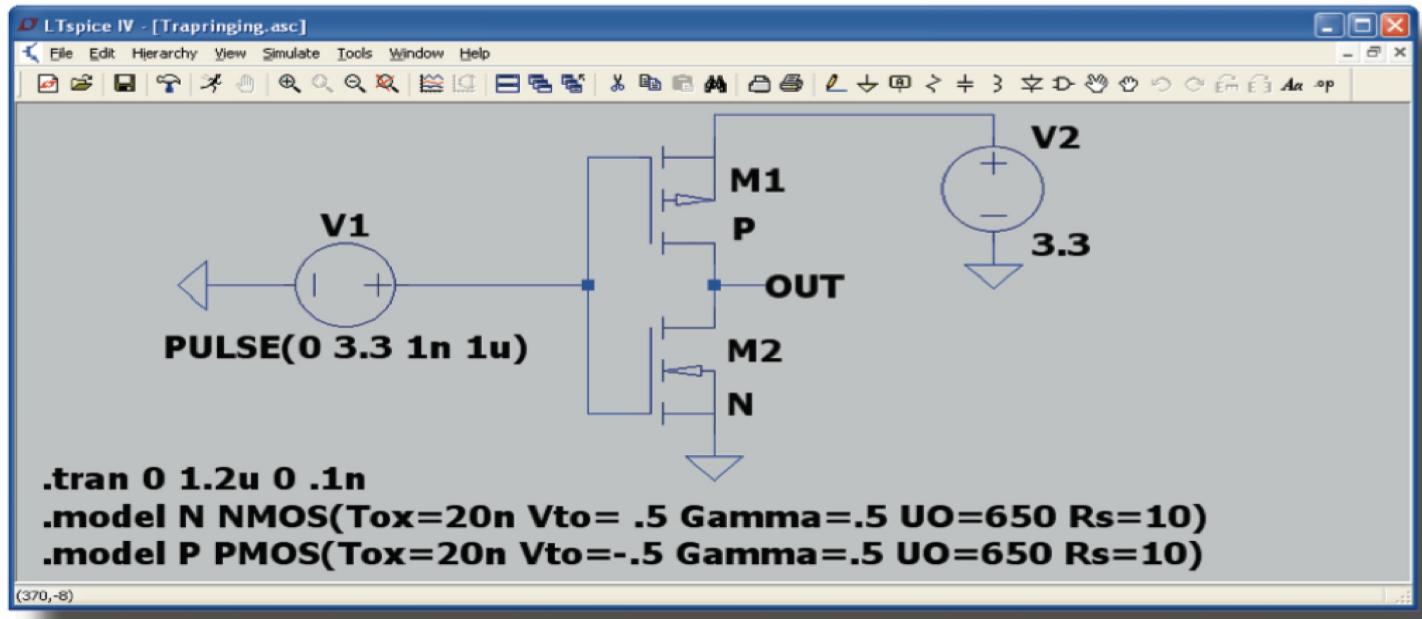
Trap



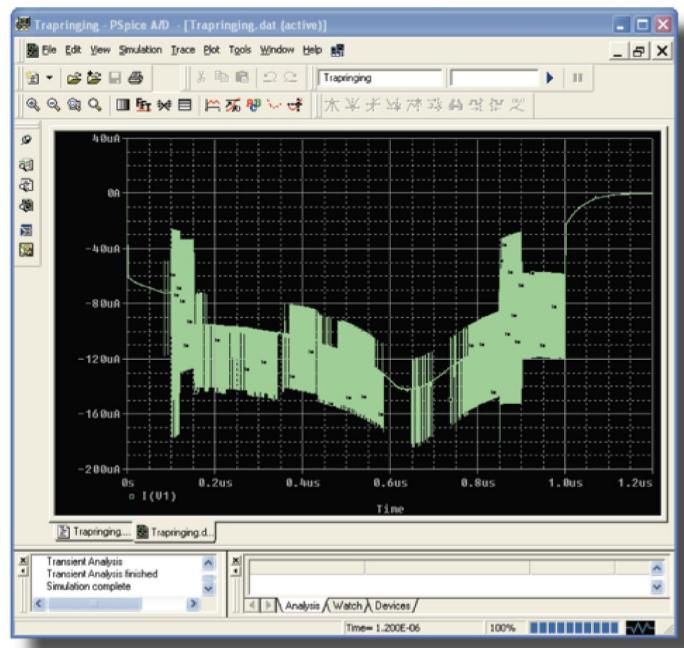
Modified Trap



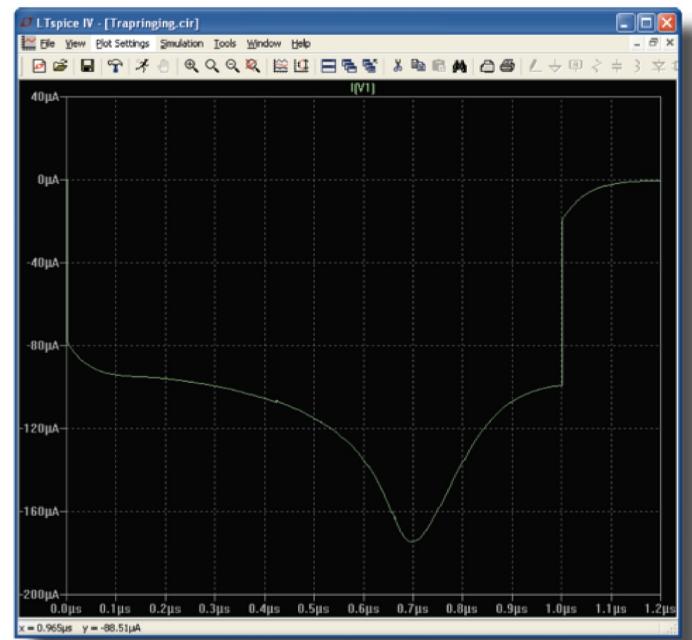
# Trap Integration Artifact Circuit



# Trap Integration Artifact Circuit



PSpice



LTspice



# Three Numerical Methods Account for the Success of SPICE

- Newton iteration
    - Need I-V curves continuous in value and slope
  - Sparse matrix methods
    - Node reduction for speed and accuracy
    - Compute matrix coefficients in parallel threads
    - Solve matrix with self-authoring code
  - Implicit integration
    - Proprietary modified trap
    - speed and accuracy of trap
    - No trap ringing
- 
- Robustness
- 
- Speed
- 
- Integrity



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.

