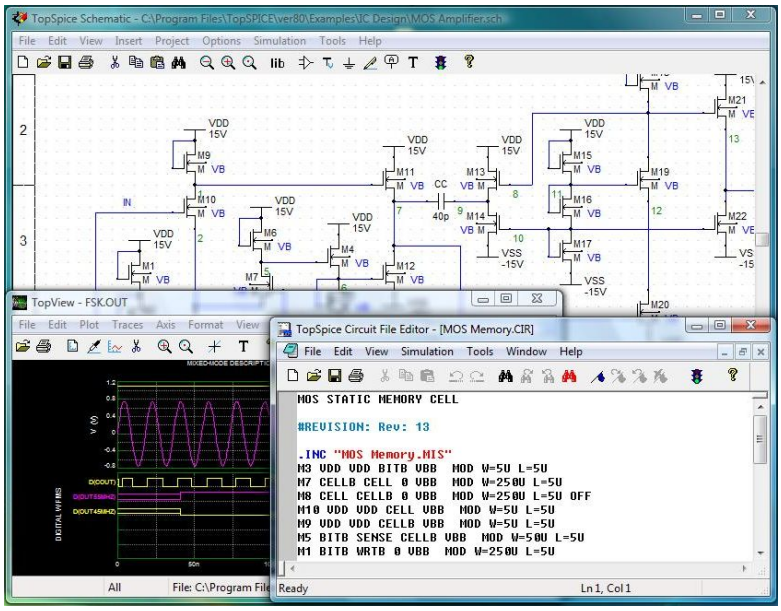
New Release

TopSpice 8

Version 8.6

Mixed-Mode Circuit Simulation



A powerful and affordable circuit design software package offering advanced native mixed-mode mixed-signal analog/ digital simulation based on the industry standard SPICE program

TopSpice is an analog and mixed-mode SPICE circuit simulation tool for PCs. It offers the most advanced simulator in its price range, industry compatibility, and a quick and easy to use integrated design environment from schematic capture to graphical waveform analysis. Its flexible working framework even allows easy integration of schematic editors, model libraries and tools from other vendors.

TopSpice has been proven by designers at leading electronic firms worldwide since its first release in 1991. TopSpice is a professional grade tool with emphasis on features to solve the needs of experienced designers. TopSpice will take your circuit design from conception to realization with ease and efficiency.

TopSpice gives you all the powerful professional features you need at a fraction of the cost:

- ✓ **Complete, integrated and efficient EDA system with schematic capture, simulator, graphical post-processor, circuit file editor with SPICE syntax color highlighting and Model Database Tool.**
- ✓ **Robust industry compatible SPICE simulator supports unlimited circuit sizes[1], analog behavioral modeling, and native mixed-mode mixed-signal simulation with built-in event-driven logic simulator.**
- ✓ **Supports PSpice™ and HSPICE™ compatible syntax extensions, HSPICE™ IC foundry model libraries, MOSFET model binning, BSIM3 level 49 model, diode level 3, VBIC BJT level 4.**
- ✓ **Schematic Editor features support for hierarchical designs, multiple sheets, auto symbol generator, wire rubber-banding and automatic part numbering.**
- ✓ **Interactive post-processor with smart auto plotting, FFT, Smith charts, histograms, waveform expressions, performance measurements, CSDF data file plotting and other powerful analysis functions.**
- ✓ **Extensive model libraries of over 30,000 analog and digital parts.**

[1] Limited by available system resources.

SPICE SIMULATION

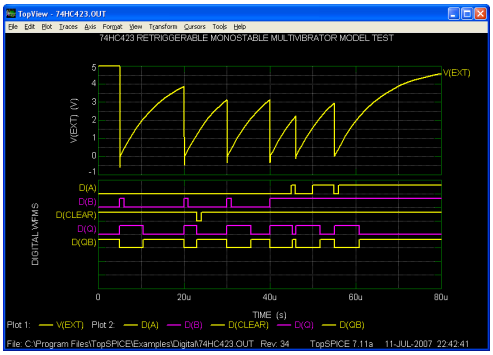
SPICE has become the de facto industry standard for analog circuit simulation. It offers the advantages of a large user base, large number of existing device models, modeling support by component manufacturers, and compatibility between vendors. TopSpice offers a native full-featured mixed-mode mixed-signal SPICE circuit simulator. It is capable of simulating circuits containing any combination of analog devices, digital functions and high-level behavioral blocks.

