

INDEX

PREFACE	5
THE AUTHOR	6
INDEX	7
FOREWORD	21
1 LTSPICE IV: INTRODUCTION AND HISTORY	25
1.1 Circuit simulation with LTspice IV	25
1.1.1 The three basic steps	25
1.1.2 Results analysis	27
1.2 The story of LTspice IV	27
1.2.1 CANCER – 1969 to 1971	27
1.2.2 SPICE1 – 1972 to 1974	28
1.2.3 SPICE2 – 1975 to 1983	28
1.2.4 SPICE3 – 1984 to 1990	28
1.2.5 The birth of LTspice – 1990–2007	29
1.2.6 LTspice/SwitcherCADIII – 1999–2008	29
1.2.7 LTspice IV version since the end of 2008	29
1.3 What are the main benefits of LTspice IV?	30
1.4 What can LTspice IV do?	30
1.5 What LTspice IV cannot do	31
1.6 Conclusion	32
2 FILES SUPPLIED WITH LTSPICE IV	33
2.1 Installation of LTspice IV	33
2.1.1 System requirements	33
2.1.2 Downloading LTspice IV	33
2.2 How does LTspice IV work?	34
2.3 LTspice IV's editors	36
2.4 Included files	37
2.4.1 Models, subcircuits, macro models, and component libraries	37
2.4.2 Application examples	37
2.5 LTspice IV file extensions	42
3 WORKING WITH LTSPICE IV AND FIRST EXAMPLE	45
3.1 First use of LTspice IV	45
3.1.1 Launching LTspice IV in Windows	45
3.2 How LTspice IV works	46
3.3 LTspice IV start-up phase menus	46
3.3.1 File Menu	48
3.3.2 View Menu	48

3.3.3	Tools Menu	48
3.3.4	Help Menu: This menu is the same in all stages of LTspice IV	49
3.3.5	Start-up page pop-up menu	49
3.4	A detailed example, step by step	50
3.4.1	Drawing a schematic	52
3.4.2	Enter the parameters of components	62
3.4.3	Enter simulation parameters	68
3.4.4	First frequency measurements	71
3.4.5	Transient measurements	72
3.4.6	FFT measurements	76
3.4.7	Measures of harmonic distortion	77
3.4.8	Maximum amplitudes before clipping	80
3.4.9	Zooming in on part of the trace to identify a defect	84
3.4.10	To conclude this first approach	86
4	SCHEMATICS EDITOR	87
4.1	The commands of LTspice IV	87
4.2	Schematics editor	88
4.2.1	File menu	89
4.2.2	Edit menu	91
4.2.3	Hierarchy Menu	93
4.2.4	View Menu.	94
4.2.5	Simulate Menu.	96
4.2.6	Tools Menu	97
4.2.7	Window Menu	97
4.2.8	Help Menu.	97
4.2.9	Schematic editor pop-up menu	98
4.3	Components databases.	99
4.4	Draw a new schematic	101
4.4.1	Open the schematic editor.	101
4.4.2	Place the first elements on the schematic.	101
4.4.3	Main commands of the schematic editor.	102
4.4.4	Connecting elements from the schematic	105
4.4.5	Enter the value or reference of a component.	105
4.4.6	Enter the values of a component with the attribute editor	108
4.4.7	Enrich the schematic (optional)	109
4.4.8	Add simulation, source and directives (optional).	110
4.4.9	Save your schematic.	111
4.4.10	Launch the simulation.	111
4.5	Incorporation of a wiring BUS.	111
4.6	Reminder of the schematic editor use rules.	115
4.6.1	You must be very careful about two points:	116
4.7	Export a schematic.	116

