



THE LTSPICE IV
SIMULATOR

MANUAL, METHODS
AND APPLICATIONS

THE LTSPICE IV SIMULATOR

GILLES BROCARD

MANUAL, METHODS
AND APPLICATIONS

Preface by Mike Engelhardt

1st edition

1st edition

Swirdoff Verlag
ISBN: 978-3-89929-258-9

Preface

It is an honor to write a preface for Gilles Brocard. I appreciate his work writing this book and hope you benefit from his labors.

LTspice has been fun to write. It let me implement a number of numerical methods that make LTspice better than traditional SPICE programs: a new numerical integration method, node reduction, a native circuit element that behaves like a power MOSFET, and new time step size control to name a few.

The biggest recent advance in LTspice was when it went multi-threaded in 2008. We found it easy to distribute the computations over multiple cores but challenging to make the simulation actually run faster. The problem was that the LTspice object code had been so optimized (much had already been implemented in optimized assembly language) that it didn't take very many microseconds per timestep and that was a short time compared to how well one can synchronize multiple threads. That's when we developed a means to dynamically adjust each threads' cache size to stochastic cool the threads to keep the work load spread evenly. Another important technique introduced at that time was code generation that generates an assembly listing optimized for your circuit. Then that code is assembled and linked by LTspice for execution. This self-authoring code is generated typically every few seconds during the simulation to help your circuit execute close to the theoretical flop limit of a modern CPU. That's why LTspice IV is fast.

But all this is for a purpose. I believe SPICE has impacted mankind more than any other simulator. Writing a better SPICE is important. LTspice offers you the ability to rapidly prototype your designs so that you understand them better and even develop intuition.

Mike ENGELHARDT
Manager of Simulation Development
Linear Technology Corporation
April 2011

Index

Preface	5
Index	7
Foreword	21
1 LTSPICE IV: PRESENTATION AND HISTORIC	25
1.1 Circuit simulation with LTspice IV	25
1.1.1 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 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 can't LTspice IV do?	31
1.6 Conclusion	32
2 FILES SUPPLIED WITH LTSPICE IV	33
2.1 Installation of LTspice IV	33
2.1.1 Required configuration	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, sub-circuits, macro models, and component libraries	37
2.4.2 Application examples	37
2.5 LTspice IV file extensions	42
3 WORKING OF 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	83
3.4.10	To conclude this first approach	85
4	SCHEMATICS EDITOR	86
4.1	The commands of LTspice IV	86
4.2	Schematics editor	87
4.2.1	File menu	88
4.2.2	Edit menu	90
4.2.3	Hierarchy Menu	92
4.2.4	View Menu	93
4.2.5	Simulate Menu	95
4.2.6	Tools Menu	96
4.2.7	Window Menu	96
4.2.8	Help Menu	96
4.2.9	Schematic editor pop-up menu	97
4.3	Components databases	98
4.4	Draw a new schematic	100
4.4.1	Open the schematic editor	100
4.4.2	Place the first elements on the schematic	100
4.4.3	Main commands of the schematic editor	101
4.4.4	Connecting elements from the schematic	104
4.4.5	Enter the value or reference of a component	104
4.4.6	Enter the values of a component with the attribute editor	107
4.4.7	Enrich the schematic (optional)	108
4.4.8	Add simulation, source and directives (optional)	109
4.4.9	Save your schematic	110
4.4.10	Launch the simulation	110
4.5	Incorporation of a wiring BUS	110
4.6	Reminder of the schematic editor use rules	114
4.6.1	You must be very careful about two points:	115
4.7	Export a schematic	115
5	SYNTAX AND COMPONENTS EDITOR	116
5.1	General syntax rules in LTspice IV	116
5.2	Component values editor	118
5.3	Procedures to access usual or complex component models	122

5.3.1	Changing the standard values of a component	123
5.3.2	Display of attributes and modification of components' values	127
5.4	Allocation of the attributes editor fields	130
5.5	Display of a component's attributes using two models	131
6	SYMBOL EDITOR AND HIERARCHY	134
6.1	Symbol editor menu	134
6.1.1	File Menu	134
6.1.2	Edit Menu	134
6.1.3	Hierarchy Menu	135
6.1.4	Draw Menu	135
6.1.5	View Menu	135
6.1.6	Tools Menu	136
6.1.7	Window Menu	136
6.1.8	Help Menu	136
6.1.9	Symbol editor pop-up menu	136
6.2	First step: Drawing the symbol	136
6.3	Second step: Adding connection terminals	137
6.4	Third step: Adding or modifying attributes	138
6.5	Possible call-ups from a symbol	140
6.6	Visible attributes connected to the symbol	142
6.7	Automatic symbol generation from a section of the schematic	143
6.8	Automatic symbol generation from a Netlist	144
6.9	Hierarchy in LTspice IV	149
6.10	Hierarchy use rules	150
6.10.1	A method similar in concept to the Matryoshka dolls	150
6.11	Hierarchic construction rules	151
6.11.1	Levels hierarchy	152
6.12	Commands of the Hierarchy menu	153
6.13	Example: Work flow of a simple two-level hierarchic construction	154
6.13.1	Screen number 1: Sub-circuit model	154
6.13.2	Screen number 2: Secondary schematic	154
6.13.3	Screen number 3: Main schematic	155
6.13.4	Screen 4: Simulation of the main schematic	155
6.14	Export of the hierarchy directory	156
6.15	Interactivity between the lower and the higher level	156
7	NETLIST EDITOR	158
7.1	The origin of Netlists	158
7.2	The Netlist: A mandatory step	158
7.5	Structure, syntax and conventions of Netlist	158
7.6	An example of Netlist	159
7.7	Netlist editor menus	160
7.7.1	File Menu	160
7.7.2	View Menu	160

