LTSpice IV

EDL Spring 2016

LTSpice IV – Importance

- **SPICE simulation of circuits (BEFORE PHYSICALLY BUILDING THE CIRCUIT!!)**
  - **Test integrity of circuits**
  - **Predict circuit behavior**

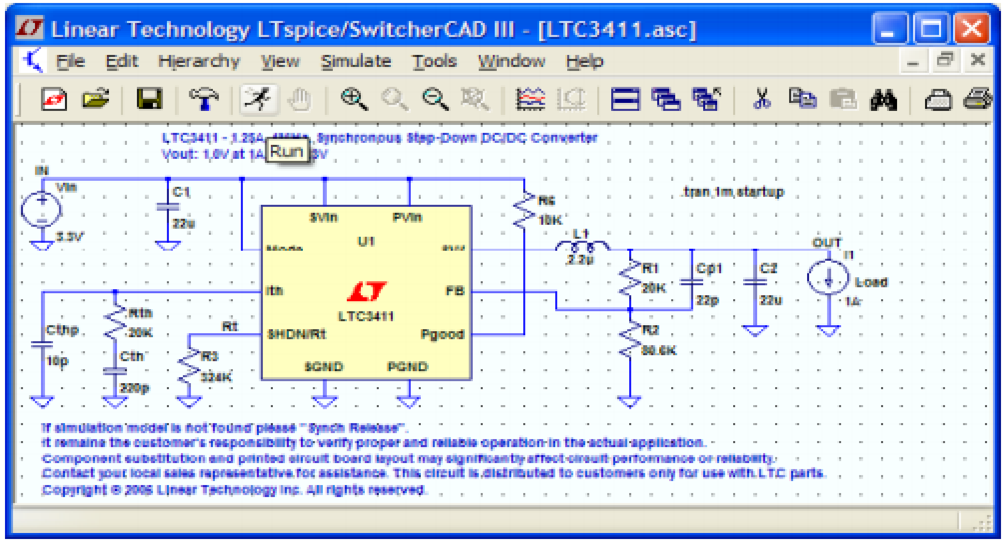
- Schematic and symbol editor

- Library of passive elements (R,L,C components)

- Library of LT (Linear Technology) components
  - Macromodels of these components

