UIC Department of Electrical and Computer Engineering
UNIVERSITY OF ILLINOIS AT CHICAGO
COLLEGE OF ENGINEERING

Introduction to LTspice

Dr. Vahe Caliskan

Department of Electrical and Computer Engineering

vahe@uic.edu

http://www.uic.edu/~vahe

Agenda

- Purpose of Presentation
- What is spice? What is LTspice? What can LTspice do for me?
- LTspice features / Running LTspice / User Interface
- LTspice Analyses
  - DC Operating Point and DC Sweep Analysis
  - Transient Analysis, Fourier, FFT
  - AC Small-Signal Analysis
  - Parameter Sweep Analysis
  - Monte-Carlo & Worst-Case Analysis
- Incorporating 3rd Party information (models, subcircuits)
- Implementing Hierarchies
- Questions / Comments

What is the purpose of this presentation?

- To introduce/familiarize you with LTspice (spice)
  - Many already familiar with PSpice w/Schematics or Multisim

- Demonstrate simulator capabilities that might be of use to your work
  - Even crude analyses in early design stages are beneficial
  - Laboratory pre-labs can be performed using a simulator
  - Most homework assignments can be validated using a simulator

- Open dialogue in the general area of modeling/simulation

- My purpose is NOT to sell/promote Linear Tech ICs
  - I do not own LTC stock ☹

What is spice?

1972 SPICE 1 (Simulation Program with Integrated Circuit Emphasis)

1975 SPICE 2 (L. Nagel's Ph.D. thesis is the user guide)

1983 SPICE 2G6

1985 SPICE 3

1993 SPICE 3F4

1996 µPower SwitcherCAD (simulation based)

1999 LTspice/SwitcherCAD III (SPICE based)

2004 500,000 base LTspice downloads

2009 LTspice IV (multi-processor), > 1 million downloads

What is LTspice?

- Rewrite of Berkeley SPICE 3F4/5
  - Reduced address calculation and function calling overhead
  - Improved timestep control and numerical methods
  - Enhanced integration methods and convergence improvements
  - Alternate solver/SPARSE matrix package 1000× more accurate
  - Circuit size limited by computer memory
- Added/Enhanced semiconductor models
  - Diode recombination current
  - JFET impact ionization current
  - BJT quasi-saturation
  - VBIC
  - Binned BSIM3v3.2.4, BSIM4.4.0, BSIMSOI3.2
  - VDMOS (a new vertical double diffused MOSFET for power MOS)
  - EKV 2.6

What can LTspice do for me?

- General circuit simulation with an unlimited, high-performance SPICE
  - Free download of the software is the full version (no restrictions)
  - No node limitations
  - Much fewer “convergence” issues
  - Already being used by many students at UIC
  - The price is right! Performance/cost ratio is ∞
  - It’s good, cheap and fast! ☺

- Simulate switch-mode power supply and associated circuitry
  - Model of nearly every IC manufactured by Linear
  - Example simulations for nearly every IC from Linear
    - These are normally the “typical application” shown page 1 of datasheet
    - This is the quickest way to get started with SMPS design

UIC UNIVERSITY OF ILLINOIS AT CHICAGO Department of Electrical and Computer Engineering COLLEGE OF ENGINEERING

LTspice Features

- General purpose schematic capture
  - Unlimited schematic size
  - Unlimited depth of hierarchy
  - Symbol editor
  - Complete documentation

- Integrated with Industry superlative SPICE simulator
  - Unlimited, professional-quality SPICE proven for IC design
  - Unmatched combination of robustness, accuracy, speed and compatibility
  - Advanced analysis/simulation options, parameter sweeps, FFTs, etc.
  - Run 3rd party models
  - Active independent users' groups (Yahoo groups)