7.7.3	Simulate Menu	161
7.7.4	Netlist editor pop-up menu	161
7.8	Writing a Netlist	161
7.9	Syntax of Netlist files .CIR, .NET or .SP	162
7.10	How to open the Netlist editor from a schematic	163
7.11	Running a Netlist	166
7.12	Exporting the Netlist of a schematic	167
7.13	System commands used in Netlists	168
8	MEASUREMENTS, VIRTUAL OSCILLOSCOPE AND FFT EDITORS.	169
8.1	LTspice IV waveform viewer	169
8.1.1	Display of the simulation calculation results	169
8.1.2	How to select measurement points on your schematic?	169
8.2	How to display a measurement on the virtual oscilloscope?	169
8.2.1	Visualise a ground referenced voltage	169
8.2.2	Visualise a current	170
8.2.3	Visualise a differential voltage (not ground referenced).	170
8.2.4	Erase previous traces	171
8.2.5	Selectively erase one or more traces	171
8.2.6	Display instantaneous power dissipation	171
8.2.7	Display average power and energy integral of power over time displayed.	172
8.2.8	Display average voltage or current over the displayed period or efficient value (RMS)	172
8.3	Use of menus	173
8.3.1	Virtual oscilloscope and FFT analyser editor	173
8.3.2	File menu	174
8.3.3	View menu.	174
8.3.4	Plot Settings Menu (Configuration of the virtual oscilloscope)	175
8.3.5	Simulation Menu (Launch simulation)	177
8.3.6	Tools Menu	177
8.3.7	Window Menu	178
8.3.8	Help Menu	178
8.3.9	Virtual oscilloscope pop-up menu.	178
8.4	Choosing the displayed measurements	179
8.5	Add a trace or a screen.	181
8.5.1	Add a trace	181
8.5.2	Add a screen	183
8.6	Zoom functions	183
8.7	Mathematical operations in the virtual oscilloscope	184
8.8	Request the calculation of an algebraic expression	184
8.8.1	Modify the appearance of a trace.	185
8.9	User defined functions	189
8.10	Modify the axes scale	190
8.10.1	Vertical axis scales	190
8.10.2	Horizontal axis scales	191
8.11	Use of the virtual oscilloscope in X-Y mode	192

8.12	Pop-up menu and scales	193
8.13	Other scales configurations	194
8.13.1	Left vertical scale	194
8.13.2	Only display the phase	195
8.13.3	Left vertical scale	195
8.14	Display management of several traces on the virtual oscilloscope	197
8.15	Information on the virtual oscilloscope traces	199
8.16	Other traces customisation in the virtual oscilloscope.	201
8.17	Control of the virtual oscilloscope's colours	201
8.18	Two measurement cursors	203
8.18.1	Placing measurement cursors on traces	204
8.19	Display of coordinates in the bottom banner	206
8.20	Save the virtual oscilloscope configuration	208
8.21	Acceleration of file loading	208
8.22	RAM and addressing space	209
9	SIMULATIONS CONFIGURATIONS DIRECTIVES	211
9.1	Definition of a simulation directive	211
9.1.1	Simulation directives editor	211
9.1.2	Syntax of simulation directives	213
9.1.3	First syntax rule	213
9.1.4	Second syntax rule	213
9.1.5	Third syntax rule.	215
9.1.6	Never forget a mandatory parameter.	216
9.2	.Options parameters modifying the execution of a simulation	217
9.3	.IC Fix initial conditions for transient simulation.	220
9.4	.Savebias Save a DC operating point	222
9.5	.Loadbias Load a DC operating point	223
9.6	.Net Calculation of a network parameter with an AC simulation.	223
9.7	.Nodeset Initial conditions for DC analysis	224
10	THE SIX MAIN SIMULATIONS	225
10.1	Presentation of the six main simulations	225
10.1.1	DC simulations (direct)	225
10.1.2	AC Simulations (frequency)	226
10.1.3	Non-linear circuits simulations.	226
10.1.4	Simulations characteristics	226
10.2	Choice criteria regarding simulations	227
10.2.1	If the only excitation source of the circuit is a direct voltage source.	227
10.2.2	The only excitation source of the circuit is low amplitude alternative voltage.	228
10.2.3	The only excitation source of the circuit is a high amplitude alternative source (or any other causing the circuit's components non-linearity)	228
10.3	.OP – Simulation of a continuous polarisation point	229
10.4	.DC – DC source sweep analysis (one to three sources)	230
10.5	.TF – Transfer function simulation (gain, input and output impedance)	232

10.6	.AC – Simulation of an AC signal around polarization point	233
10.7	.NOISE – Noise simulation.	235
10.8	.TEMP – Temperature sweep simulation	236
10.9	.TRAN – Transient simulation (non-linear)	239
10.10	Configuration of the transient simulation .TRAN.	241
10.10.1	Be careful with the Maximum Timestep value	241
10.10.2	Parameter: .uic (transient simulation)	245
10.10.3	Parameter: startup (transient simulation).	246
10.10.4	Parameter: steady (transient simulation)	247
10.10.5	Parameter: .nodiscard (transient simulation).	249
10.10.6	Parameter: .step (transient simulation)	251
10.11	.FOUR – Edit harmonics as numeric format.	254
10.11.1	How does the FFT analysis work?	254
10.11.2	Conditions to fulfil to obtain a representative FFT analysis	258
10.11.3	Influence of Stop Time and Time step parameters on the FFT	270
10.12	Monte Carlo statistic simulations	272
10.12.1	First step.	274
10.12.2	Second step.	274
10.12.3	Third step	275
12.12.4	Comment on the Monte Carlo method.	275
10.13	Simulations configuration	276
11	NUMERICAL MEASUREMENTS, DOWNLOADS, BACKUP AND MODELS.	278
11.1	Retrieving measurements as numerical data	278
11.1.1	Retrieving measurement files as numerical data	278
11.1.2	Variables detection	278
11.2	MEAS – Display measurements values numerically	278
11.2.1	First type of measurements: For only one X-axis point	279
11.2.2	Examples of use of .meas for only one X-axis point	282
11.2.3	Second type of measurement: For an interval between two points on the X-axis	284
11.2.4	Examples of use of parameters rise, fall, last and cross	286
11.2.5	Case of a NOISE simulation	294
11.2.6	Creation of a measurement script: File_name.meas	294
11.2.7	Precision of results obtained with the command .meas	300
11.3	.PARAM – Variables & Parameters.	301
11.4	.STEP – Configurable intervals.	304
11.4.1	The commands .step and select steps, step by step	306
11.5	.FUNC – User functions	311
11.6	Efficiency report of a DC/DC convertor: steady.	312
11.7	.FERRET – Download a file online.	313
11.8	.GLOBAL General declaration	314
11.9	.SAVE Limitation of the quantity of saved data.	314
11.10	.WAVE – Transform the output signal in .wav	315
11.10.1	Comments on .wav files	316
11.11	Configuration of a component value with the command .param	316

