

Index

Why a new book for LTspice XVII?	18
Preface	20
1 What's new in LTspice XVII	21
1.1 What's new in the new LTspice XVII version.	22
1.2 Creating and adding attributes to new directories	23
1.2.1 LTspice symbol.	23
1.3 LTspice XVII dialogue improved on Windows	26
1.4 New features in the input graphical interface.	28
1.5 Easier access to the eye diagram.	32
1.6 Random state machine	33
1.7 Briefly revisiting LTspice IV	36
2 Functionality and first example.	37
2.1 First use of LTspice XVII	37
2.1.1 Running LTspice XVII on Windows	37
2.2 Functionality of LTspice XVII.	38
2.3 Menus of the LTspice XVII launch phase	38
2.3.1 File menu	40
2.3.2 View menu.	40
2.3.3 Tools menu	40
2.3.4 Help menu: common to all operation phases of LTspice XVII	41
2.3.5 Pop-up menu of the launch page	41
2.4 A detailed example described step by step	43
2.4.1 Entering a schematic	47
2.4.2 Configuring components	55
2.4.3 Configuring a simulation	61
2.4.4 First frequency measurements.	64
2.4.5 Temporal measurements	64
2.4.6 FFT measurements.	68
2.4.7 Harmonic distortion measurements	69
2.4.8 Maximum amplitudes before clipping	71
2.4.9 Zooming into a part of the curve to observe a fault	74
2.4.10 To conclude this first approach	76
3 Schematics graphical editor	77
3.1 The commands of LTspice XVII.	77
3.2 The schematics graphical editors.	77
3.2.1 File menu	77
3.2.2 Edit menu	79
3.2.3 Hierarchy menu	82
3.2.4 View menu.	83

3.2.5	Simulate menu	86
3.2.6	Tools menu	88
3.2.7	Window menu (Organising display windows)	90
3.2.8	Help menu	91
3.2.9	Pop-up menu of the schematics graphical editor	91
3.3	Component databases	99
3.4	Creating a new schematic	100
3.4.1	Opening the schematics graphical editor	100
3.4.2	Placing the first elements on the schematic page	101
3.4.3	The main commands of the schematics editor	102
3.4.4	Interconnect the elements of a schematic	104
3.4.5	Entering a component value or characteristic	105
3.4.6	Entering component values using the attribute editor	108
3.4.7	Enriching the schematic (optional)	109
3.4.8	Adding the simulation, the source and the (optional) directives	110
3.4.9	Saving your schematic	110
3.4.10	Launch the simulation.	111
3.5	Revisiting the schematics editor usage rules	111
3.5.1	Two points deserve your full attention.	111
3.5.2	Exporting a schematic.	112
4	Syntax and component editor	113
4.1	General syntax rules of LTspice XVII	113
4.2	Component value editors	115
4.3	Procedures to access common or complex component models	119
4.3.1	Modifying a component's current values	119
4.3.2	Displaying component attributes and modifying component values	124
4.4	Usage of the attribute editor lines.	127
4.5	Displaying the attributes of a component using two models	128
5	Symbol editor and hierarchical links	132
5.1	Symbol editor menu	132
5.1.1	File menu	132
5.1.2	Edit menu	132
5.1.3	Hierarchy menu	133
5.1.4	Draw menu	133
5.1.5	View menu.	133
5.1.6	Tools menu	133
5.1.7	Window menu (Organising display windows)	134
5.1.8	Help menu	134
5.1.9	Pop-up menu of the symbol editor	134
5.2	First step: draw the symbol	134
5.3	Second step: add connection pins	134
5.4	Third step: adding or modifying attributes	135
5.5	Possible calls by a symbol	137