- Simulate and view waveforms

- Transient, AC and DC/Stead-state analysis, DC Operating points



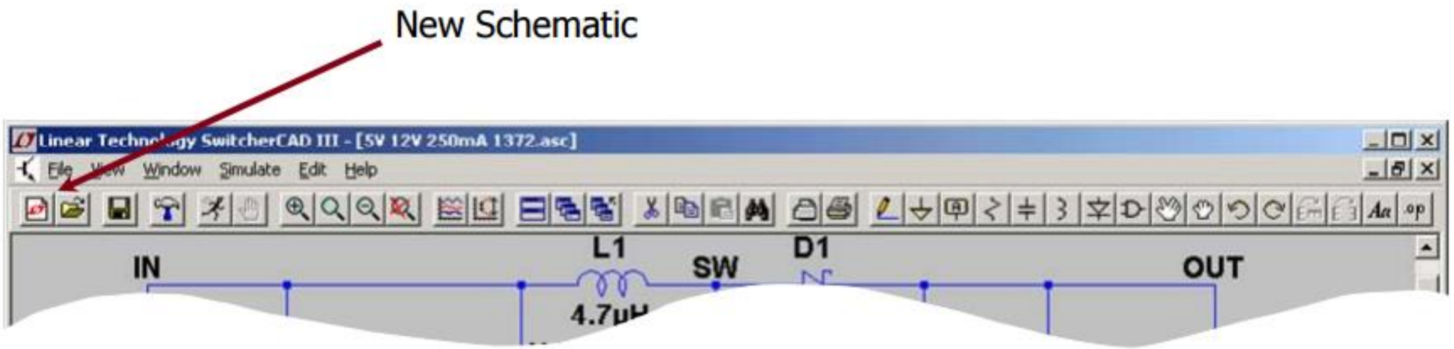
LTSpice IV – Download/Install

- LTSpice IV (Free)
- http://www.linear.com/LTspice
- For Windows or Mac

GETTING STARTED

Getting Started – New Schematic

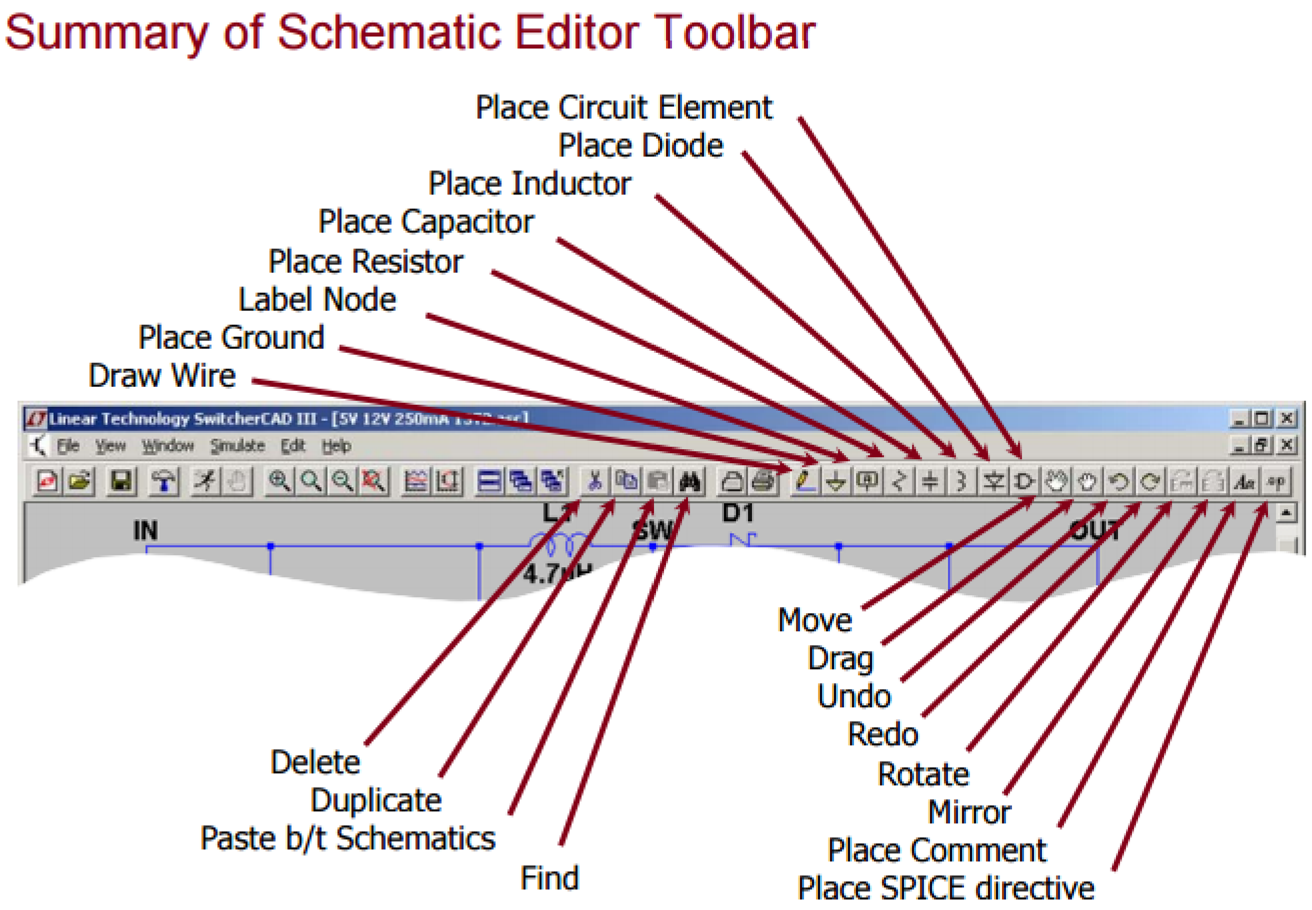
Start with a New Schematic



- Left click on the **New Schematic** symbol in the Schematic Editor Toolbar

LTspice is also a great schematic capture

Getting Started - Toolbar



LTSpice IV - Hotkeys

LTspice HotKeys

	Schematic	Symbol	Waveform	Netlist
Modes	ESC - Exit Mode	ESC - Exit Mode		
	F3 – Draw Wire			
	F5 – Delete	F5 – Delete	F5 – Delete	
	F6 – Duplicate	F6 – Duplicate		
	F7 – Move	F7 – Move		
	F8 – Drag	F8 – Drag		
	F9 – Undo	F9 – Undo	F9 – Undo	F9 – Undo
	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo
View	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	
	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	
	Space – Zoom Fit		Ctrl+E – Zoom Extents	
	Ctrl+G – Toggle Grid		Ctrl+G – Toggle Grid	Ctrl+G – Goto Line #
	U – Mark Unncon. Pins	Ctrl+W – Attribute Window	'0' - Clear	
	A – Mark Text Anchors	Ctrl+A – Attribute Editor	Ctrl+A – Add Trace	
	Alt+Click - Power		Ctrl+Y – Vertical Autorange	Ctrl+R – Run Simulation
	Ctrl+Click - Attr. Edit		Ctrl+Click - Average	
	Ctrl+H – Halt Simulation		Ctrl+H – Halt Simulation	Ctrl+H – Halt Simulation
Place	R – Resistor	R – Rectangle		
	C – Capacitor	C – Circle		
	L – Inductor	L – Line		
	D – Diode	A – Arc		
	G – GND			
	S – Spice Directive			
	T – Text	T – Text		
	F2 – Component			
	F4 – Label Net			
	Ctrl+E – Mirror	Ctrl+E – Mirror		
	Ctrl+R – Rotate	Ctrl+R – Rotate		