11.12	.MODEL – Define a SPICE model	317
11.13	.SUBCKT – Define a sub-circuit	319
11.14	.INCLUDE – include a new library	320
11.15	.LIB – Models or sub-circuits library	321
11.15.1	Encrypted library	322
12	IMPORT OF COMPONENTS MODELS	323
12.1	Does LTspice IV need to download components models?	323
12.2	Macro-models and models	323
12.2.1	Macro-models or models file extensions: .MODEL or .MOD	323
12.3	Sub-circuits	324
12.4	Libraries and models	324
12.5	A component model consists of two elements	325
12.6	Symbols to call up components	325
12.7	Downloading a component model	325
12.8	Three extensions for three manners to add components	326
12.9	One symbol can call up several elements	326
12.10	Models libraries	327
12.10.1	How does the compiler detect that one component rather than another is used?	328
12.10.2	Each component has several possible models	329
12.11	Models	329
12.12	Example: Sub-circuits library 74htc.lib	329
12.3	Example: Darlington bipolar transistor MJ11015	330
12.13.1	First step, download	332
12.13.2	Second step, automated symbol creation	333
12.13.3	Third step, symbol adaptation	333
12.14	Example: Operational amplifier TL071	336
12.15	How to create a sub-circuit?	341
12.16	Creation steps of a new circuit	342
12.17	Illustrated example of a sub-circuit creation	343
13	VOLTAGE AND CURRENT SOURCES EDITOR	350
13.1	Two types of sources and two editors	350
13.2	Two types of sources: dependent or independent	352
13.3	All simulations require independent sources	352
13.3.1	Voltage or current sources must be adapted to the requirements of each type of simulation	352
13.4	How to place a source on a schematic	353
13.4.1	Three independent sources	354
13.4.2	Nine independent source, 6 linear sources and 3 non-linear sources	355
13.4.3	Two independent sources (obsolete)	355
13.5	Independent sources	356
13.6	∇ Independent voltage source	356
13.6.1	PULSE tension source	357
13.6.2	SINE voltage source	357
13.6.3	EXP voltage source (exponential)	358

13.6.4	Frequency modulated voltage source	358
13.6.5	Voltage arbitrary source modulated by PWL	359
13.6.6	Voltage source modulated by a .wav file	359
13.7	I Independent current source	360
13.7.1	PULSE current source	360
13.7.2	SINE current source	360
13.7.3	EXP current source (exponential)	361
13.7.4	Frequency modulated current source	361
13.7.5	Modulated current sources	362
13.8	Iload Independent active load	364
13.9	Independent sources editor	364
13.9.1	Independent source frequency sweep configuration for an Ac simulation	367
13.9.2	Configuration of independent sources for a DC simulation (small amplitudes)	368
13.9.3	Configuration of independent sources for an AC simulation (small amplitudes)	372
13.9.4	Configuration of independent sources for transient simulation (high amplitude)	376
13.10	Independent sources.	397
13.11	E Voltage controlled voltage sources	398
13.11.1	First model: The transfer function is a constant value.	399
13.11.2	Second model: The transfer function is a table of couples of values.	401
13.11.3	Third model: Transfer function is a Laplace transform and is a function of S	402
13.12	F Current controlled current source	404
13.12.1	Example	405
13.13	G Voltage controlled current source	409
13.13.1	First model	409
13.13.2	Second model	409
13.13.3	Third model	410
13.14	H Current controlled voltage source	410
13.15	B Non-linear arbitrary voltage source.	411
13.15.1	For an arbitrary voltage source	412
13.16	B Non-linear arbitrary current sources	416
13.17	Epoly Non-linear polynomial voltage source	417
13.18	Gpoly Non-linear polynomial current source	418
13.19	Attributes editor for dependent sources	420
14	PASSIVE COMPONENTS	422
14.1	Passive components	422
14.1.1	Preamble to the use of component model parameters	422
14.2	R – Resistor (one model)	423
14.3	C – Capacitor (two models)	426
14.3.1	First model of capacitor.	426
14.3.2	Second model of capacitor	428
14.4	L. Inductor.	429
14.4.1	First inductor model (linear without saturation)	429
14.4.2	Second model (non-linear).	431
14.4.3	Third model: CHAN (non- linear with saturation and hysteresis taken into account)	432

14.5	Hysteresis cycle	434
14.6	Differences between inductors with and without magnetic circuit	436
14.6.1	Winded inductor without magnetic circuit	436
14.6.2	Winded inductor with magnetic circuit	437
14.7	K Transformers (mutual inductance)	438
14.8	Mutual inductance with several windings	439
14.9	Other ways to make a transformer with saturation and hysteresis	440
15	SEMI-CONDUCTOR COMPONENTS	441
15.1	Semi-conductor components	441
15.1.1	How to choose a component model	441
15.2	D Diode (three models)	442
15.2.1	First standard model of diode	442
15.2.2	Second model of diode	443
15.2.3	Power parameters common to both models	444
15.3	Q Bipolar transistor (three models: Ebers-Moll, Gummel-Poon and VBIC)	445
15.3.1	First (Ebers-Moll) and second (Gummel-Poon) model	446
15.3.2	Third model (VBIC)	448
15.4	J JFET transistor (one model)	453
15.5	M Monolithic MOSFET (several models)	454
15.5.1	Monolithic MOSFET	455
15.5.2	MOSFET transistor models	456
15.6	M Double vertical diffusion MOSFET (one model)	459
15.7	Z MESFET transistor (one model)	463
16	ACCESSORY COMPONENTS	464
16.1	Other accessory components	464
16.11	Preamble to the use of component model parameters	464
16.2	S – Voltage controlled switch (two models)	465
16.2.1	First standard model Level=1	467
16.2.2	Second complete model Level=2	468
16.3	W Current controlled switch (three models)	469
16.3.1	First standard model	470
16.4	O Lossy transmission line (one model)	473
16.5	T. Lossless transmission line (one model)	475
16.6	U RC transmission line (one model)	476
16.7	A. Special functions	477
16.7.1	Special functions INV, BUF, AND, OR, XOR	478
16.7.2	Special functions SCHMITT, SCHMTBUF, SCHMTINV, DIFFSCHMITT, DIFFSCHMITTINV and DIFFSCHMITTBUF	480
16.7.3	Special functions DFLOP and SRFLOP	480
16.7.4	Special function PHIDET	481
16.7.5	Special function VARISTOR	481
16.7.6	Special function MODULATE	482