5.6	Visible attributes attached to the symbol	139
5.7	Automatic generation of symbols from a schematic section	140
5.8	Automatic generation of symbols from a Netlist	141
5.9	Hierarchical links in LTspice XVII	146
5.10	Hierarchy usage rules	147
5.10.1	A method that resembles Russian dolls.	147
5.11	Rules to follow for a hierarchical structure	147
5.11.1	Hierarchy of levels	149
5.12	Hierarchy menu commands	150
5.13	Example: development of a two-level simple hierarchical structure	150
5.13.1	Screen 1: a sub-circuit model	151
5.13.2	Screen 2: a secondary schematic.	151
5.13.3	Screen 3: the main schematic	152
5.13.4	Screen 4: simulation of the main schematic	152
5.14	Exporting the Hierarchy directory	152
5.15	Interaction between the low and the high level	153
6	Netlist editor	154
6.1	Historical origin of Netlists.	154
6.2	Netlist, a mandatory passage.	154
6.3	Structure, syntax and conventions of Netlists.	154
6.4	Example of a Netlist	155
6.5	Netlist editor menus	156
6.5.1	Edit menu	156
6.5.2	View menu.	156
6.5.3	Simulate menu.	156
6.5.4	Netlist editor pop-up menu	156
6.6	Editing a Netlist	157
6.7	Syntax of the Netlist file .cir, .net or .sp.	158
6.8	How to open the Netlist editor from a schematic	158
6.9	Running a Netlist	162
6.10	Exporting a Netlist corresponding to a schematic	163
6.11	System commands used in Netlists	164
7	Graphical editor and numerical output	165
7.1	Displaying simulation computation results.	165
7.2	How to select measurement points on your schematic	165
7.3	How to display a measurement on the virtual oscilloscope	165
7.3.1	Viewing a voltage referenced to ground	166
7.3.2	Viewing a current	166
7.3.3	Viewing a differential voltage (not referenced to ground).	166
7.3.4	Deleting prior traces	167
7.3.5	Selectively deleting one or more traces.	167
7.3.6	Displaying instantaneous power.	167

7.3.7	Displaying the average power and the integral of the energy over the displayed period	168
7.3.8	Displaying the average voltage, the average current or the true RMS value	168
7.4	Using the menus	169
7.4.1	Virtual oscilloscope editor and FFT analysis	169
7.4.2	File menu	170
7.4.3	View menu	170
7.4.4	Plot Settings menu (Virtual oscilloscope configuration)	171
7.4.5	Simulation menu (running the simulation)	173
7.4.6	Tools menu	173
7.4.7	Window menu	174
7.4.8	Help menu	174
7.4.9	Virtual oscilloscope pop-up menu	174
7.5	Choosing which measurements to display	175
7.6	Adding an additional trace or pane	177
7.6.1	Adding an additional trace	177
7.6.2	Adding an additional pane	179
7.7	Zoom functions	179
7.8	Mathematical operations in the virtual oscilloscope	179
7.9	Request the calculation of a mathematical expression	180
7.9.1	Modifying the appearance of a trace	181
7.10	User-defined functions	181
7.11	Axis scale modifications	183
7.11.1	Vertical axis scales	183
7.11.2	Horizontal axis scales	184
7.12	Using the virtual oscilloscope in X-Y mode	185
7.13	Pop-up menu and scales	186
7.14	Other settings for scales	187
7.14.1	Left vertical scale	188
7.14.2	Displaying only the phase	188
7.14.3	Right vertical scale	189
7.15	Managing the multi-trace virtual oscilloscope	190
7.16	Information concerning the traces of the virtual oscilloscope	192
7.17	Other arrangements of the virtual oscilloscope traces	194
7.18	Colour controls of the virtual oscilloscope	194
7.19	Two measurement cursors	196
7.19.1	Placement of measurement cursors on curves	197
7.20	Display of coordinates on the bottom bar	199
7.21	Saving the configuration of the virtual oscilloscope	200
7.22	Faster loading of files	201
7.23	RAM memory and address space	202
7.24	Presentation of the SPICE Error Log buffer file	202
7.25	The .FOUR command	202
7.26	The combination of the .STEP and .MEAS commands	206
7.26.1	Computation steps using the .MEAS command	209
7.26.2	Level 2 display with the SPICE Error Log file	211