5	SYNTAX AND COMPONENTS EDITOR	117
5.1	General syntax rules in LTspice IV	117
5.2	Component values editor	119
5.3	Procedures to access usual or complex component models	123
5.3.1	Changing the current values of a component	124
5.3.2	Display of attributes and modification of components' values	128
5.4	Allocation of the attributes editor fields	131
5.5	Display of a component's attributes using two models	132
6	SYMBOL EDITOR AND HIERARCHY	135
6.1	Symbol editor menu	135
6.1.1	File Menu	135
6.1.2	Edit Menu	135
6.1.3	Hierarchy Menu	136
6.1.4	Draw Menu	136
6.1.5	View Menu	136
6.1.6	Tools Menu	137
6.1.7	Window Menu (management of the display windows)	137
6.1.8	Help Menu	137
6.1.9	Symbol editor context menu	137
6.2	First step: Drawing the symbol	137
6.3	Second step: Adding connection terminals	138
6.4	Third step: Adding or modifying attributes	139
6.5	Possible calls from a symbol	141
6.6	Visible attributes connected to the symbol	143
6.7	Automatic symbol generation from a section of the schematic	144
6.8	Automatic symbol generation from a Netlist	145
6.9	Hierarchical links in LTspice IV	150
6.10	Hierarchy usage rules	151
6.10.1	A method similar in concept to the Matryoshka dolls	151
6.11	Hierarchy structure rules	152
6.11.1	Levels hierarchy	153
6.12	Commands of the Hierarchy menu	154
6.13	Example: Work flow of a simple two-level hierarchic structure	155
6.13.1	Screen number 1: Subcircuit model	155
6.13.2	Screen number 2: Secondary schematic	155
6.13.3	Screen number 3: Main schematic	156
6.13.4	Screen 4: Simulation of the main schematic	156
6.14	Export of the hierarchy directory	157
6.15	Interactivity between the lower and the higher level	157
7	NETLIST EDITOR	159
7.1	The origin of Netlists	159
7.2	The Netlist: A mandatory step	159
7.3	Structure, syntax and conventions of Netlist	159

7.4	A Netlist example	160
7.5	Netlist editor menus	161
7.5.1	Edit Menu	161
7.5.2	View Menu	161
7.5.3	Simulate Menu	162
7.5.4	Netlist editor context menu	162
7.6	Writing a Netlist	162
7.7	Syntax of Netlist files .CIR, .NET or .SP	163
7.8	How to open the Netlist editor from a schematic	164
7.9	Running a Netlist	167
7.10	Exporting the Netlist of a schematic	168
7.11	System commands used in Netlists	169
8	MEASUREMENTS, VIRTUAL OSCILLOSCOPE AND FFT EDITORS.	170
8.1	LTspice IV waveform viewer	170
8.1.1	Display of the simulation calculation results	170
8.1.2	How to select measurement points on your schematic?	170
8.2	How to display a measurement on the virtual oscilloscope?	170
8.2.1	Visualise a ground referenced voltage	170
8.2.2	Visualise a current	171
8.2.3	Visualise a differential voltage (not ground referenced)	171
8.2.4	Erase previous traces	172
8.2.5	Selectively erase one or more traces	172
8.2.6	Display instantaneous power dissipation	172
8.2.7	Display average power and energy integral of power over time displayed.	173
8.2.8	Display average voltage or current over the displayed period or effective value (RMS)	173
8.3	Use of menus	174
8.3.1	Virtual oscilloscope and FFT analyser editor	174
8.3.2	File menu	175
8.3.3	View menu	175
8.3.4	Plot Settings Menu (Configuration of the virtual oscilloscope)	176
8.3.5	Simulation Menu (Launch simulation)	178
8.3.6	Tools Menu	178
8.3.7	Window Menu	179
8.3.8	Help Menu	179
8.3.9	Virtual oscilloscope context menu	179
8.4	Choosing the measurements to display	180
8.5	Add a trace or a screen	182
8.5.1	Add a trace	182
8.5.2	Add a screen	184
8.6	Zoom functions	184
8.7	Mathematical operations in the virtual oscilloscope	185
8.8	Request the calculation of an algebraic expression	185
8.8.1	Modify the appearance of a trace	186
8.9	User-defined functions	190