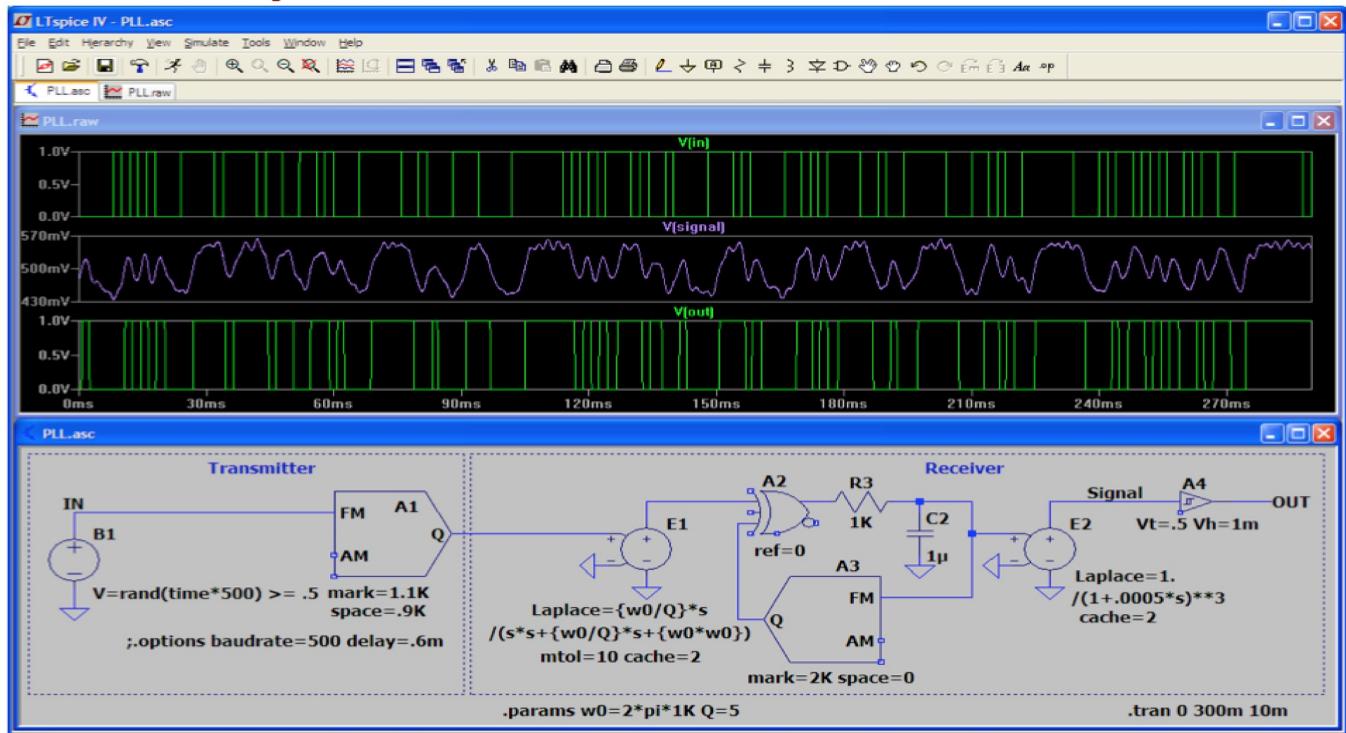


# LTspice Behavioral Simulator

- 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



## Example Mixed-Mode Simulation

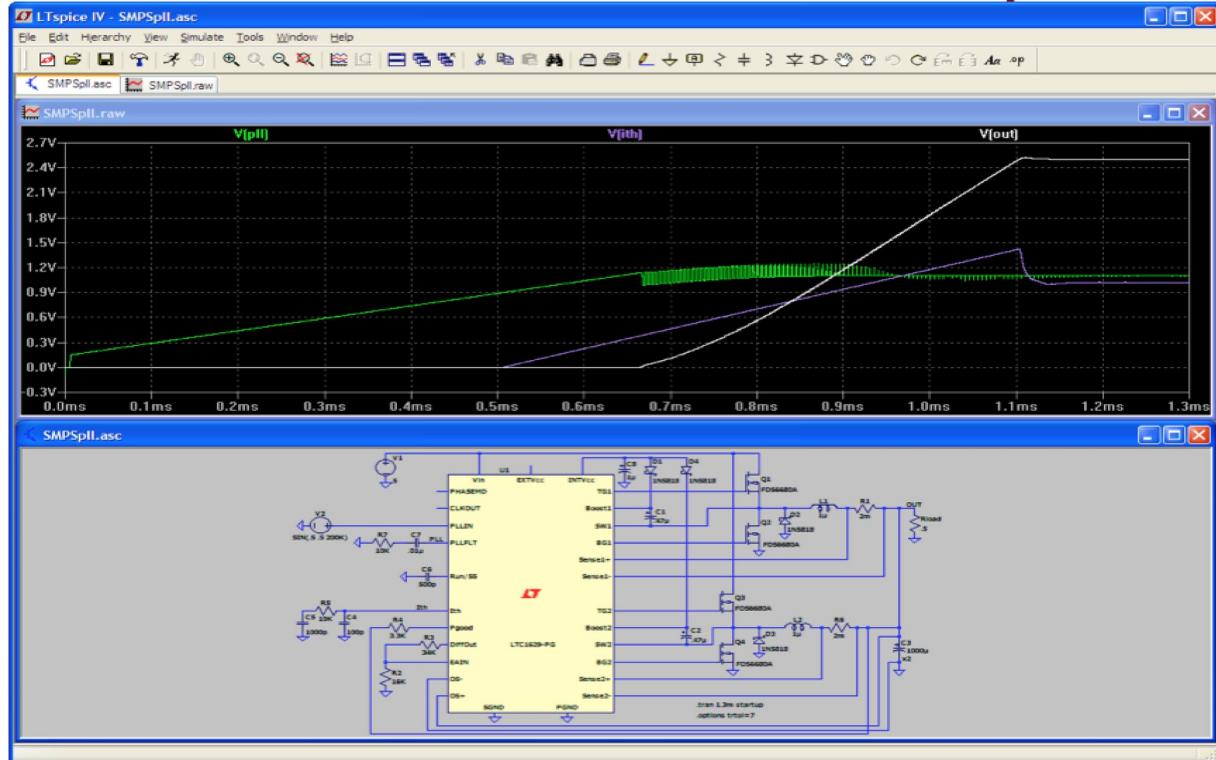


# Mixed-Mode Simulator

- 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.



## Two-Phase SMPS & PLL Capture



Total elapsed time: 5.508 seconds.