16.7.7	Special function SAMPLE	482
16.8	X Calling up a sub-circuit	483
17	SATURABLE INDUCTANCE, HYSTERESIS CYCLE, TRANSFORMER AND MUTUAL INDUCTANCE	485
17.1	Interest of using a magnetic circuit	485
17.1.1	Operation of a magnetic circuit	485
17.1.2	Some useful definitions	485
17.2	Paths on the hysteresis cycle.	487
17.2.1	First magnetisation curve (dotted line)	487
17.2.2	Path of the hysteresis cycle (full line)	487
17.3	Measurements of inductance, magnetic field and induction	488
17.3.1	The CHAN model (saturation and hysteresis)	489
17.3.2	Inductance measurement	492
17.3.3	Measurement of the magnetic induction flow density	493
17.4	Three examples of hysteresis cycles.	495
17.5	Hysteresis cycle with airgap.	498
17.6	Hysteresis cycle with several values of H	499
17.7	Hysteresis cycle with continuous polarisation	500
17.8	Presentation of four LTspice IV transformer models	501
17.9	First two models of transformers without consideration of saturation and hysteresis	502
17.10	Four values are necessary for models 1 and 2.	503
17.11	Two important values, the coupling coefficient K and the transformation ratio N	503
17.12	Two equivalent schematics for models 1 and 2	504
17.13	Transformer model 1, $K=1$ and explicit leakage inductance	505
17.14	Transformer model 2: K difference from 1 and implied leakage inductance (calculated by LTspice IV)	505
17.15	Case of transformers consisting of several windings	507
17.16	Determination of a transformer according to the characteristics of an SMPS	508
17.17	Our choice of transformer	510
17.18	Calculations of the model's values from measurements or characteristics	510
17.19	Models 1 and 2 of the transformer.	511
17.20	Schematic of the SMPS with transformer n°1	512
17.21	Schematic of the SMPS with transformer n°2	516
17.22	Conclusions about these two methods	518
17.22.1	Case of transformers with multiple windings	520
17.23	Saturation problems of the transformer.	521
17.24	Transformer model n°3 (with saturation and hysteresis)	524
17.25	Transformer model n°3 made with a sub-circuit	524
17.26	Sub-circuits with only one secondary	526
17.27	Transformer sub-circuit with several secondaries.	530
17.28	Integration of the sub-circuit (transformer model n°3) in a SMPS	534
17.29	Setting of a snubber (overvoltage clipper)	537
17.30	Exceeding a component's characteristics	541

17.31	Similitude between simulation results and measurements taken on a wired prototype	544
17.32	Conclusions on the similitude between simulation and real measurements	548
18	CONTROL PANEL AND KEYBOARD SHORTCUTS	549
18.1	Presentation of the control panel in nine tabs	549
18.2	Compression tab (options related to data compression)	550
18.3	Save Default tab (options related to saving)	552
18.4	SPICE tab (LTspice IV simulation core operating options)	554
18.4.1	Precautions concerning SPICE configuration	554
18.4.2	Simulation calculation control parameters	555
18.4.3	Integration method control parameters	556
18.4.4	Solver control parameters	556
18.5	Drafting options tab (drawing options)	558
18.6	Netlist option tab (Netlist syntax or writing options)	562
18.7	Waveform tab (waveform viewer or virtual oscilloscope)	564
18.8	Operation tab (general LTspice IV configuration)	568
18.9	Hacks tab (internal operation of LTspice IV)	570
18.9.1	Precautions regarding Hacks! configuration	570
18.9.2	Hacks! control parameters	571
18.10	Internet tab (internet connection)	572
18.11	Keyboard shortcuts configuration	574
18.11.1	Interactivity of the schematic editor	575
18.11.2	Schematic editor keyboard shortcuts	575
18.12	Colours configuration (colour preferences)	576
19	A FEW EXAMPLES	577
19.1	Characteristic network trace of a semi-conductor component	577
19.1.1	Characteristics of a N-Channel JFET, the 2N3819	577
19.1.2	Characteristics of a N-Channel bipolar transistor, the 2N2222	581
19.1.3	Evolution of characteristics with temperature	581
19.1.4	Characteristics of a Zener diode according to temperature	582
19.2	Amplifier circuit	584
19.2.1	Amplifier specifications sheet	584
19.2.2	Amplifier assembly	584
19.2.3	Verification of the circuit's component values	587
19.2.4	Yield	588
19.3	Average power	589
19.3.1	Harmonic distortion measurements	590
19.3.2	Tracing the FFT curve	594
19.3.3	Intermodulation distortion measurements	595
19.3.4	Response to a square signal	597
19.3.5	Let's trace the Bode diagram	600
19.3.6	Noise generated by the amplifier	601
19.3.7	Transfer function of this amplifier	602