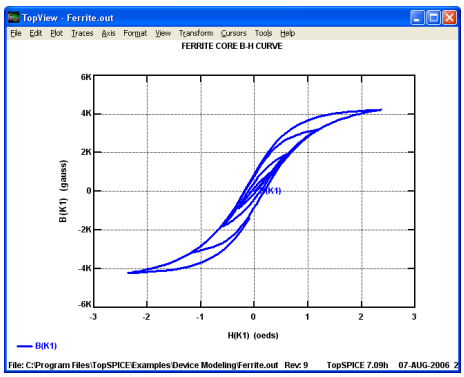
```
*Behavioral model of comparator
ECOMP 1 0 TABLE {V(in)} = (-1mV,0.2V) (1mV,3.6V)
RES 1 0 150    ;source load

*Transmission line used as a delay line
TDLY 1 0  delay 0  (ZO=150 TD=20ns)
RTL delay 0 150    ;termination

*Digital D flip-flop element
U1 DFF outlo outhi $D_HI $D_HI in delay FFD
.MODEL FFD UEFF (TPCLKLH=2ns TPCLKHL=2ns)
```

TopSpice mixed-mode netlist of a FSK demodulator.

State-of-Art Features

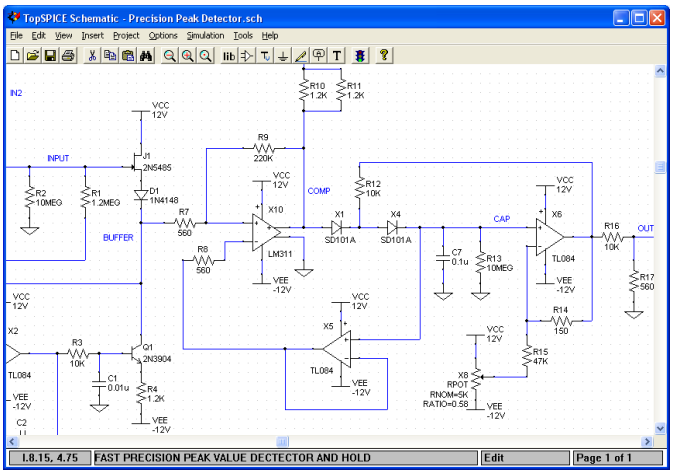
- Native mixed-mode, mixed-signal simulation with built-in event-driven logic simulator.
- Industry standard SPICE simulator supports many PSpice™ and HSPICE™ syntax enhancements.
- Unlimited circuit sizes (system resource limited).
- Multiple/parallel simulation runs for same circuit.
- BSIM3 and EKV submicron MOSFET models with HSPICE™ compatible ACM and binning support.
- "Curtice", "Statz" and Triquint GaAs FET models.
- Jiles-Atherton nonlinear magnetic core model.
- Table look-up modeling of capacitors and inductors.
- Built-in advanced ferroelectric capacitor model.
- Lossy transmission line model.
- Analog behavioral modeling using arbitrary equations, logical expressions and look-up tables.
- Frequency domain modeling using Laplace transforms, frequency response and s-parameter tables.
- Advanced noise analysis features.
- Macromodeling (subcircuits) with parameter passing.
- User defined parameters and parametric expressions.
- Automatic model library search.
- Support for HSPICE™ IC foundry model library files.
- Advanced convergence aid algorithms solve tough convergence problems.
- Monte Carlo and worst case analyses allow statistical analysis based on device and lot tolerances.
- Parametric stepping of component, user defined parameter and tolerance values.
- Non-uniform device temperatures.
- "ALTER" iteration command supports all devices, models and subcircuits.
- "Pause-plot-resume" simulation capability to view partial run results.
- Output of device current, source impedance, magnetic flux and digital state values.
- Binary file output option saves all voltages, device currents and digital node states.
- Supports encrypted model files.