Command Line Switches

Flag	Short Description
-ascii	Use ASCII .raw files. (Degrades performance!)
-b	Run in batch mode.
-big or -max	Start as a maximized window.
-encrypt	Encrypt a model library.
-FastAccess	Convert a binary .raw file to Fast Access Format.
-netlist	Convert a schematic to a netlist.
-nowine	Prevent use of WINE(Linux) workarounds.
-PCBnetlist	Convert a schematic to a PCB netlist.
-registry	Store user preferences in the registry.
-Run	Start simulating the schematic on open.
-SOI	Allow MOSFET's to have up to 7 nodes in subcircuit.
-uninstall	Executes one step of the uninstallation process.
-wine	Force use of WINE(Linux) workarounds.

Simulator Directives - Dot Commands

Command	Short Description
.AC	Perform a Small Signal AC Analysis
.BACKANNO	Annotate the Subcircuit Pin Names on Port currents
.DC	Perform a DC Source Sweep Analysis
.END	End of Netlist
.ENDS	End of Subcircuit Definition
.FOUR	Compute a Fourier Component
.FUNC	User Defined Functions
.FERRET	Download a File Given the URL
.GLOBAL	Declare Global Nodes
.IC	Set Initial Conditions
.INCLUDE	Include another File
.LIB	Include a Library
.LOADBIAS	Load a Previously Solved DC Solution
.MEASURE	Evaluate User-Defined Electrical Quantities
.MODEL	Define a SPICE Model
.NET	Compute Network Parameters in a .AC Analysis
.NODESET	Supply Hints for Initial DC Solution
.NOISE	Perform a Noise Analysis
.OP	Find the DC Operating Point
.OPTIONS	Set Simulator Options
.PARAM	User-Defined Parameters
.SAVE	Limit the Quantity of Saved Data
.SAVEBIAS	Save Operating Point to Disk
.STEP	Parameter Sweeps
.SUBCKT	Define a Subcircuit
.TEMP	Temperature Sweeps
.TF	Find the DC Small-Signal Transfer Function
.TRAN	Do a Nonlinear Transient Analysis
.WAVE	Write Selected Nodes to a .WAV file

Suffix		Suffix	
		f	1e-15
T	1e12	p	1e-12
G	1e9	n	1e-9
Meg	1e6	u	1e-6
K	1e3	M	1e-3
		Mil	25.4e-6

Constants	
E	2.7182818284590452354
Pi	3.14159265358979323846
K	1.3806503e-23
Q	1.602176462e-19
TRUE	1
FALSE	0

LTspice IV

See Demo

LTSpice IV – Specifying Units

Use Labels to Specify Units in Circuit Elements Attributes

- **K** = k = kilo = 10^3
- **MEG** = meg = 10^6
- **G** = g = giga = 10^9
- **T** = t = terra = 10^{12}

- **m** = M = milli = 10^{-3}
- **u** = U = micro = 10^{-6}
- **n** = N = nano = 10^{-9}
- **p** = P = pico = 10^{-12}
- **f** = F = femto = 10^{-15}