# Chan et al. Nonlinear Magnetics

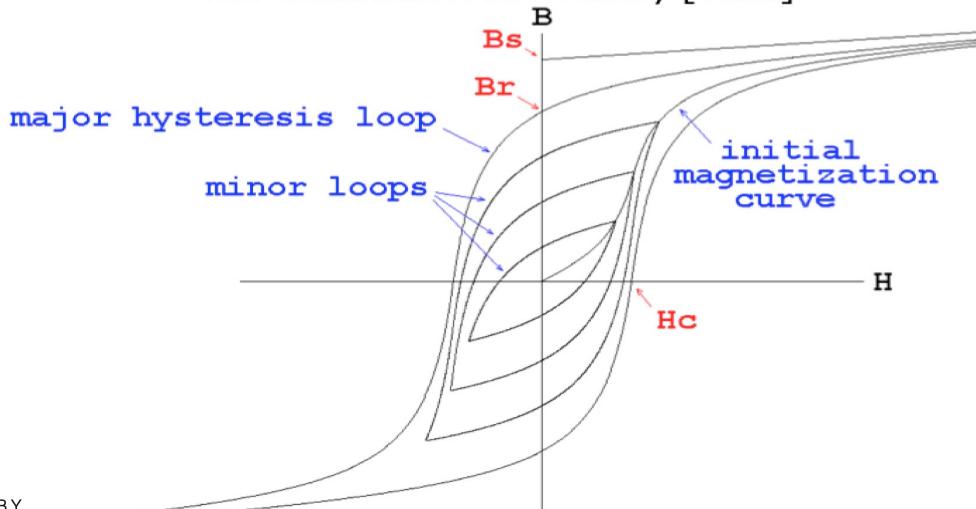
Extended per US Patent 7,502,723

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]



## Gapped Core Magnetic Solver

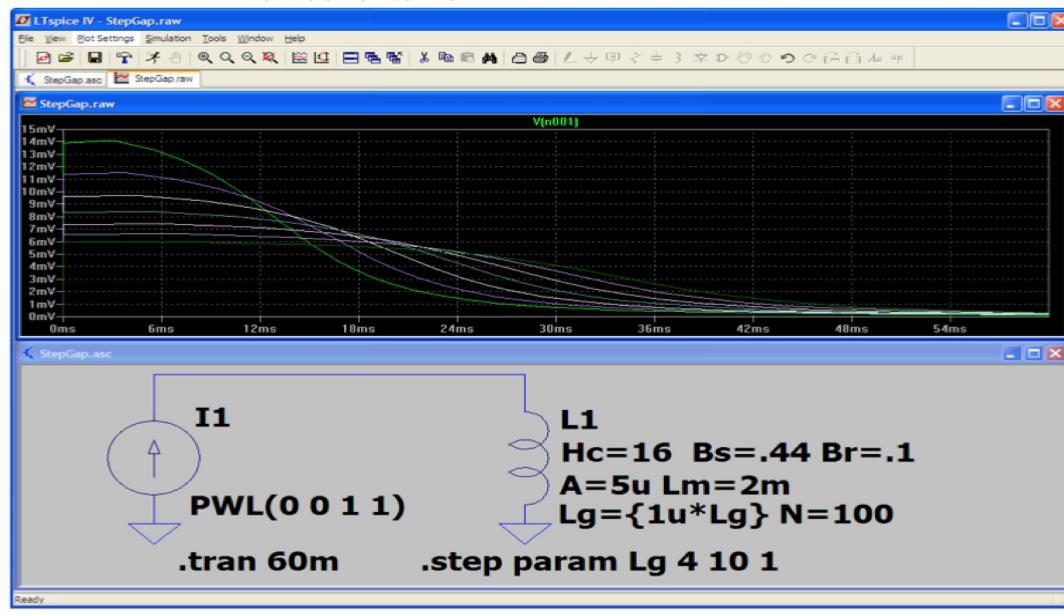
- Core physical dimensions specified with four parameters:

Lm: Magnetic Length(excl. gap)[meter]