Native Mixed-Mode Simulation

State-of-art mixed technology circuit designs require a hierarchical top-down design environment with a circuit simulator capable of handling different levels of abstraction simultaneously. TopSpice implements a fully integrated mixed-mode simulation technology. The analog electrical simulation and the logic simulation algorithms are merged into one simulator architecture. The logic simulator implemented in TopSpice is fully event-driven. The digital section of the circuit is only



evaluated when a logic event takes place during the simulation.

TopSpice also implements a common SPICE like input syntax for both analog and digital sections for getting those designers familiar with SPICE rapidly on board. By using the built-in logic simulator to simulate the digital sections of your circuit instead of analog equivalents, mixed-mode simulation times can be reduced by orders of magnitude.

Analog Behavioral Modeling

The analog behavioral modeling capability allows the designer to describe electronic components and circuit functional blocks using transfer functions. This "black box" approach is a very powerful tool for modeling complex devices, speeding up simulation of large systems and developing a top-down design methodology. The following options for specifying transfer functions are available in TopSpice:

- Arbitrary equations of V, I, time and temperature; supports math functions, user parameters, logical and relational operators
- Look-up tables
- Laplace transforms
- Frequency response tables
- S-parameter tables

INTEGRATED DESIGN ENVIRONMENT

TopSpice offers a fully integrated environment to capture, simulate, and analyze your circuit designs. Its flexible architecture allows the designer to integrate all the design tools, including third party tools and model libraries, into a complete EDA system.

With TopSpice you have the choice to design from schematic drawings or text SPICE netlist files, or switch between them. All design and simulation functions are available from either the schematic or netlist editor front-ends.

Schematic Capture

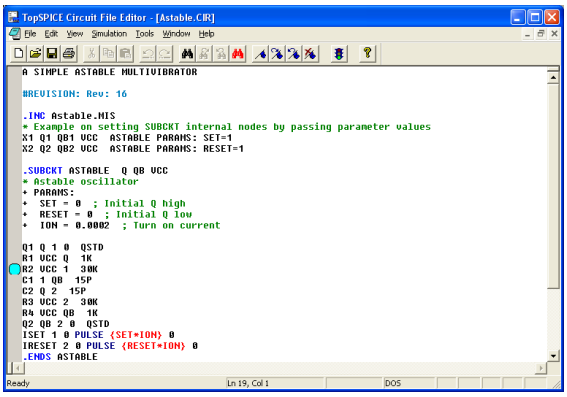
The Schematic Editor front-end provides an easy way to create and edit circuit schematic drawings. It features:

- Multiple schematic pages (up to 100).
- Support for hierarchical designs.
- Powerful, general, symbol attribute handling.
- Auto-repeat, auto-junction, wire rubberbanding and auto-incrementing of part names.
- Auto symbol generator for subcircuits and logic functions.
- Group, area and subcircuit edit operations.
- Bias voltage and current display on schematic drawing.
- Menu commands integrate all the simulation setup, run and plot functions.
- Over 400 symbols for all SPICE elements and model library parts, plus many commonly used analog and digital devices.

Circuit File Editor

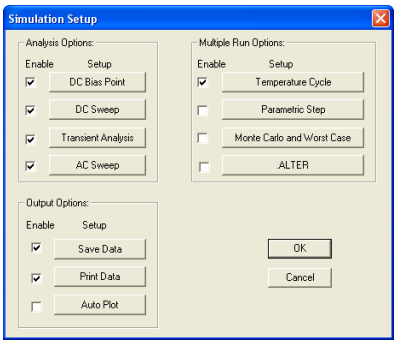
The Circuit File Editor is the text editor front-end for TopSpice. It allows you to create, edit and simulate SPICE netlists. It can also be used to edit command and model files, or any other text file. It operates much like any other Windows text editor, but with added features that makes it convenient to perform all the simulation functions:

- Color highlighting for SPICE syntax.
- Menu commands integrate all the simulation setup, run and plot functions.
- Common Simulation Setup menu allows switching between schematic and circuit file.
- Model library database function.
- Toolbar “Go” icon runs simulation with a single click.
- “Bookmarking” feature.
- Online simulator syntax and command help.