Important

- Use ***MEG*** to specify 10^6, not *M*
- Enter ***1*** for 1 Farad, not *1F*

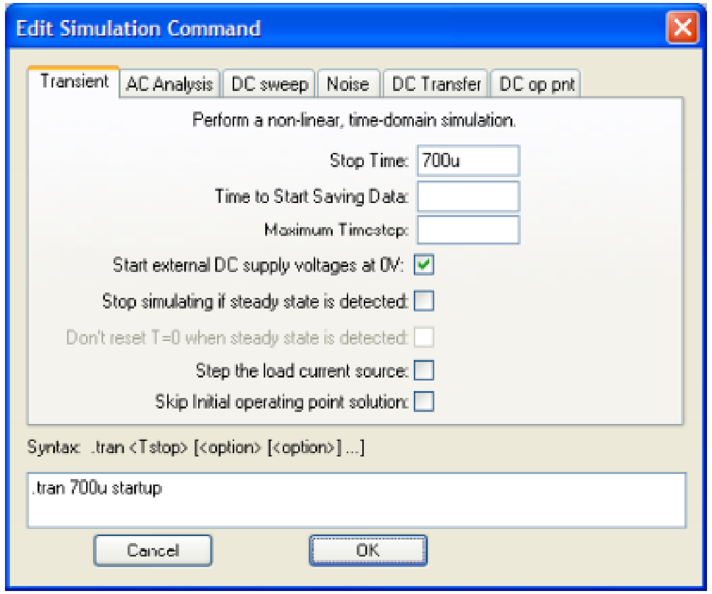
Running Simulations

Setup Simulations (i.e. Transient simulation)

Editing Simulation Commands

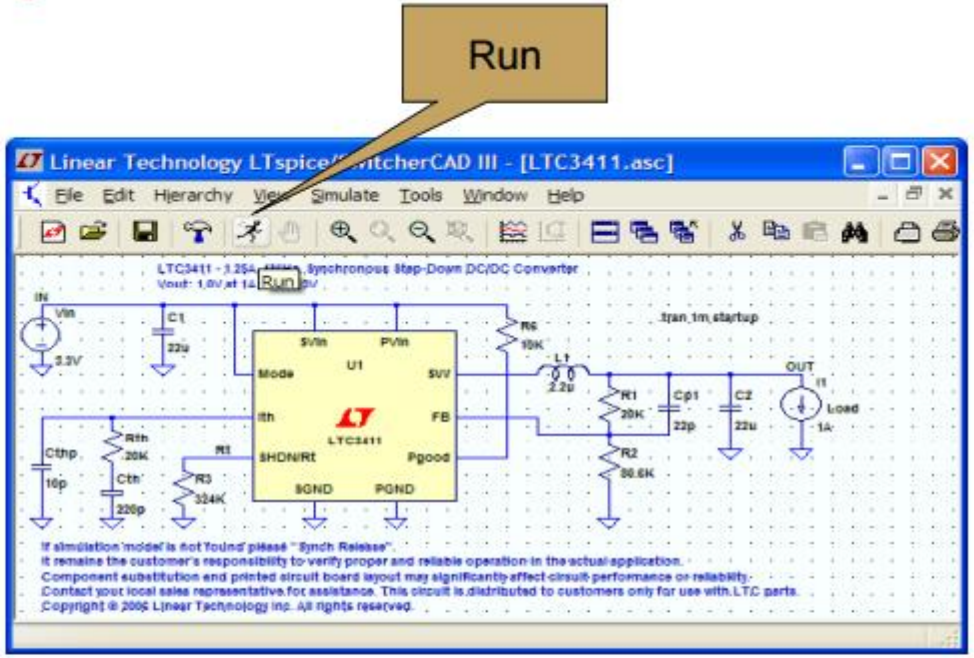
- Left click on **Simulation** menu
- Left click on **Edit Simulation Cmd**
- As a starting point in a simulation
  - Left click on **Transient** tab
  - Enter a ***Stop Time***
    - You may need to adjust this again later
- Select **OK**

Demo Circuits and **Test Fixtures** have predefined Simulations Commands



Running Simulations

Running a Circuit



If model is not found please Sync Release under Help menu to update LTspice

Probing Circuit (to view waveforms)

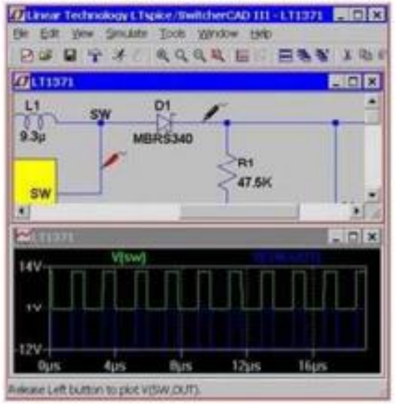
Probing a Circuit & Waveform Viewer

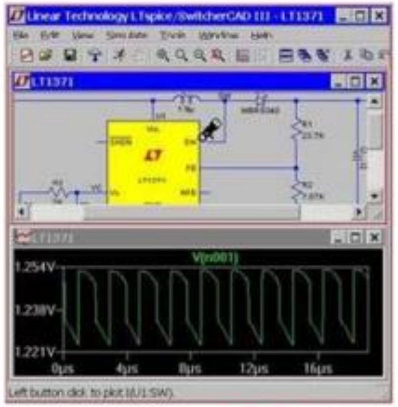
- Left click on any wire to plot the voltage on the waveform viewer