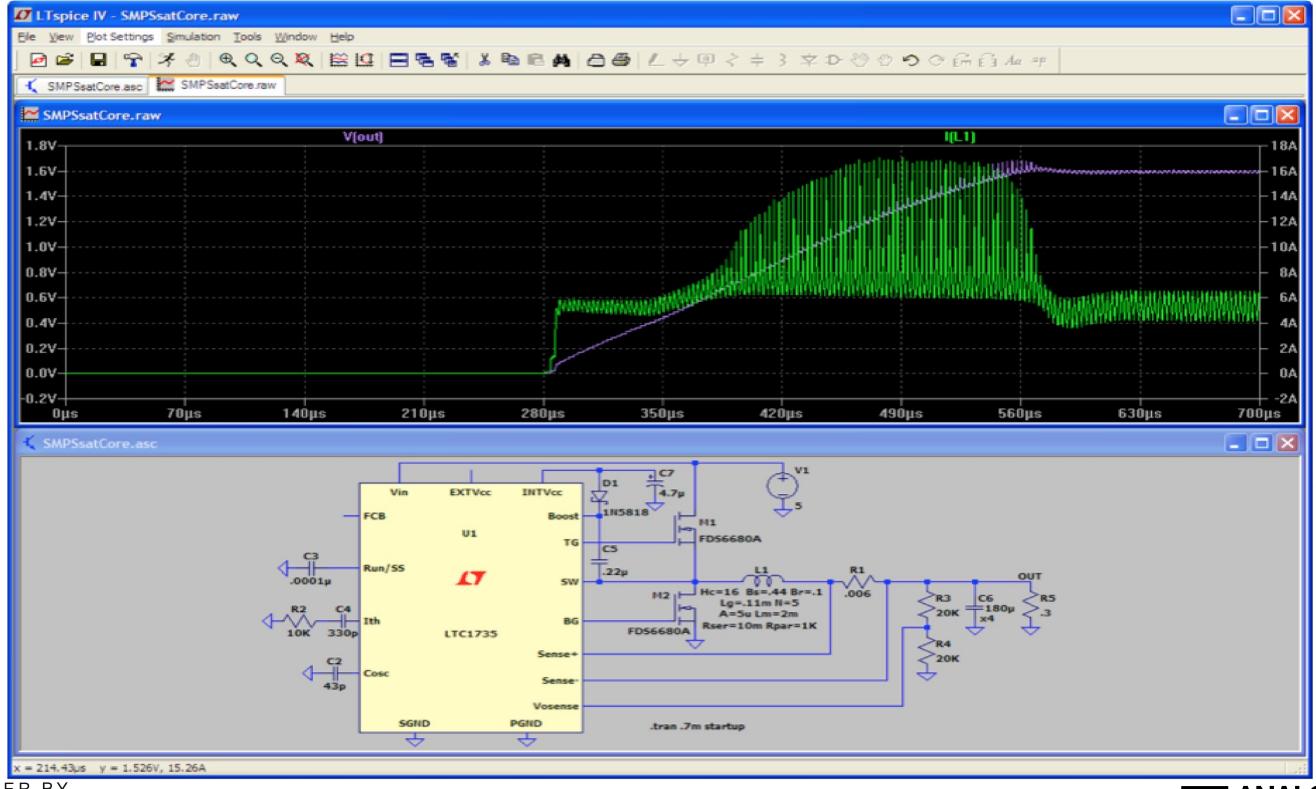
Lg : Length of gap [meter]

A: cross sectional area [meter\*\*2]

N: number of turns



# SMPS Inductor Saturation



## Core Saturation Considerations

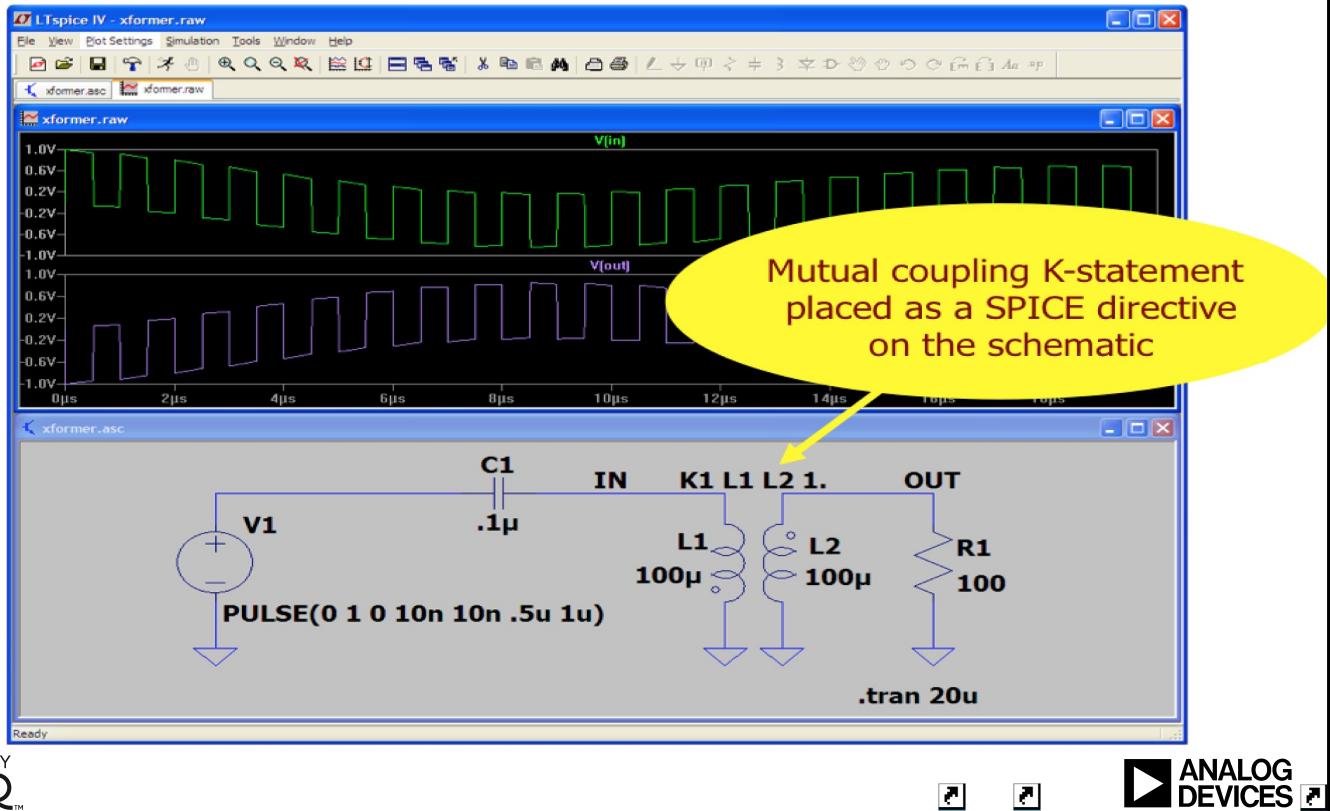
- Saturation flux density goes down monotonically with temperature
- Maximum service temperature plus self-heating
- Controller peak current production scatter
- Startup/transient/short circuit conditions