19.4	Bode diagram of a regulation loop (SMPS application)	604
19.4.1	Inconvenient of the standard methods	605
19.4.2	Advantages of the new method	605
19.4.3	Gain Bode diagram	606
19.4.4	Impedance Bode diagram	609
19.5	A simple wattmeter, application of a source B	613
19.6	Parametric analysis of an RLC circuit	615
19.7	Incorporation of a wiring BUS	618
19.8	DC/DC Converter (SMPS).	621
19.8.1	Use of a non-saturable air inductance	625
19.8.2	Use of self-inductance with a saturated magnetic circuit	625
19.8.3	Use of self-inductance with a non-saturated magnetic circuit	627
19.8.4	DC/DC converter efficiency report	628
19.9	Analysis according to the dispersion of component values according to the Monte Carlo method.	630
20	QUESTIONS AND ANSWERS	634
20.1	What is the impact of the computer on the calculation time of a simulation in LTspice IV?	634
20.1.1	Three examples of laptop computers	634
21.1.2	Four examples of desktop computers	634
20.2	What are the limits of LTspice IV ?	635
20.3	Is LTspice IV really helpful?	640
20.4	How to retrieve the list of a circuit's components?	641
20.5	How to easily toggle between schematic pages?	641
20.6	How to copy/paste part of a circuit from one schematic page to another?	641
20.7	What are the most common mistakes made when using LTspice IV?	641
20.8	What hints and tips can save time?	642
20.9	In which situation can LTspice IV freeze?	642
20.10	Can LTspice IV be installed on any computer?	643
20.11	Which concrete help does LTspice IV provide in terms of electronic circuits simulation?	643
20.12	Do many steps are required for a simulation with LTspice IV?	643
20.13	How helpful is LTspice IV in terms of measurements?	644
20.14	Why does LTspice IV allow the testing of more solutions?	644
20.15	Can one become addicted to LTspice IV?	644
20.16	Is LTspice IV really useful to learn about electronics?	645
20.17	You cannot find the indicated menus or their content is different from what you expected?	645
20.18	What flags are associated to the launch of LTspice IV?.	646
20.19	Which actions allow a simulation to be carried out?	647
20.20	Are LTspice IV SMPS circuit models also compatible with other SPICE software versions?	648
20.21	Where can we find reliable information, models and application examples for LTspice IV users?	648
20.22	Is there is Linux version of LTspice IV?	648

ANNEXES	649
1 Values of L_m and A to be used in the CHAN model	649
2 Values of B_s , B_r and H_c to be used in the CHAN model	650
2.1 Parameters directly useable for the CHAN model by LTspice IV	650
Bibliographie	659
Hier geht es weiter mit dem Text von der französischen Seite 609	659
Index	663
Symboles	663

1 LTSPICE IV: PRESENTATION AND HISTORIC

1.1 Circuit simulation with LTspice IV

1.1.1 Three basic steps

With LTspice IV, circuit simulation is easy as 1, 2, 3 (see fig.1.1):

- **First step: Create the schematic and choose the type of simulation.** With the **schematics editor**, you can draw your circuit and add your comments. LTspice IV offers a wide range of components, however, if one or more models of components are missing, you can download them online. (See chapter 12 for more information about this).
Depending on your requirements (continuous, alternative or transient analysis), choose a simulation directive and add the relevant source (see chapters 9 to 11 and 13). You can also add other commands (simulation directives) and configure all the elements, components values, etc.
- **Second step: Run the simulation.** You are asking the computer to carry out all calculations required to perform the chosen simulation, this is called running the simulation. This step takes a little time for all calculations to be done.
- **Third step: Take the measurements.** You choose the measures you require (current, voltage, frequency, power, etc.). The measurements are displayed instantly in a screen looking like the display of a high-performance digital oscilloscope. You can configure the display parameters in the same manner as you would on a real oscilloscope (reframing, change of scale, zoom, colour choice, screen fractioning, traces multiplication).