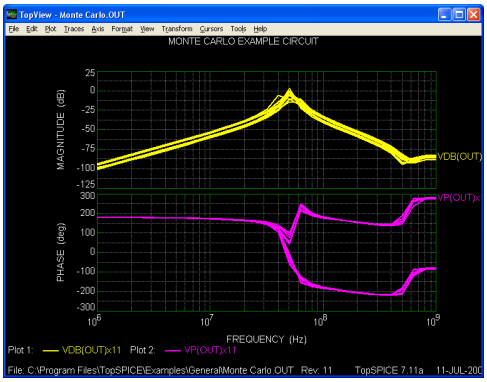
Easy Analysis Setup

TopSpice is easy to use for both beginners and experts. All simulation analysis commands can be specified by selecting menu and dialog options. Expert SPICE users also have the option of entering commands in text form directly on the schematic drawing or a separate command file.

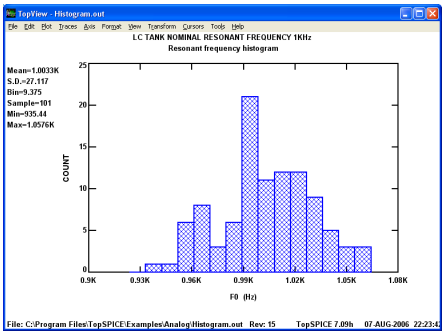


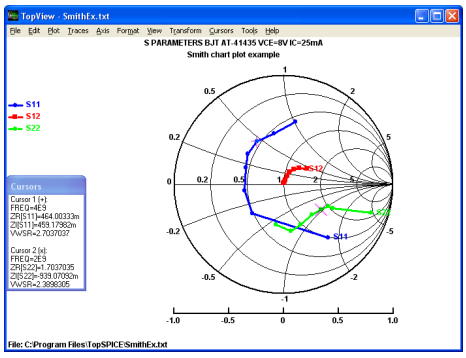
PLOTTING AND WAVEFORM ANALYSIS

TopView is an interactive SPICE post-processor program to plot and analyze SPICE simulation output data. TopView provides many powerful features:



- Smart auto-plotting.
- Multiple plot axes.
- Zooming and scrolling of plots.
- Cursor measurements.
- Waveform expression plots.
- Powerful waveform analysis functions.
- Statistical and performance measurements.
- FFT, Smith chart, polar chart, histogram and “eye display” plots.
- Complex-number expressions and plots.
- Support for importing and plotting SPICE2, SPICE3, HSPICE, CSDF, Touchstone s-parameter, .WAV audio and user data files.
- Many formatting options for creating custom graphs.
- Options to export graphs to other Windows applications.





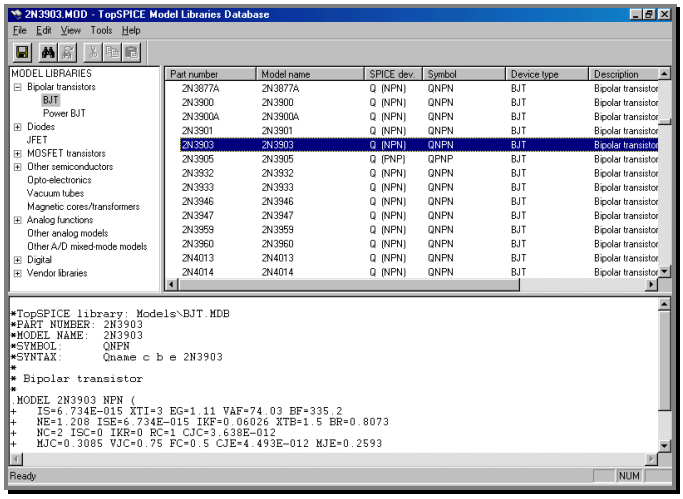
- Publication quality printed output.