7.27	Controller adjustment	216
7.27.1	Measurement principle	216
7.28	Choosing the resolution of the .STEP loop correctly	224
8	The commands	228
8.1	Command definition	228
8.1.1	The command editor	228
8.1.2	Command syntax	232
8.1.3	First syntax rule	233
8.1.4	Second syntax rule	233
8.1.5	Third syntax rule	234
8.2	.options parameters that modify how a simulation runs	235
8.3	.ic setting the initial conditions for a time simulation	239
8.4	.savebias saving a DC operating point	241
8.5	.loadbias loading a DC operating point	242
8.6	.net computing network parameters with an AC simulation	242
8.7	.nodeset initial conditions for DC analysis	243
9	The six main simulations	245
9.1	Presentation of the six main simulations	245
9.1.1	DC (continuous) simulations	246
9.1.2	AC (frequency) simulations	246
9.1.3	Non-linear circuit simulations	247
9.1.4	Simulation characteristics	247
9.2	Selection criteria in simulations	248
9.2.1	When the circuit's only excitation source is a DC voltage	248
9.2.2	When the circuit's only excitation source is a low-amplitude AC voltage	250
9.2.3	When the circuit's only excitation source is a high-amplitude AC voltage (or any other action triggering the non-linearity of the circuit components)	251
9.3	.op – simulation of a DC operating point	251
9.4	.dc – DC simulation with sweeping (one to three sources)	253
9.5	.tf – simulation of the transfer function (gain, input impedance, output impedance)	254
9.6	.ac – simulation of an AC signal around an operating point	257
9.7	.noise – noise simulation	260
9.8	.tran – time simulation (non-linear)	261
9.9	.tran time simulation configuration	264
9.9.1	Caution with the Maximum Timestep value	264
9.9.2	uic parameter (time simulation)	270
9.9.3	startup parameter (time simulation)	271
9.9.4	steady parameter (time simulation)	273
9.9.5	.nodiscard parameter (time simulation)	275
9.9.6	.step parameter (time simulation)	276

10	Two dedicated types of analysis	280
10.1	Description of these two types of analysis	280
10.2	.temp – temperature sweep simulation	280
10.3	.four – edit the harmonics in numerical form	285
10.4	How the FFT analysis works	286
10.4.1	FFT configuration	286
10.4.2	Conditions to meet to obtain a representative FFT analysis	290
11	Passive and active components	303
11.1	Introduction to using the parameters of a component model	303
11.2	Passive components	303
11.3	Generic resistor R model	304
11.4	Generic capacitor C model	306
11.4.1	The standard generic capacitor sub-model	307
11.4.2	Second generic capacitor sub-model	309
11.5	Generic inductor L model	309
11.6	Active components	310
11.7	Generic diode D model	310
11.7.1	The first standard and linear generic sub-model of diode D	311
11.7.2	The second non-linear generic sub-model of diode D	312
11.7.3	The third Berkeley generic sub-model of diode D (non-linear)	313
11.8	Generic bipolar transistor Q model	315
11.8.1	The Ebers-Moll and Gummel-Poon generic sub-models	315
11.8.2	VBIC generic sub-model	318
11.9	Generic JFET J model	322
11.10	Generic MOSFET M model	324
11.11	Generic model of MOSFET with double vertical distribution	328
11.12	Generic MESFET Z model	332
11.13	Generic IGBT model	334
11.14	Unijunction, phototransistor, thyristor and TRIAC transistor models	335
11.15	Multiplier of components in parallel or in series	335
11.15.1	Components in parallel	335
11.15.2	Components in series	337
12	Inductors, transformers and mutual induction	339
12.1	Generic inductor L model (without saturation)	339
12.2	Coil and air-core inductor without a saturable magnetic circuit	339
12.2.1	Coil and wound inductor without a magnetic circuit	340
12.2.2	Coil and inductor with a saturable magnetic circuit	341
12.3	Generic L inductor model	343
12.3.1	Generic inductor sub-model (linear and without saturation)	343
12.3.2	Default values applied to inductors by LTspice	344
12.4	CHAN model of inductor L	345
12.4.1	Model with saturation and hysteresis	345
12.4.2	First group of parameters	346