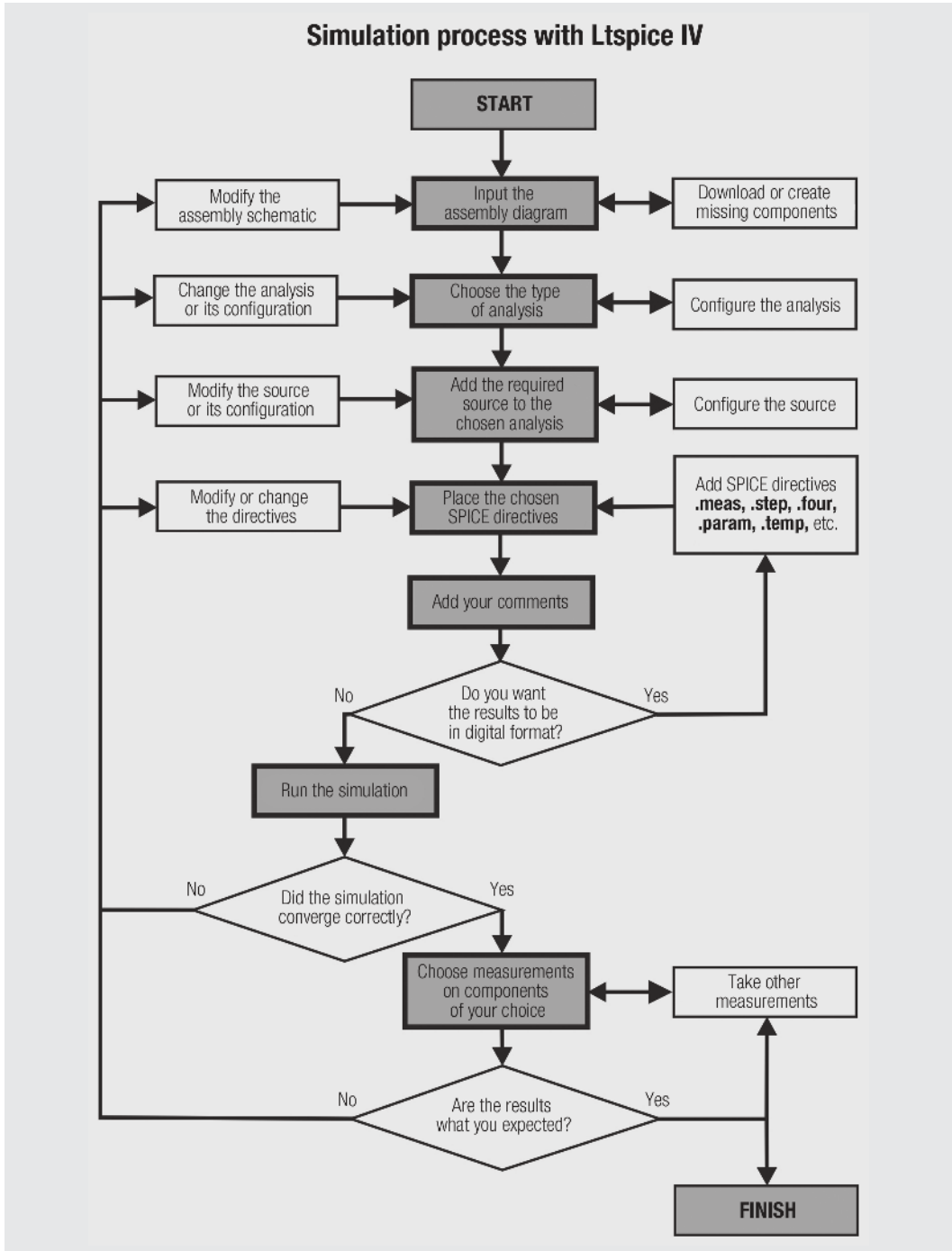



Figure 1.1

3.4 A detailed example, step by step

2.  You create a new page for your schematic by clicking on the red icon on white background in the top left corner. You will see the background turn a lighter shade of grey and all 37 icons are now active.

The number of menus increases as well and their content is now relevant to the schematics editor you have just accessed.

LTspice IV is now waiting for you to either draw or open a schematic. The complete description of the new menus of the schematic editor can be found in chapter 4.

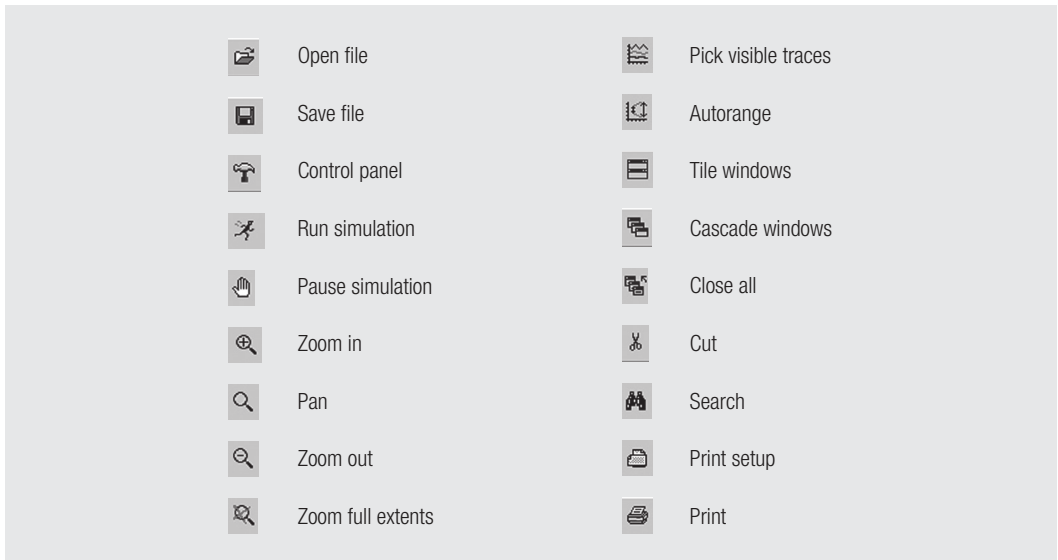


Figure 3.4

3 WORKING OF LTSPICE IV AND FIRST EXAMPLE

3.4 A detailed example, step by step

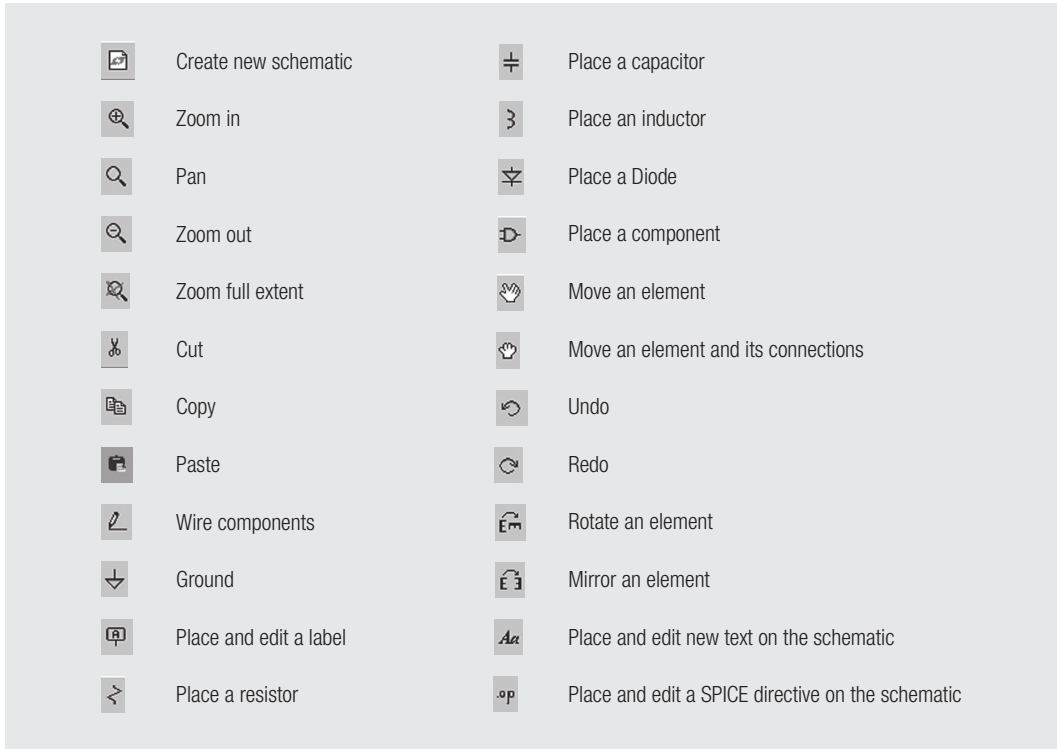


Figure 3.5

3.4.1 Drawing a schematic

3. We are not going to open a saved schematic, but we are going to create our own in order to go through all the creation steps. To start with, let's position on our schematic an operational amplifier symbol. To do so, click on

the icon representing the **AND** logic gate symbol . Click once on this icon and another window appears. The left-hand column contains a series of words in brackets. In this column, double-click on **[Opamps]** to access the operational amplifier directory. (See figure 3.6).