8.10	Modify the axes scales	191
8.10.1	Vertical axis scales	191
8.10.2	Horizontal axis scales	192
8.11	Use of the virtual oscilloscope in X-Y mode	193
8.12	Context menu and scales	194
8.13	Other scales configurations	195
8.13.1	Left vertical scale	195
8.13.2	Only display the phase	196
8.13.3	Left vertical scale	196
8.14	Display management of several traces on the virtual oscilloscope	198
8.15	Information about the virtual oscilloscope traces	200
8.16	Other traces customisation in the virtual oscilloscope	202
8.17	Control of the virtual oscilloscope's colours	202
8.18	Two measurement cursors	204
8.18.1	Placing measurement cursors on traces	205
8.19	Display of coordinates in the bottom banner	207
8.20	Save the virtual oscilloscope configuration	209
8.21	Acceleration of file loading	209
8.22	RAM and addressing space	210
9	SIMULATIONS CONFIGURATION DIRECTIVES	212
9.1	Definition of a simulation directive	212
9.1.1	Simulation directives editor	212
9.1.2	Syntax of simulation directives	214
9.1.3	First syntax rule	214
9.1.4	Second syntax rule	214
9.1.5	Third syntax rule	216
9.1.6	Never forget a mandatory parameter	217
9.2	.Options parameters modifying the execution of a simulation	218
9.3	.IC Fix initial conditions for transient simulation	222
9.4	.Savebias Save a DC operating point	223
9.5	.Loadbias Load a DC operating point	224
9.6	.Net Calculation of a network parameter with an AC simulation	224
9.7	.Nodeset Initial conditions for DC analysis	225
10	THE SIX MAIN SIMULATIONS	227
10.1	Presentation of the six main simulations	227
10.1.1	DC simulations (continuous)	227
10.1.2	AC Simulations (frequency)	228
10.1.3	Non-linear circuits simulations	228
10.1.4	Simulations characteristics	228
10.2	Choice criteria regarding simulations	229
10.2.1	If the only excitation source of the circuit is a direct voltage source	229
10.2.2	The only excitation source of the circuit is low amplitude alternating voltage	230

10.2.3	The only excitation source of the circuit is a high amplitude alternating voltage source (or any other causing the non-linearity of the components of the circuit)	230
10.3	.OP – Simulation of a continuous polarisation point	231
10.4	.DC – DC source sweep analysis (one to three sources)	232
10.5	.TF – Transfer function simulation (gain, input and output impedance)	234
10.6	.AC – Simulation of an AC signal around a polarisation point	235
10.7	.NOISE – Noise simulation	237
10.8	.TEMP – Temperature sweep simulation	238
10.9	.TRAN – Transient simulation (non-linear)	241
10.10	Configuration of the transient simulation .TRAN	243
10.10.1	Be careful with the Maximum Timestep value	243
10.10.2	Parameter: .uic (transient simulation)	248
10.10.3	Parameter: startup (transient simulation)	249
10.10.4	Parameter: steady (transient simulation)	250
10.10.5	Parameter: .nodiscard (transient simulation)	253
10.10.6	Parameter: .step (transient simulation)	254
10.11	.FOUR – Edit harmonics as numeric format	257
10.11.1	How does the FFT analysis work?	257
10.11.2	Conditions to fulfil to obtain a representative FFT analysis	261
10.11.3	Influence of Stop Time and Time step parameters on the FFT	273
10.12	Monte Carlo statistic simulations	275
10.12.1	First step	277
10.12.2	Second step	277
10.12.3	Third step	278
12.12.4	Comment on the Monte Carlo method	278
10.13	Simulations configuration	279
11	NUMERIC MEASUREMENTS, DOWNLOADS, BACKUP AND MODELS	281
11.1	Retrieving measurements as numeric data	281
11.1.1	Retrieving measurement files as numeric data	281
11.1.2	Declaration of variables	281
11.2	MEAS – Display measurements values numerically	281
11.2.1	First type of measurement: For only one X-axis point	282
11.2.2	Examples of use of .meas for only one X-axis point	285
11.2.3	Second type of measurement: For an interval between two points on the X-axis	287
11.2.4	Examples of use of parameters rise, fall, last and cross	289
11.2.5	Case of a NOISE simulation	297
11.2.6	Creation of a measurement script: File_name.meas	297
11.2.7	Precision of results obtained with the command .meas	303
11.3	.PARAM – Variables & Parameters	304
11.4	.STEP – Configurable intervals	307
11.4.1	The commands .step and select steps, step by step	309
11.5	.FUNC – User functions	314
11.6	Efficiency report of a DC/DC converter: steady	315
11.7	.FERRET – Download a file online	316