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



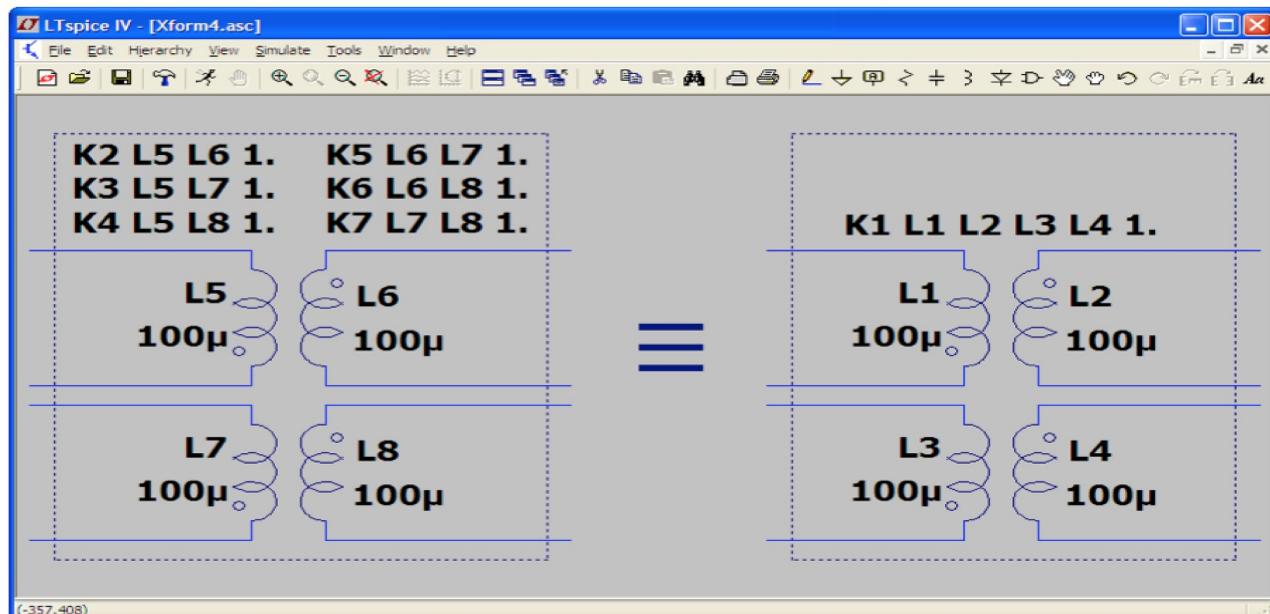
# Simulating Transformers



POWER BY  
LINEAR™

ANALOG  
DEVICES

## Multiple Windings



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

POWER BY  
LINEAR™

ANALOG  
DEVICES

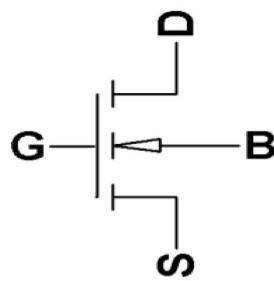
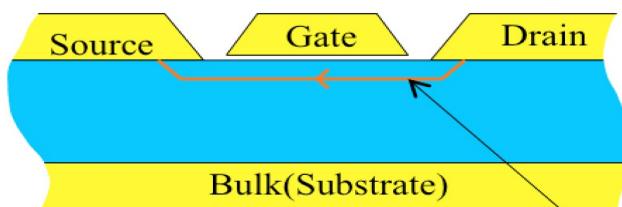
# LTspice's Special Enhancements for SMPS Simulation

- 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)

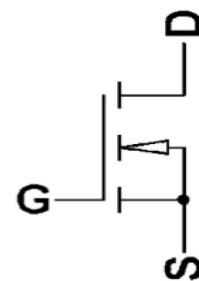
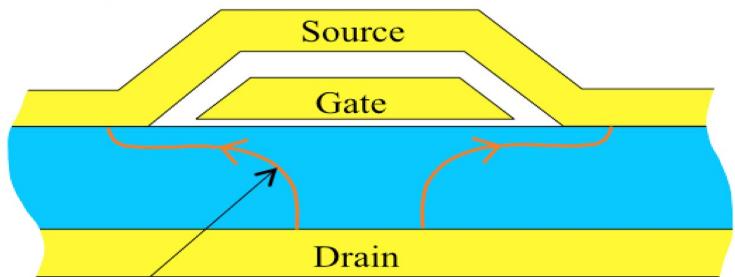


## VDMOS MOSFET

Normal Monolithic MOSFET  
(Used in IC's)

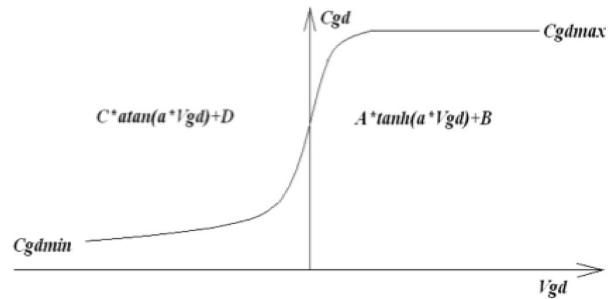
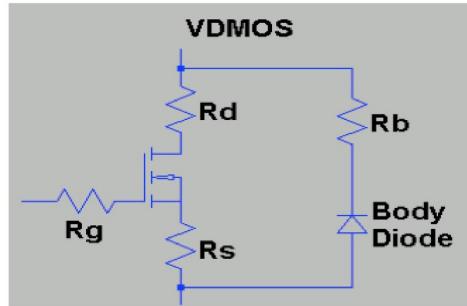


VDMOS  
(Discrete Power MOSFET)