3.4 A detailed example, step by step

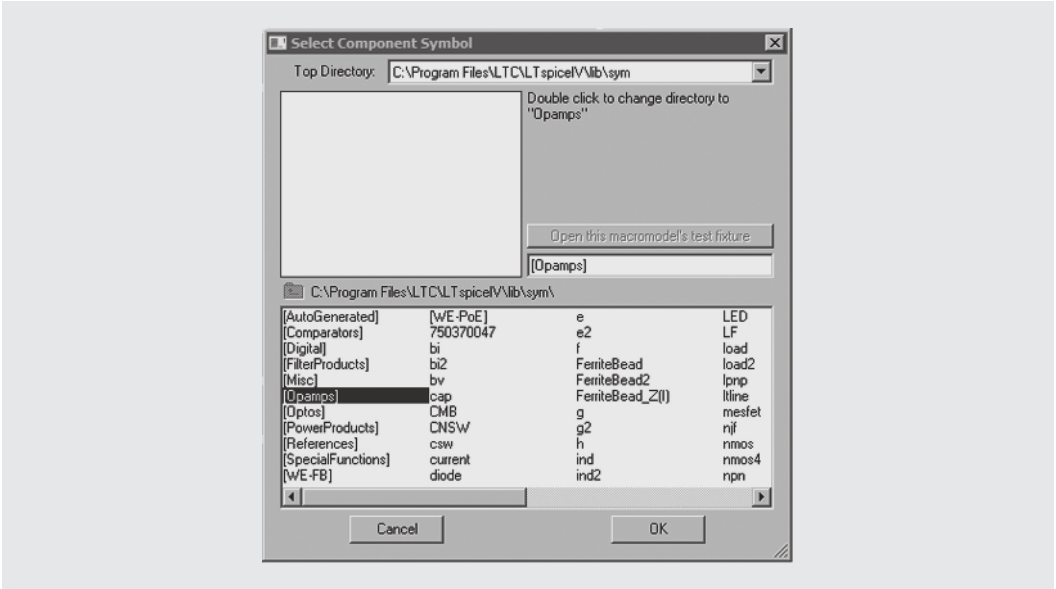


Figure 3.6

A list of operational amplifier appears, scroll to the very end of this list and double-click on **UniversalOpamp2**, which is currently the last item in the list. It is a standard operational amplifier with average performances. (Figure 3.7).

4. You automatically come back to the schematic page and your cursor is now shaped like an operational amplifier symbol. Go to the centre of the page and click left to drop the symbol on the schematic, figure 3.8. Another copy of this symbol appears straight away and follows the movement of your mouse.

3 WORKING OF LTSPICE IV AND FIRST EXAMPLE

3.4 A detailed example, step by step

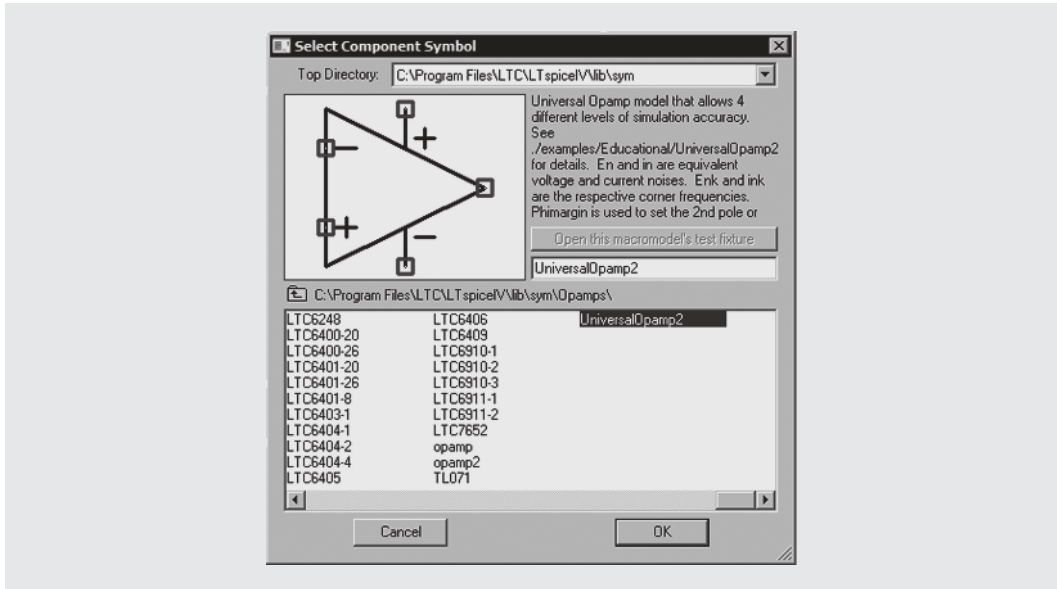


Figure 3.7

To get rid of it (we only want to use one operational amplifier on this schematic), right click anywhere, and the symbol disappears. So at the moment, the only item on our schematic is the operational amplifier in the centre.