11.8	.GLOBAL General declaration	317
11.9	.SAVE Limitation of the quantity of saved data	317
11.10	.WAVE – Transform the output signal to .wav	318
11.10.1	Comments for .wav files	319
11.11	Configuration of a component value with the command .param	319
11.12	.MODEL – Define a SPICE model	320
11.13	.SUBCKT – Define a subcircuit	321
11.14	.INCLUDE – include a new library.	323
11.15	.LIB – Models or subcircuits library	324
11.15.1	Encrypted library	325
12	IMPORT OF COMPONENTS MODELS	326
12.1	Does LTspice IV need to download components models?	326
12.2	Macromodels and models	326
12.2.1	Macromodels or models file extensions: .MODEL or .MOD.	326
12.3	Subcircuits.	327
12.4	Libraries and models	327
12.5	A component model consists of two elements	328
12.6	Symbols to call components	328
12.7	Downloading a component model.	328
12.8	Three extensions for three ways to add components	329
12.9	One symbol can call several elements	329
12.10	Models libraries	330
12.10.1	How does the compiler detect that one component rather than another is used?.	331
12.10.2	Each component has several possible models	332
12.11	Models	332
12.12	Example: Subcircuit library 74htc.lib.	332
12.13	Example: Darlington bipolar transistor MJ11015	333
12.13.1	First step, download	335
12.13.2	Second step, automated symbol creation	336
12.13.3	Third step, symbol adaptation	336
12.14	Example: Operational amplifier TL071.	339
12.15	How to create a subcircuit?	344
12.16	Creation steps of a new circuit.	345
12.17	Illustrated example of creating a subcircuit	346
13	VOLTAGE AND CURRENT SOURCES EDITOR	353
13.1	Two types of sources and two editors	353
13.2	Two types of sources: dependent or independent.	355
13.3	All simulations require independent sources	355
13.3.1	Voltage or current sources must be adapted to the requirements of each type of simulation.	355
13.4	How to place a source on a schematic	356
13.4.1	Three independent sources	357
13.4.2	Nine independent source, 6 linear sources and 3 non-linear sources.	358
13.4.3	Two dependent sources (obsolete)	358

13.5	Independent sources	359
13.6	V Independent voltage source	359
13.6.1	PULSE tension source	360
13.6.2	SINE voltage source	360
13.6.3	EXP voltage source (exponential)	361
13.6.4	Frequency modulated voltage source (SFFM)	361
13.6.5	Voltage arbitrary source modulated by PWL	362
13.6.6	Voltage source modulated by a .wav file	362
13.7	I Independent current source	363
13.7.1	PULSE current source	363
13.7.2	SINE current source (sinusoidal)	363
13.7.3	EXP current source (exponential)	364
13.7.4	Frequency modulated current source (SFFM)	364
13.7.5	Modulated current sources	365
13.8	Load Independent active load	367
13.9	Independent sources editor	367
13.9.1	Independent source frequency sweep configuration for an AC simulation	370
13.9.2	Configuration of independent sources for a DC simulation (small amplitudes)	371
13.9.3	Configuration of independent sources for an AC simulation (small amplitudes)	375
13.9.4	Configuration of independent sources for transient simulation (high amplitude)	379
13.10	Independent sources	401
13.11	E Voltage controlled voltage sources	401
13.11.1	First model: The transfer function is a constant value	402
13.11.2	Second model: The transfer function is a table of couples of values	404
13.11.3	Third model: Transfer function is a Laplace transform and is a function of S	405
13.12	F Current controlled current source	407
13.12.1	Example	408
13.13	G Voltage controlled current source	412
13.13.1	First model	412
13.13.2	Second model	413
13.13.3	Third model	413
13.14	H Current controlled voltage source	413
13.15	B Non-linear arbitrary voltage source	414
13.15.1	For an arbitrary voltage source	416
13.16	B Non-linear arbitrary current sources	419
13.17	Epoly Non-linear polynomial voltage source	420
13.18	Gpoly Non-linear polynomial current source	421
13.19	Attributes editor for dependent sources	423
14	PASSIVE COMPONENTS	425
14.1	Passive components	425
14.1.1	Preamble to the use of component model parameters	425
14.2	R – Resistor (one model)	426
14.3	C – Capacitor (two models)	429
14.3.1	First model of standard capacitor	429