12.4.3	Second group of parameters	347
12.4.4	Third group of parameters	348
12.5	The transformer model	350
12.5.1	How to create a transformer using the standard model	350
12.5.2	Adding a transformer: first method	352
12.5.3	Adding a transformer: second method	353
12.5.4	Conclusion on the two transformer creation methods	354
12.5.5	Multiple operating modes of a transformer	357
12.6	Transformer with saturation and hysteresis of magnetic material	358
13	Importing a component from the internet or creating it	361
13.1	Modelled components compatible with SPICE	361
13.1.1	Modelled components compatible with LTspice	361
13.1.2	Composition of a model	362
13.2	Various websites	362
13.3	The NXP website	363
13.4	Cachan Technical College's download website	369
13.5	The Texas Instruments website	371
13.6	Download websites	373
13.6.1	The LTwiki.org website	374
13.6.2	The Groups.io website: LTspice	378
13.6.3	List of manufacturer websites providing SPICE models	378
13.7	Differences between a model and a sub-circuit	379
13.7.1	The model	379
13.7.2	The sub-circuit	381
13.8	How to modify a model	383
13.8.1	Observation on the creation of models	383
13.9	How is a new model added to LTspice?	384
13.9.1	First method: adding a model to an existing library	384
13.9.2	Second method: adding a model to a directory on the hard drive	388
13.9.3	Third method: adding a model directly to the schematic	389
13.9.4	A few guidelines regarding the three methods	390
13.10	How to add a sub-circuit	390
13.10.1	First method: adding a symbol to the symbol directory	391
13.10.2	Second method: reusing a schematic symbol	395
13.10.3	Third method: adding the new components to the schematic	396
13.10.4	Automatic symbol creation from the Netlist	400
13.10.5	Automatic symbol creation from the sub-circuit schematic	400
13.10.6	Creating a symbol by modifying the attributes of an existing symbol	402
13.11	Step-by-step sub-circuit creation	405
13.11.1	Creation of a sub-circuit using the automatic generator in 14 simple steps	405
13.11.2	Creation of a sub-circuit using the reassigned SRC symbol in 16 simple steps	405
13.11.3	Second example: creating a configurable sub-circuit	406
13.12	Using models and sub-circuits	413
13.13	.LIB and .INC commands	413

14	The commands .FUNC, .MEAS, .PARAM, .STEP and AKO	415
14.1	The commands	415
14.2	The .FUNC command	416
14.2.1	First example	416
14.2.2	Benefits of using the .FUNC commands	418
14.2.3	Second example	418
14.3	The .MEAS command	419
14.3.1	First use: computation using off-curve variables	420
14.3.2	Second use: computation using a single curve value	421
14.3.3	Third use: computation using all the curve values	423
14.3.4	Seven examples for the use of the .MEAS command	425
14.4	Curly brackets, .param, ako and .step commands	433
14.5	Curly brackets	434
14.6	The .PARAM command and the PARAM function	437
14.7	The .STEP command	440
14.7.1	Be careful not to run too many loops	440
14.7.2	The .step command is powerful	440
14.7.3	The order of execution of multiple loops	441
14.7.4	Example 1	442
14.7.5	Example 2	443
14.7.6	Example 3	446
14.7.7	Example 4	448
14.8	Frequently asked questions regarding the .step command	450
14.8.1	Can a simulation computation be automated?	450
14.8.2	How are simulations performed with the temperature?	450
14.8.3	How is a simulation configured with a component value?	453
14.8.4	How to create a parametric simulation with a source	454
14.8.5	How to create a three-level parametric simulation	454
14.8.6	The Select Step command in the pop-up menu	457
14.8.7	How can curve values be identified?	459
14.8.8	How can the order of several nested .step loops be changed?	462
14.8.9	Maximum number of increments of a .step loop	463
14.8.10	Different ways a .step loop can be incremented	464
14.8.11	Loop defined per decade with a logarithmic distribution	464
14.8.12	Loop defined per octave with a logarithmic distribution	464
14.8.13	Loop with a linear distribution of intervals	465
14.8.14	Loop with a list of increment values	465
14.8.15	How can .step loop variation units be included in the x-axis?	466
14.8.16	How can non-numerical values be incremented?	466
14.8.17	Additional information regarding the .step command	467
14.8.18	Maximum number limit for a single .step loop	468