MODEL LIBRARIES

The TopSpice software package comes with an extensive set of model libraries with over 30,000 parts. They include discrete devices (diodes, transistors, power semiconductors, etc.), analog ICs and functions, optoelectronic and RF devices, magnetic parts, digital logic families (TTL, CMOS and ECL), SMPS design models, and vendor model libraries.

The Model Database tool allows convenient searching, browsing and extraction of models in the libraries. The simulator also automatically searches the model database when a circuit uses undefined device models and extracts the necessary model codes from the libraries.

TopSpice also supports standard SPICE library files, HSPICE™ IC foundry library files and most model files. These are available from component vendors, EDA vendors, IC foundries, other third parties, Internet, or they can be user created.



TopSpice simulator built-in devices, models, functions and commands:

Circuit Elements

B	GaAs FET (same as J)
C	capacitor
	POLY nonlinear capacitor
	PWL piece-wise linear capacitor
	TABLE capacitance vs. voltage table
	FECAP ferroelectric capacitor
D	diode
E	voltage-controlled voltage source (linear, POLY, ADD, DIVIDE, MULTIPLY, POWER, ABM options)
F	current-controlled current source (linear, POLY)
G	voltage-controlled current source (linear, POLY, POWER, ABM options)
H	current-controlled voltage source (linear, POLY)
I	independent current source
J	JFET, GASFET and MESFET
K	inductive coupling and magnetic core
L	inductor
	POLY nonlinear inductor
	PWL piece-wise linear inductor
M	MOSFET
O	analog/digital interface
Q	bipolar transistor (BJT)
R	resistor
S	switch
T	transmission line (ideal and lossy)
U	digital device
V	independent voltage source
X	subcircuit (macromodel)
Z	MESFET (same as J)

Built-in Device Models

ATOD	analog/digital interface
C	integrated capacitor
CAP	capacitor model
CORE	nonlinear magnetic core
D	diode (levels 1 and 3)
DTOA	digital/analog interface
FECAP	ferroelectric capacitor
GASFET	GaAs FET (levels 1, 2, 3, 6)
IND	inductor model
LTRA	lossy transmission line
NJF	n-channel JFET
NMF	n-channel MESFET
NMOS	n-channel MOSFET (levels 1, 2, 3, 7, 8, 44, 49, 53, 55)
NPN	npn BJT (levels1 and 4)
PJF	p-channel JFET
PMF	p-channel MESFET
PMOS	p-channel MOSFET (levels 1, 2, 3, 7, 8, 44, 49, 53, 55)
PNP	pnp BJT (levels 1 and 4)
R	integrated resistor
RES	resistor model
TRN	lossy transmission line
U3GATE	tri-state gates
UALU	ALU functions
UCOUNT	counters
UEFF	edge-triggered flip-flops
UGATE	boolean gates, delay, buffers
UGFF	gated flip-flops
UIO	analog/digital interface
UPULSE	digital pulse generator
URAM	RAM memory
UROM	ROM memory
USEL	decoders and encoders
USREG	shift registers
USTIM	digital stimulus generators
VSWITCH	voltage-controlled switch

Signal Sources

AM	amplitude modulated
EXP	exponential pulse
FILE	user data file
NOISE	noise generator
PULSE	pulsed waveform
PWL	piece-wise linear (table driven)
RANDOM	random waveform
SFAM	single frequency AM waveform
SFFM	single frequency FM waveform
SIN	sine wave

Analog Behavioral Modeling

Options:

FREQ	frequency response and s-parameter tables
LAPLACE	Laplace transform
TABLE	look-up table
VALUE	arbitrary transfer function using math, logical or relational expression

Predefined Math Functions:

ABS	absolute value
ACOS	arc cosine
ASIN	arc sine
ARCTAN	arc tangent
ATAN	arc tangent
ACOSH	arc hyperbolic cosine
COS	cosine
EXP	exponential
IF	IF-THEN-ELSE logical evaluation
LIMIT	limit value
LN	logarithm base e
LOG10	logarithm base 10
LOG	logarithm base e
MAX	maximum of *x* and *y*
MIN	minimum of *x* and *y*
POW10	power of 10
PWR	power
PWRS	signed power
ROUND	round off to nearest integer
SGN	sign
SIGN	sign (same as SGN)
SINH	hyperbolic sine
SIN	sine
SQRT	square root
SQR	square
STP	step function
TABLE	table look up
TANH	hyperbolic tangent
TAN	tangent
TRUNC	truncate to integer value
U	step function
URAMP	ramp function

Control Statements

.AC	AC analysis (frequency sweep)
.ALTER	alter circuit
.DC	DC sweep
.DISTO	distortion
.END	end of circuit
.ENDL	end library entry section
.ENDM	end macro
.ENDS	end subcircuit
.FOUR	Fourier analysis
.FUNC	function definition
.GLOBAL	global node
.HSPICE$	select HSPICE™ syntax
.IC	initial condition
.INCLUDE	include file
.LIB	library file or library entry
.LOADBIAS	load initial condition data from bias data file
.MACRO	start macro definition
.MC	Monte Carlo analysis
.MODEL	device model definition
.NODESET	set node initial guess
.NOISE	noise analysis
.OP	operating point information
.OPTIONS	run time options
.PARAM	define parameters
.PLOT	“line printer” plots
.PRINT	print data to OUT file
.PROBE	save data in binary forma
.PROTECT	protect netlist
.RENUMBER	renumber reference names
.SAVE	save data in binary format
.SAVEBIAS	save bias data to file
.SENS	sensitivity analysis
.STAT	statistical distribution
.STEP	step circuit parameter
.SUBCKT	start subcircuit definition
.TEMP	set temperature
.TF	transfer function
.TRAN	transient analysis
.WATCH	same as #AUTOPLOT
.WC	worst case analysis
.WIDTH	file line width

Post-Processor Directives

#AUTOPLOT	automatically plot specified variables or expressions
#CALC	calculate expressions
#CALC FFT	calculate the Fast Fourier transforms
#MEASURE	measure performance specs
#REVISION:	revision information
#RUNSCRIPT	run script file
#SUBTITLE	mark following line as the plot subtitle
#TABULATE	tabulate OUT file variables
#WRITEDATA	write data to file

DOCUMENTATION AND SUPPORT

Full Documentation

Complete manuals with over 600 pages in PDF format. Extensive online help. More than 350 sample circuits are included to illustrate program features and simulation techniques.

User Support

We provide expert, comprehensive and prompt support for our products to registered users:

- Free email and telephone technical support.[2]
- Free software maintenance service. [2]
- Discount offers for version upgrades and new products.

[2] For active products only. Limitations apply, call or write for complete details.

Updates

We promptly and frequently correct reported software problems or add minor changes by posting free software service updates and patches for download. We also make timely new version releases, with significant new and improved functionality, based on our research, recent technological and industry developments, and user feedback. Registered users have the option to purchase individual major version updates at substantial discount over the full product.

Download your free fully functional Demo version from our Web site:

penzar.com

ORDERING

Penzar Development accepts direct orders from US and international customers. You can order TopSpice products online at our website or by returning the Order Form with payment to Penzar Development. Payment may be made using check or money order drawn on US bank, credit card (online only), PayPal account (online only), or bank transfer (international customers only). Penzar Development also accepts purchase orders from qualifying US businesses, educational institutions and government agencies. Penzar Development does not accept COD orders or letters of credit. California customers must add applicable sales tax or attach signed resale card. Pricing and complete ordering information are available at our website.

Email questions to **sales.info@penzar.com**

Penzar Development

P.O. Box 900847
Palmdale, CA 93590
U.S.A.

Tel: 1-818-884-2246
Website: penzar.com
Email: sales.info@penzar.com

Rev. 6/2015