3.4 A detailed example, step by step

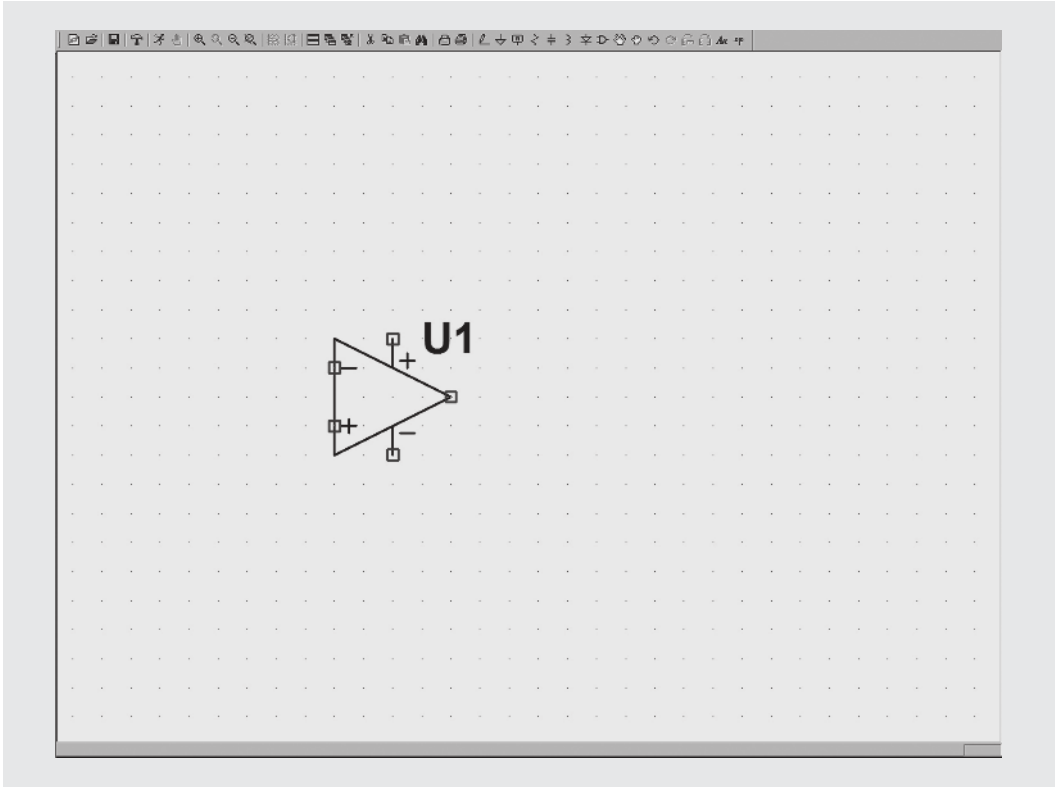



Figure 3.8

5. We are now going to add two transistors to our schematic. Once again, click on the **AND** icon  , you can see that you are still in the operational amplifiers directory.

So click on the yellow folder icon on the left to go back up one level in the directory.

We are now in the standard component directory, in the 4th column, you'll see the word **npn**, double-click on it, see figure 3.9.

As before, you find yourself straight back in the schematic page where you can place your NPN bipolar transistor by following the same procedure as for the amplifier: Click to drop, then, right-click to remove the copy.

3 WORKING OF LTSPICE IV AND FIRST EXAMPLE

3.4 A detailed example, step by step

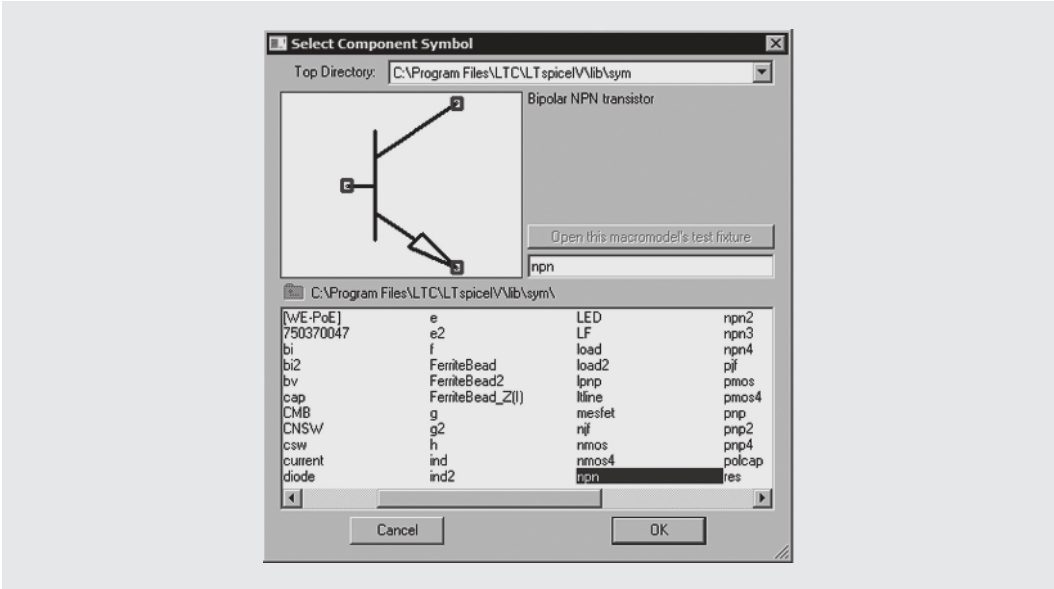







Figure 3.9

6. If you're not happy with the location of a component, you can remove it with . Click on the scissors, then click on the component you wish to remove, and lastly, right-click to exit the cut function. You can also use the open hand  to move a component. Click on the hand, then click on the component, it changes colour, move your mouse to place it where you want and click again to drop it, then right-click to exit the move function. You can also click and drag your mouse to draw a box around the section to move, then, move the box, click to drop it and right-click to exit.
7. We are now going to add another PNP bipolar transistor. Click on **AND** , and double-click on the word **pnp**, figure 3.10. Position the symbol as you see it on the schematic, but do not click to drop it, at this stage, you only have the outline of the symbol. Then to flip the symbol, you must use **twice** the rotation tool (command **Ctrl + R**) or  then once the symmetry tool (command **Ctrl + E**) or , figure 3.11. Once the symbol has the required orientation, click to drop it.