15	The “AKO:” commands: .FOUR .WAVE .MODEL .SUBCKT .INCLUDE .LIB .IC .SAVE	470
15.1	The “AKO:” function	470
15.1.1	The AKO: command (type changer).	470
15.1.2	First example: changing a model parameter	471
15.1.3	Second example: changing several model parameters	473
15.1.4	Third example: changing 7 parameters using 3 .step loops.	475
15.1.5	Fourth example: changing the component model	476
15.2	The .FOUR command	479
15.3	The .WAVE command	480
15.4	The .MODEL command	481
15.5	The .SUBCKT command	483
15.6	The .INCLUDE and .LIB commands.	484
15.7	The .IC command	485
15.8	The .SAVE command.	486
16	The .options command and the control panel	488
16.1	The .OPTIONS command	488
16.1.1	Eye diagram control parameters (transmission)	490
16.1.2	Transient simulation control parameters (non-linear signals).	490
16.1.3	Control parameters of the .meas command	490
16.1.4	Control and convergence help parameters	491
16.1.5	Newton-Raphson method control parameters	492
16.1.6	Saving and compression control parameters	492
16.1.7	Component control parameters	493
16.1.8	Control parameters of DC bias simulations	493
16.1.9	General control parameters	494
16.1.10	New parameters in LTspice XVII	495
16.2	The nine tabs of the control panel	495
16.2.1	Compression tab	496
16.2.2	Save tab	497
16.2.3	SPICE tab	499
16.2.4	Drafting tab	503
16.2.5	Netlist tab	505
16.2.6	Tools/Control Panel/Sym. & Lib. Search Paths Operation tab.	506
16.2.7	Waveform tab.	509
16.2.8	Operation tab.	510
16.2.9	Hacks tab	511
16.2.10	Internet tab	513
16.3	Keyboard shortcuts.	514
16.4	Color selection tab: Color Preferences	518
16.5	Opening an editor.	520

17	Voltage and current source editor	521
17.1	Two types of source and two editors	521
17.2	Two major types of source, dependent or independent	522
17.3	All simulations need an independent source	523
17.4	How to place a source on a schematic	524
17.4.1	Three independent sources	524
17.4.2	Nine dependent sources, six linear and three non-linear	525
17.5	V – independent voltage source (NCVS)	526
17.5.1	PULSE voltage source	527
17.5.2	SINE voltage source (sinusoidal)	527
17.5.3	EXP voltage source (exponential)	527
17.5.4	SFFM voltage source (frequency-modulated)	528
17.5.5	Arbitrary voltage source modulated by a PWL command	528
17.5.6	Voltage source modulated by a .wav file	528
17.6	I – independent current source (NCCS)	529
17.6.1	PULSE current source	530
17.6.2	SINE current source (sinusoidal)	530
17.6.3	EXP current source (exponential)	530
17.6.4	SFFM current source (frequency-modulated)	531
17.6.5	Modulated current sources	531
17.7	Load – independent active load (CSVL)	533
17.8	The independent source editor	534
17.8.1	Configuration of the frequency sweep of an independent source for an AC simulation	536
17.8.2	Configuration of independent sources for a DC simulation (low amplitudes)	537
17.8.3	Configuration of independent sources for an AC simulation (low amplitudes)	541
17.9	Dependent sources	568
17.10	E – voltage-controlled voltage source (VCVS)	568
17.10.1	First model: the transfer function is a constant	569
17.10.2	Second model: the transfer function is a table of value pairs	571
17.10.3	Third model: the transfer function is a Laplace transform in parameter S	572
17.11	F – current-controlled current source (CCCS)	574
17.12	G – voltage-controlled current source (VCCS)	577
17.12.1	First model	578
17.12.2	Second model	578
17.12.3	Third model	578
17.13	H – current-controlled voltage source (CCVS)	579
17.14	Voltage or current source B	580
17.15	B – non-linear, arbitrary voltage source (CVS)	580
17.16	B – arbitrary, non-linear current sources (CCS)	582
17.17	Attribute editor for dependent sources	583