14.3.2	Second model of capacitor	431
14.4	L, Inductor	432
14.4.1	First inductor model (linear without saturation)	432
14.4.2	Second model (non-linear).	434
14.4.3	Third model: CHAN (non- linear with saturation and hysteresis taken into account)	435
14.5	Hysteresis cycle	437
14.6	Differences between inductors with and without magnetic circuit	439
14.6.1	Wound inductor without magnetic circuit	439
14.6.2	Wound inductor with magnetic circuit.	440
14.7	K Transformers (mutual inductance).	441
14.8	Mutual inductance with several windings	442
14.9	Other ways to make a transformer with saturation and hysteresis	443
15	SEMI-CONDUCTOR COMPONENTS	444
15.1	Semi-conductor components	444
15.1.1	How to choose a component model	444
15.2	D Diode (three models).	445
15.2.1	First standard model of diode	445
15.2.2	Second diode model	446
15.2.3	Power parameters common to both models.	447
15.3	Q Bipolar transistor (three models: Ebers-Moll, Gummel-Poon and VBIC)	448
15.3.1	First (Ebers-Moll) and second (Gummel-Poon) model	449
15.3.2	Third model (VBIC)	451
15.4	J JFET transistor (one model)	456
15.5	M Monolithic MOSFET (several models)	457
15.5.1	Monolithic MOSFET.	458
15.5.2	MOSFET transistor models	459
15.6	M Double vertical diffusion MOFSET (one model)	462
15.7	Z MESFET transistor (one model)	466
16	ACCESSORY COMPONENTS	467
16.1	Other accessory components.	467
16.1.1	Preamble to the use of component model parameters	467
16.2	S – Voltage controlled switch (two models).	468
16.2.1	Standard model Level=1	470
16.2.2	Second complete model Level=2	471
16.3	W Current controlled switch (three models).	472
16.3.1	First standard model.	473
16.4	O Lossy transmission line (one model)	476
16.5	T. Lossless transmission line (one model)	478
16.6	U RC transmission line (one model)	479
16.7	A. Special functions	480
16.7.1	Special functions INV, BUF, AND, OR, XOR.	481
16.7.2	Special functions SCHMITT, SCHMTBUF, SCHMTINV, DIFFSCHMITT, DIFFSCHMITTINV and DIFFSCHMITTBUF	483

16.7.3	Special functions DFLOP and SRFLOP	483
16.7.4	Special function PHIDET	484
16.7.5	Special function VARISTOR	484
16.7.6	Special function MODULATE	485
16.7.7	Special function SAMPLE	485
16.8	X Calling a subcircuit	486
17	INDUCTOR, HYSTERESIS CYCLE, TRANSFORMER AND MUTUAL INDUCTANCE	488
17.1	Interest in using a magnetic circuit	488
17.1.1	Operation of a magnetic circuit	488
17.1.2	Some useful definitions	488
17.2	Paths on the hysteresis cycle	490
17.2.1	First magnetisation curve (dotted line)	490
17.2.2	Path of the hysteresis cycle (full line)	490
17.3	Measurements of inductance, magnetic field and induction	491
17.3.1	The CHAN model (saturation and hysteresis)	492
17.3.2	Inductance measurement	495
17.3.3	Measurement of the magnetic induction flux density	496
17.4	Three examples of hysteresis cycles	498
17.5	Hysteresis cycle with airgap	501
17.6	Hysteresis cycle with several values of H	502
17.7	Hysteresis cycle with continuous polarisation	503
17.8	Presentation of four LTspice IV transformer models	504
17.9	First two transformer models without consideration of saturation and hysteresis	505
17.10	Four values are necessary for models 1 and 2	506
17.11	Two important values, the coupling coefficient K and the transformation ratio N	506
17.12	Two equivalent schematics for models 1 and 2	507
17.13	Transformer model 1, $K=1$ and explicit leakage inductance	508
17.14	Transformer model 2: K different from 1 and implied leakage inductance (calculated by LTspice IV)	508
17.15	Case of transformers consisting of several windings	510
17.16	Determination of a transformer according to the characteristics of an SMPS	511
17.17	Our choice of transformer	513
17.18	Calculations of the model's values from measurements or characteristics	513
17.19	Models 1 and 2 of the transformer	514
17.20	Schematic of the SMPS with transformer n°1	515
17.21	Schematic of the SMPS with transformer n°2	519
17.22	Conclusions about these two methods	521
17.22.1	Case of transformers with multiple windings	523
17.23	Saturation problems of the transformer	524
17.24	Transformer model n°3 (with saturation and hysteresis)	527
17.25	Transformer model No. 3 made with a subcircuit	527
17.26	Subcircuits with only one secondary	529
17.27	Transformer subcircuit with several secondaries	533
17.28	Integration of the subcircuit (transformer model No. 3) in an SMPS	537