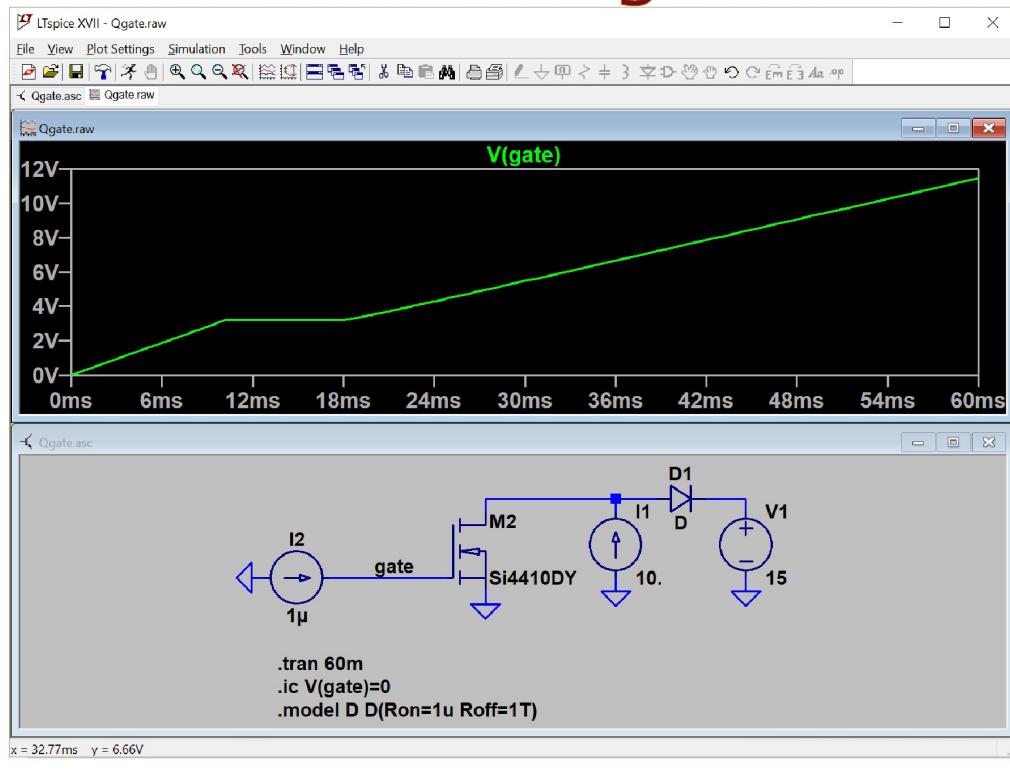


# LTC Proprietary VDMOS Model

Replace a problematic subcircuit with a single new intrinsic SPICE device



## VDMOS Gate Charge Behavior



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

To get the charge correct: 

The I-V curve got botched: 



## How Do I Add 3<sup>rd</sup> 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 Librарied Circuitry
  - Automatic symbol generation!



# 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

The screenshot shows the LTspiceHelp application window. The left pane is a tree view of help topics, and the right pane is the main content area.

**Circuit Description**

Circuits are defined by a text netlist. The netlist consists of a list of circuit elements and their nodes, model definitions, and other SPICE commands.

The netlist is usually graphically entered. To start a new schematic, select the File=>Open menu item. A windows file browser will appear. Either select an existing schematic and save it under a new name or type in a new name to create a new blank schematic file. LTspice uses many different types of files and documents. You will want to make a file with a file name extension of ".asc". The schematic capture commands are under the Edit menu. Keyboard shortcuts for the commands are listed under Schematic Editor Overview.

When you simulate a schematic, the netlist information is extracted from the schematic graphical information to a file with the same name as the schematic but with a file extension of ".net". LTspice reads in this netlist.

You can also open, simulate, and edit a text netlist generated either by hand or externally generated. Files with the extensions ".net", ".cir", or ".sp" are recognized by LTspice as netlists.

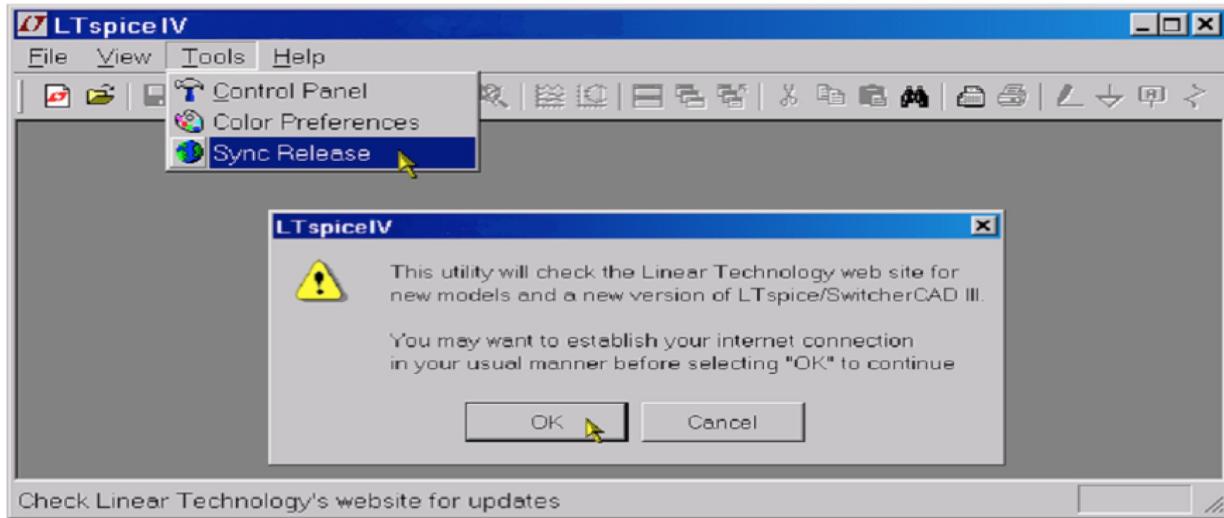
This section of the help documents the syntax used in netlists, but occasionally gives schematic-level advice.

[General Structure and Conventions](#)



# Updates With Field Sync

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



*Thanks for Listening!*