18	Logic and secondary functions	586
18.1	Shared characteristics of logic circuits	586
18.2	Standard logic gates	587
18.3	Logic gates with Schmitt triggers	587
18.4	Logic flip-flops	588
18.4.1	SRflop	588
18.4.2	Dflop	588
18.5	PhaseDet: phase detector with current output	589
18.6	SampleHold: sample and hold	592
18.6.1	First mode – S/H command	593
18.6.2	Second mode – CLK command	594
18.7	Modulate and Modulate2: frequency and amplitude modulator	594
18.8	The eye diagram and the BAUDRATE function	598
18.9	Configuring the .AC LIST x-axis	607
18.9.1	Example one	608
18.9.2	Example two	610
18.10	Encryption	612
18.11	Options to add when starting LTspice XVII	615
18.12	Reserved words	616
18.13	Computation precision with LTspice XVII	616
18.13.1	Distribution of the 64 bits	616
18.13.2	Limit of this option	616
18.13.3	Increasing the precision	617
18.14	Attribution table of the generic component model	617
18.15	S and W controlled switches	618
18.16	S – voltage-controlled switch (two models)	619
18.16.1	First standard model Level=1	621
18.16.2	Second complete model Level=2	622
18.17	W – current-controlled switch (one model)	623
18.18	O – lossy transmission line (one model)	628
18.19	T – lossless transmission line (one model)	629
18.20	U – RC transmission line (one model)	630
19	Monte Carlo and Worst Case	632
19.1	Monte Carlo overview	632
19.1.1	Monte Carlo 1 (uniform distribution and simple schematic)	632
19.1.2	Monte Carlo 1	633
19.1.3	Monte Carlo 2 (Gaussian distribution and simple schematic)	634
19.1.4	Monte Carlo 2 (Gaussian distribution)	635
19.1.5	Monte Carlo 2 (with semiconductors, sub-circuits or models)	638
19.1.6	Monte Carlo 1 (uniform distribution with sub-circuits or models)	639
19.1.7	Monte Carlo 1 (uniform distribution and semiconductors)	643
19.2	Worst Case overview	647
19.2.1	Worst Case 1 (random selection)	647
19.2.2	Worst Case 2 (automatic sweep)	647

19.2.3	Worst Case 3 (extended application)	647
19.2.4	Worst Case 1 (random selection and simple schematic)	648
19.2.5	Worst Case 1	648
19.2.6	Worst Case 2 (automatic sweep and limitation to 16 components)	651
19.2.7	Worst Case (with semiconductors, sub-circuits or models)	658
19.2.8	Worst Case 1 (with sub-circuits or models)	659
19.2.9	Worst Case 1 (with semiconductors)	661
19.2.10	Worst Case 2 (with models and limited to 16 components)	663
19.2.11	Worst Case 2 (with semiconductors, limited to 16 components)	664
19.2.12	Worst Case 3 (no limitation, 2 runs required)	664
20	SOAtherm: thermal model for MOSFET with heatsink	690
20.1	SOAtherm for LTspice XVII	690
20.2	How to use the SOAtherm model with LTspice XVII	690
20.3	The SOAtherm models	691
20.4	How to operate the SOAtherm model in LTspice XVII	691
20.5	Does the SOAtherm model work with all MOSFETs in LTspice XVII?	691
20.6	Calling a MOS that is incompatible with SOAtherm	692
20.7	How to recognise a MOS-compatible SOAtherm	693
20.8	How to access the terminals	696
20.9	How to use the second SOAtherm-HeatSink model	703
20.10	How thermal flux really works	709
20.11	How SOAtherm models work	709
20.11.1	SOAtherm-HeatSink model	709
20.11.2	SOAtherm-PCB model	710
20.12	Influence of the time constant	711
20.13	Characteristics of the heat sink	713
20.14	How to achieve thermal equilibrium of MOSFETs in parallel	717
20.14.1	Schematic with MOSFETs in parallel	718
20.14.2	The first schematic (the worst solution)	718
20.14.3	The second schematic (a less bad solution)	721
20.14.4	The best solution	723
20.15	General conclusion	724
21	A few examples	725
21.1	Difference between LTspice XVII simulations and reality	725
21.2	The .noise simulation	726
21.3	A PWL function to create a generator of arbitrary voltage or current signals	730
21.3.1	Specific PWL commands	731
21.4	The virtual oscilloscope: changing the configuration of the oscilloscope trace axes	732
21.5	Stability of operational amplifiers with an AC simulation (or how to juggle poles and zeros)	734
21.5.1	First example (Figure 21.11)	735
21.5.2	Second example	739