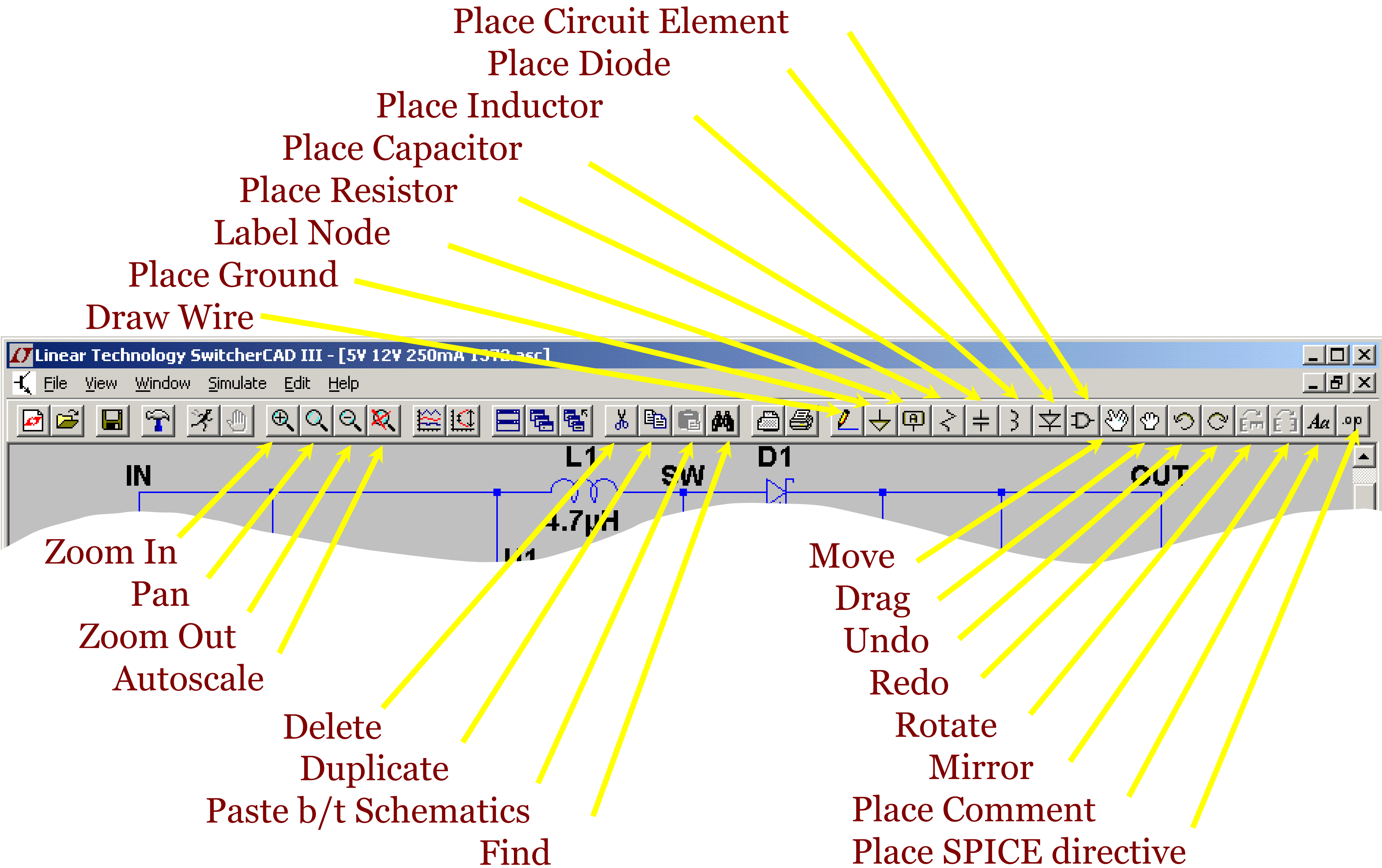
Two ways to run LTspice

- Schematic editor is usually used to enter the circuit information
  - Component selected from parts list
  - Wiring of circuit
  - Choosing type of analysis
  - Running simulation
  - Post-processing of results (viewing and analyzing waveforms)

- Direct simulation of netlist files and batch-mode simulation also available
  - Circuit is in the form of a netlist (text input, no schematic)
  - Running simulation, analysis and post-processing is identical

Demonstration

LTspice User Interface



UIC Department of Electrical and Computer Engineering
UNIVERSITY OF ILLINOIS AT CHICAGO
COLLEGE OF ENGINEERING

DC Operation Point Analysis (**`.op`**)

- Solve for system variables assuming equilibrium (**`d/dt`** $\rightarrow$ **`0`**)
  - Capacitors $\rightarrow$ open-circuit
  - Inductors $\rightarrow$ short-circuit

- Operating point results given in textual form
  - Node voltages may also be displayed after a **`.op`** run

- Newton-Raphson method is used for nonlinear systems

Demonstration

UIC UNIVERSITY OF ILLINOIS AT CHICAGO Department of Electrical and Computer Engineering COLLEGE OF ENGINEERING

Useful Features

- Frequently used parameters can be defined using a **`.param`** statement
  - **`.param fs=10kHz, Ts={1/fs}`**

- User-defined functions declared using the **`.func (.function)`** statement
  - **`.func myfunc(f1,f2) (f1+f2-f1*f2)`**

- Fourier Analysis using **`.four`** statement (used with **`.tran`**)
  - **`.four 2kHz v(out)`**

- FFT available directly from Waveform Viewer

Demonstration

UIC UNIVERSITY OF ILLINOIS AT CHICAGO Department of Electrical and Computer Engineering COLLEGE OF ENGINEERING

DC Sweep Analysis (`.dc`)

- Very similar to DC Operating Point Analysis
- Basically an **`.op`** analysis where one (or more) dc sources are varied
- Up to three sources may be “swept”
- DC Sweep results reported in plot
- Special dc sweep → **`.temp`**

Demonstration

UIC Department of Electrical and Computer Engineering
UNIVERSITY OF ILLINOIS AT CHICAGO
COLLEGE OF ENGINEERING

Transient Analysis (`.tran`)