17.29	Setting of a snubber (overvoltage clipper)	540
17.30	Exceeding a component's characteristics	544
17.31	Similitude between simulation results and measurements taken on a wired prototype	547
17.32	Conclusions for the similitude between simulation and real measurements.	551
18	CONTROL PANEL AND KEYBOARD SHORTCUTS.	552
18.1	Presentation of the control panel in nine tabs	552
18.2	Compression tab (options related to data compression)	553
18.3	Save Default tab (options related to saving).	555
18.4	SPICE tab (LTspice IV simulation core operating options)	557
18.4.1	Precautions concerning SPICE configuration	557
18.4.2	Simulation calculation control parameters	558
18.4.3	Integration method control parameters	559
18.4.4	Solver control parameters	559
18.5	Drafting options tab (drafting options).	561
18.6	Netlist option tab (Netlist syntax or writing options)	565
18.7	Waveform tab (waveform viewer or virtual oscilloscope)	567
18.8	Operation tab (general LTspice IV configuration)	571
18.9	Hacks tab (internal operation of LTspice IV)	573
18.9.1	Precautions regarding Hacks! configuration.	573
18.9.2	Hacks! control parameters.	574
18.10	Web tab (internet connection)	575
18.11	Keyboard shortcuts configuration	577
18.11.1	Interactivity of the schematic editor	578
18.11.2	Schematic editor keyboard shortcuts	578
18.12	Colours configuration (colour preferences)	579
19	SOME EXAMPLES.	580
19.1	Characteristic network trace of a semiconductor component.	580
19.1.1	Characteristics of an N-Channel JFET, the 2N3819	580
19.1.2	Characteristics of an N-Channel bipolar transistor, the 2N2222.	584
19.1.3	Evolution of characteristics with temperature.	584
19.1.4	Characteristics of a Zener diode according to temperature	585
19.2	Amplifier circuit	587
19.2.1	Amplifier specifications sheet	587
19.2.2	Amplifier assembly	587
19.2.3	Verification of the circuit's component values	590
19.2.4	Yield	591
19.3	Average power	592
19.3.1	Harmonic distortion measurements	593
19.3.2	Tracing the FFT curve	597
19.3.3	Intermodulation distortion measurements	598
19.3.4	Response to a square signal	600
19.3.5	Let's trace the Bode diagram.	603
19.3.6	Noise generated by the amplifier	604