21.6	How to improve the running speed of LTspice XVII	744
21.6.1	If you do not need to see the start-up phase of an SMPS power supply, you can reduce the start-up time by reducing the soft-start setting	744
21.6.2	Delaying the application of the load for an SMPS	746
21.6.3	Defining the initial conditions	747
21.6.4	Reducing the amount of data of a transient simulation	750
21.6.5	Avoiding the step of finding the initial operating point	752
21.6.6	Converting to Fast Access format when viewing traces in the oscilloscope	753
21.7	Saving time with synchronous detection	754
Index	763

To see the syntax of each of these three passive components in detail, refer to the corresponding chapters.

4.3 Procedures to access common or complex component models

In LTspice XVII, each component is represented by a common model.

A model is the mathematical representation of a component's behaviour. This representation is made on the basis of an equivalent schematic consisting of basic components (resistors, capacitors, inductors and sources). Each basic component is characterised by one or more numerical values which accurately define its behaviour.

LTspice XVII offers, for a certain number of components, **several models** of increasing complexity to represent more accurately the particular features of each component. It is down to you to choose the model that best satisfies the requirements of your system or its environment.

Let us look at an example. A resistor can be represented by a common model, with a limited resistance value, potentially the tolerance of this value and the power that this resistor can dissipate. In general, this is enough but if you wish to test your system as a function of temperature, you will need the temperature coefficient of this resistor. To introduce this new parameter, you must necessarily use a more complete resistor model which has this temperature coefficient as a parameter. On the other hand, it is most likely that this more complete model has many other parameters quantifying other aspects of resistor operation. If you are not interested in these characteristics or do not know their values, you do not need to enter them; LTspice XVII will replace these missing values with average values called "default parameters".

- **First procedure: the common model.** This procedure involves entering standard values of a component represented by its common model.
- **Second procedure: the more complete models.** This procedure lets you access less standard values. You can enter these more numerous parameters in the **attribute editor**.

Each component has its own models and dedicated parameters. In a subsequent chapter, we have dedicated an entire section to the detailed description of each component's models.

4.3.1 Modifying a component's current values

Certain components have a common model that offers the ability to use a quick-input window such as that of a resistor (see this section).

Two examples of a monolithic power MOSFET:

- M1 Nd Ng Ns 0 MyNMOSFET
.model MyNMOSFET NMOS (KP=.001)
- M1 Nd Ng Ns Nb MyPMOSFET
.model MyPMOSFET PMOS (KP=.001)

This model uses the reserved keywords NMOS and PMOS to specify that a monolithic MOSFET uses an N or P channel.

Another generic VDMOS sub-model is used for the double vertical power distribution MOSFET (refer to the next section).