- Nonlinear time-domain analysis

- Most utilized analysis and the one that produces most difficulties

- Simulation problems can often be resolved by …
  - Using more realistic (non-ideal) models
  - Addition of parasitic elements not on ideal schematic
  - Adjusting simulator settings (integration method, error tolerance, etc.)
  - Developing your own models
  - Not using PSpice ☺

- Fourier Analysis and FFT available after running transient analysis

Demonstration

AC Small-Signal Analysis (`.ac`)

- Simulation of linearized system to ac excitation
- Linearization around operating point from `.op` analysis
  - Actually an `.op` analysis is always run before any of the other analyses
- Linearized system simulation valid if excitation is "small"
- AC Analysis usually runs quite fast since models are already linearized

Demonstration

UIC Department of Electrical and Computer Engineering
UNIVERSITY OF ILLINOIS AT CHICAGO
COLLEGE OF ENGINEERING

Parameter Sweep Analysis (`.step`)

- Any parameter declared with a **`.param`** statement can be used with **`.step`**
- Using **`.step`** will run the chosen analysis (**`.tran`**, **`.ac`**, etc.) multiple times
- The actual stepping can be linear, logarithmic or given by a list

Demonstration

UIC University of Illinois at Chicago Department of Electrical and Computer Engineering
College of Engineering

Monte Carlo & Worst-Case Analysis

- Main idea is to run an analysis (**`.tran`**, **`.ac`**, etc.) multiple times with
  - Gaussian (normal) distribution
  - Worst-Case distribution
  - Analysis: **`.step param run 1 1000 1`** (**`run=1`** is nominal case)

- Custom functions make these analyses a bit easier
  - `.function normal(nom,tol) nom*(1+gauss(tol/3))`
  - `.function wc(nom,tol) if(flat(0.5)+0.5,nom*(1+tol),nom*(1-tol))`
  - `.function normal(nom,tol) if(run==1, nom, nom*(1+gauss(tol/3)))`
  - `.function wc(nom,tol) if(run==1, nom, if(flat(0.5)>0,nom*(1+tol),nom*(1-tol)))`

Demonstration

UIC University of Illinois at Chicago Department of Electrical and Computer Engineering
College of Engineering

Importing Models, Creating Symbols

- Multiple ways of importing models (`.model`) and subcircuits (`.subckt`)
  - Place model/subcircuit text directly on schematic
  - Place model/subcircuit text in files, then use (`.inc <file>`)

- Main parts databases can also be updated

- Creating a symbol is pretty easy
  - Use already existing symbols unless a special symbol is necessary

Demonstration

UIC University of Illinois at Chicago Department of Electrical and Computer Engineering
College of Engineering

Hierarchical Modeling

- Hierarchical schematic drafting has powerful advantages
- Larger circuits can be drafted while retaining clarity of smaller schematics
- Repeated circuitry to be easily handled in an abstract manner
- Blocks of circuitry can be stored in libraries for later use different projects

Demonstration

Model Parameter Stepping

- How do we handle stepping a device parameter? (e.g. BJT β)
- Be sure to understand **`all`** device parameters before doing this
- Stepping can easily be done by setting device parameter using **`.param`**
- Then simply use **`.step`** with you choice of analysis (e.g. MC, WC)

Demonstration

UIC UNIVERSITY OF ILLINOIS AT CHICAGO Department of Electrical and Computer Engineering COLLEGE OF ENGINEERING

BJT dc Model

- Custom BJT hierarchical model used for dc analysis
- Input parameters: β_F, β_R, $V_{BE,on}$, $V_{BC,on}$
- Model automatically determines mode
  - cutoff, saturation, forward active, reverse active
- Works great on ECE 340 homework problems ☺

Demonstration

MOSFET dc Model

- Custom MOSFET hierarchical model used for dc analysis
- Input parameters: K and V_t
- Model also determines mode (cutoff, ohmic, saturation)
- Works great on ECE 340 homework problems ☺

Demonstration

Filter Analysis / Bode Plots

- Circuits based on op-amp (ideal or non-ideal)
- Pure s-domain expressions are also allowed
- Custom blocks can determine asymptotic Bode plots
- Works great on ECE 342 / ECE 412 homework problems

Demonstration

UIC University of Illinois at Chicago Department of Electrical and Computer Engineering
College of Engineering

Step Response, Laplace Transforms, FFT

- ECE 310/350 (step response, Laplace transforms, etc.)
- ECE 311/411 (FFT)

Demonstration

LTspice Resources

- LTspice (free download)
  - http://ltspice.linear.com/software/LTspiceIV.exe
- LTspice Getting Started Guide
  - http://ltspice.linear.com/software/LTspiceGettingStartedGuide.pdf
- LTspice User's Guide
  - http://ltspice.linear.com/software/scad3.pdf
- LTspice Users' Group (free — registration required)
  - http://tech.groups.yahoo.com/group/LTspice/
- LTspice Users' Group Documentation (free — registration required)
  - http://tech.groups.yahoo.com/group/LTspiceDocs/