Chipsmall Limited consists of a professional team with an average of over 10 year of expertise in the distribution of electronic components. Based in Hongkong, we have already established firm and mutual-benefit business relationships with customers from,Europe,America and south Asia,supplying obsolete and hard-to-find components to meet their specific needs.

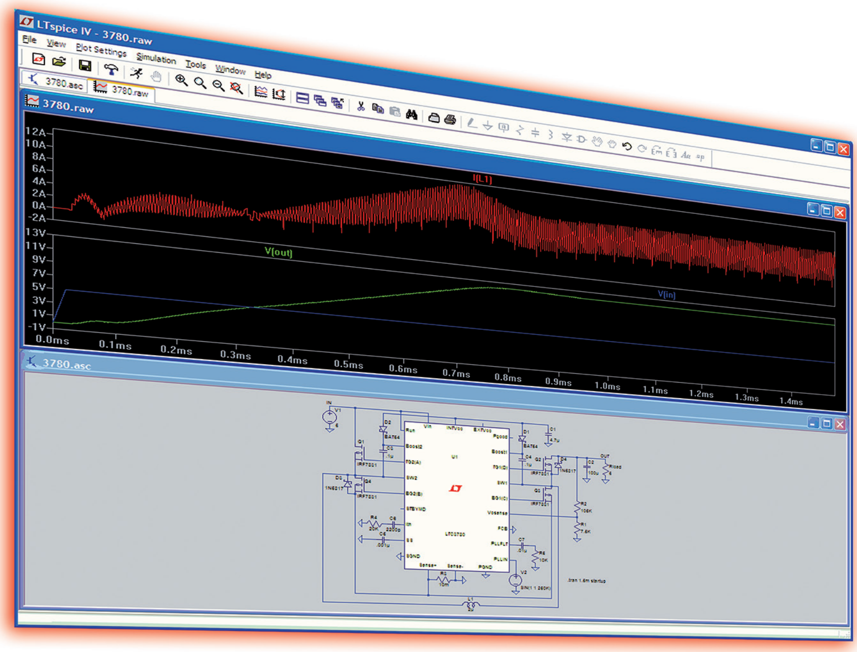
With the principle of “Quality Parts,Customers Priority,Honest Operation,and Considerate Service”,our business mainly focus on the distribution of electronic components. Line cards we deal with include Microchip,ALPS,ROHM,Xilinx,Pulse,ON,Everlight and Freescale. Main products comprise IC,Modules,Potentiometer,IC Socket,Relay,Connector.Our parts cover such applications as commercial,industrial, and automotives areas.

We are looking forward to setting up business relationship with you and hope to provide you with the best service and solution. Let us make a better world for our industry!

Contact us

Tel: +86-755-8981 8866 Fax: +86-755-8427 6832
Email & Skype: info@chipsmall.com Web: www.chipsmall.com
Address: A1208, Overseas Decoration Building, #122 Zhenhua RD., Futian, Shenzhen, China

LTspice



- Free Analog Circuit Simulator
- Unlimited Nodes/Nets
- Fast Simulations

LTspice® is a high performance SPICE simulator, schematic capture and waveform viewer designed to speed the process of power supply design. LTspice adds enhancements and models to SPICE, significantly reducing simulation time compared to typical SPICE simulators, allowing one to view waveforms for most switching regulators in minutes compared to hours for other SPICE simulators.

twitter

www.twitter.com/ltspice
Follow @ LTspice on Twitter for up-to-date information on models, demo circuits, events and user tips.

FREE LTspice® Download

www.linear.com/ltspice
Included in the download is a complete and fully functional SPICE program, help files, macro models for Linear Technology's power products, over 200 op amp models, as well as models for resistors, transistors and MOSFETs.

video.linear.com/ltspice
View Instructional Videos.

Sign up for the Linear Insider email newsletter to receive LTspice updates @ www.linear.com/mylinear

See Demo

LINEAR TECHNOLOGY

NOW PART OF ANALOG DEVICES

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	
	Atl+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

See Demo

www.linear.com/LTspice ▪ 1-800-4-LINEAR

LINEAR TECHNOLOGY NOW PART OF ANALOG DEVICES

0417C