CAUTION

MOSFETs include many **generic sub-models**, and this makes them quite difficult to understand. Each **generic sub-model** is adapted to the **MOSFET** component it represents. The choice of sub-model is made by the manufacturer during the SPICE model production phase. It is automatically recognised by LTspice based on the parameters entered in the model, and the process is fully transparent to the user.

The **generic sub-model** of the monolithic **MOSFET** has three or four output terminals according to its reference: **Nd**, **Ng**, **Ns**, and **Nb** respectively for the **drain**, the **gate**, the **source** and the **substrate** (or **bulk**).

L and **W** are the channel length and width in meters.

Ad and **As** are the drain and source diffusion areas respectively, in square meters.

If any of the four parameters **L**, **W**, **Ad** or **As** is not specified, the default value is automatically added, as with all other generic sub-models.

Pd and **Ps** are the **drain** and **source** junction perimeters, in metres.

Nrd and **Nrs** are the equivalent number of squares of the drain and source diffusions respectively; these values are the multipliers of the sheet resistance **Rsh** in the model.

Pd and **Ps** have a default value of 0, whereas **Nrd** and **Nrs** are equal to 1.

Off indicates an initial condition on a DC simulation device. Initial conditions can be specified with the command **.ic = VDS, VGS, VBS** when the **UIC** option is active (transient simulation). This occurs when a **.tran** simulation starts with a value other than **t = 0**.

20 SOAtherm: thermal model for MOSFET with heatsink

20.1 SOAtherm for LTspice XVII

The ability to insert or remove SMPS cards when hot requires the use of a MOSFET, which has both a **low R_{on} resistance** and a safe operating area (**SOA**), especially for high-temperature transients. In fact, it is common to find **MOSFETs** optimised for **low R_{on} resistance**, but which have proved unsuitable for use with high-temperature transients (**SOA**).

For applications with high-temperature transients (**Hot Swap**) that allow a card to be directly inserted and removed hot from a backplane with safety, the **SOAtherm model** is a valuable tool. The safe operating area (SOA) of the **MOSFET** is guaranteed by the manufacturers for **Hot Swap** applications (high-temperature transients). Twenty-four MOSFET models of this type are available in the LTspice XVII database.

A thermal model called SOAtherm was developed based on the **Cauer model** by **Dan Eddleman**, an engineer specialised in analogue electronics who has more than 15 years of experience at Linear Technology as designer of integrated circuits, manager of the integrated circuit design centre of Singapore and application engineer.

The **SOAtherm model** is included in LTspice XVII. In circuit simulations, it makes it possible to verify that the **safe operating area (SOA)** of a MOSFET is not exceeded, thus avoiding many design errors which cause failures in pulse circuits and SMPSs.

Several examples illustrate the advantages of using the SOAtherm model to verify the correct use of MOSFETs.

20.2 How to use the SOAtherm model with LTspice XVII

If you installed LTspice XVII before 18 January 2019, you have the symbol **SOAtherm-NMOS** on the list of available models, as was the case in LTspice IV, but don't be surprised to find that it doesn't work, because it was deactivated on this date. On the other hand, if you installed LTspice XVII after this date, this model no longer appears on the list of available symbols because new methods have been implemented to achieve SOAtherm functionality.

Today, with LTspice XVII, the procedure for **activating the SOAtherm model** is radically different. The functionality and the result of the simulations are exactly the same as they were in LTspice IV, but we must change our working habits to continue to take advantage of the full power of the **SOAtherm model**.

A circuit designer can use the **SOAtherm model** to verify that the **SOA** of a MOSFET is suitable for one of the many applications involved. The **SOA** corresponds to the **temperature safe operating area of a MOSFET chip**.

No additional heat sink or model of thermal dissipation through the copper surface of an electronic card is required to use this model. However, in certain applications, particularly during power transients of more than 10 milliseconds, it may be desirable to add a means of removing heat. This is why the **SOAtherm model** has two other models which can be used to either add a manufactured heat sink when the power is high (**SOAtherm-HeatSink**) or use a copper-plated surface of the PCB as a heat sink when the power is low (**SOAtherm-PCB**).