19.3.7	Transfer function of this amplifier	605
19.4	Bode diagram of a regulation loop (SMPS application)	607
19.4.1	Disadvantages of the classic methods	608
19.4.2	Advantages of the new method	608
19.4.3	Gain Bode diagram	609
19.4.4	Impedance Bode diagram	612
19.5	A simple wattmeter, application of a source B	616
19.6	Parametric analysis of an RLC circuit	618
19.7	Incorporation of a wiring BUS	621
19.8	DC/DC Converter (SMPS)	624
19.8.1	Use of a non-saturable air inductor	628
19.8.2	Use of an inductor with a saturated magnetic circuit	628
19.8.3	Use of an inductor with a non-saturated magnetic circuit	630
19.8.4	DC/DC converter efficiency report	631
19.9	Analysis according to the dispersion of component values according to the Monte Carlo method.	633
20	QUESTIONS AND ANSWERS	637
20.1	What is the impact of the computer on the calculation time of a simulation in LTspice IV?	637
20.1.1	Three examples of laptop computers	637
21.1.2	Four examples of desktop computers	637
20.2	What are the limits of LTspice IV ?	638
20.3	Is LTspice IV really helpful?	643
20.4	How to retrieve the list of a circuit's components?	644
20.5	How to easily toggle between schematic pages?	644
20.6	How to copy/paste part of a circuit from one schematic page to another?	644
20.7	What are the most common mistakes made when using LTspice IV?	644
20.8	What hints and tips can save time?	645
20.9	In which situation can LTspice IV freeze?	645
20.10	Can LTspice IV be installed on any computer?	646
20.11	Which concrete help does LTspice IV provide in terms of electronic circuits simulation?	646
20.12	Are many steps required for a simulation using LTspice IV?	646
20.13	How helpful is LTspice IV in terms of measurements?	647
20.14	Why does LTspice IV allow the testing of more solutions?	647
20.15	Is there a risk of becoming addicted to LTspice IV?	647
20.16	Is LTspice IV really useful to learn about electronics?	648
20.17	You cannot find the indicated menus or their content is different from what you expected?	648
20.18	What flags are associated with the launch of LTspice IV?	649
20.19	Which actions allow a simulation to be carried out?	650
20.20	Are LTspice IV SMPS circuit models compatible with other SPICE software versions?	651
20.21	Where can we find reliable information, models and application examples for LTspice IV users?	651
20.22	Is there a Linux version of LTspice IV?	651

21	LTSPICE MODELS OF INDUCTORS AND TRANSFORMERS	652
21.1	Content of the Online Model Package	653
21.2	SMPS design and development tools	653
21.3	Presentation of the WE-FLEX and WE-FLEX+ transformers range	654
21.4	The LTspice CHAN inductor model	656
21.5	A sufficient precision with the CHAN model	661
21.6	Advantages of the LTspice CHAN model	664
21.7	First magnetisation curve	665
21.8	Similarities between simulation and measurements on the tabletop prototype	668
21.9	Nominal current I_n depends on the windings wiring	669
21.10	How is saturation visible in LTspice IV?	669
21.11	Two modelling methods of inductors	670
21.12	Three methods to model transformers	672
21.13	WE-FLEX and WE-FLEX+ transformer model	676
21.14	Modelling methods equivalencies, outside the saturation zone	678
21.15	Content of the Online Model Package and method of using these WE-FLEX transformer models	679
21.16	To make a transformer or a inductor, configure the value of AG	684
21.17	To make a transformer using the WE-FLEX model, the values of PR, PL, SR and SL must also be configured	686
21.18	To make a inductor using the WE-FLEX model, the values of IR and IL must also be configured	688
21.19	Tables S2 and T2: A precious help to choose the inductor or the transformer for your SMPS	689
21.20	A complete and illustrated example of an LTspice IV WE-FLEX model used for a Flyback SMPS	692
21.21	Transformers and inductors LTspice models	699
21.22	How to obtain the value of the magnetic material losses?	702
21.23	Windings wiring of LTspice CHAN WE-FLEX and WE-FLEX+ models	704
21.24	Series and/or parallel winding of the WE-FLEX range	708
21.25	Limits of the LTspice CHAN model for transformers	710
21.26	The value of a inductor made with a magnetic circuit varies according to the current flowing through it	711
21.27	Frequently asked questions	713
APPENDICES		723
1	Values of L_m and A to be used in the CHAN model	723
2	Values of B_s , B_r and H_c to be used in the CHAN model	724
2.1	Parameters directly useable for the CHAN model by LTspice IV	724
BIBLIOGRAPHY		733
INDEX		735