Voltage probe cursor

- Left click on the body of the component to plot the current on the waveform viewer
  - Convention of positive current is in the direction into the pin

Current probe cursor



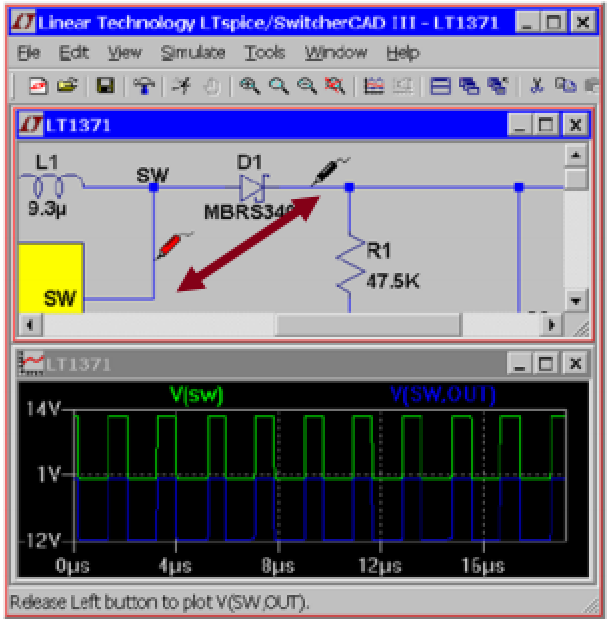


Viewing Voltage Across Two Different Nodes

Voltage Differences Across Nodes

- Left click and hold on one node and drag the mouse to another node
  - Red voltage probe at the first node
  - Black probe on the second

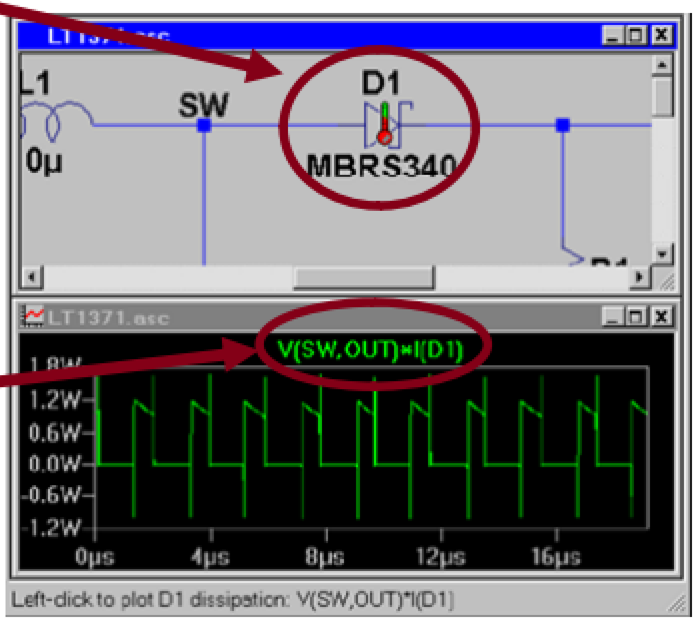
Differential voltages are displayed in the waveform viewer



Viewing Power Dissipation

Instantaneous & Average Power Dissipation

- Instantaneous Power Dissipation
  - Hold down the ALT key and left click on the symbol of the component
  - Pointer will change to a thermometer
  - Plotted in units of Watts
- Average Power Dissipation
  - Hold down the Ctrl key and left click on the ***trace label*** power dissipation waveform



LTSpice IV Helpful Links

- **Download:** http://www.linear.com/designtools/software/#LTspice
- **Getting-Started Tutorial:** http://cds.linear.com/docs/en/software-and-simulation/LTspiceGettingStartedGuide.pdf
- **Simulating - AC Analysis:** http://www.linear.com/solutions/4581
- **Simulating - DC/Transient Analysis:** http://www.seas.upenn.edu/~jan/LTspice/ESE216LTSpiceDC&TransientSimulations.pdf
- **Simulating - DC Operating Points:** http://www.zen22142.zen.co.uk/ltspice/dccircuits.htm
- **Importing/Exporting PWL (Piecewise Linear) Signals:** http://www.linear.com/solutions/1815
- **Adding Third-Party Models:** http://www.linear.com/solutions/1083
- **Importing/Exporting WAVE file (audio signals):** http://electrostud.wikia.com/wiki/Using_WAVE_files_as_input/output_in_LTSpice
- **LTSpice for Mac:** https://www.youtube.com/watch?v=gdRqZwrrXwU