WÜRTH ELEKTRONIK

THE LTSPICE IV SIMULATOR

MANUAL, METHODS AND APPLICATIONS

WÜRTH ELEKTRONIK

THE LTSPICE IV SIMULATOR

GILLES BROCARD

MANUAL, METHODS AND APPLICATIONS

Preface by Mike Engelhardt

1st edition

1st edition

Swiridoff 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 V 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 Load 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 indictor 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 MOFSET (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).

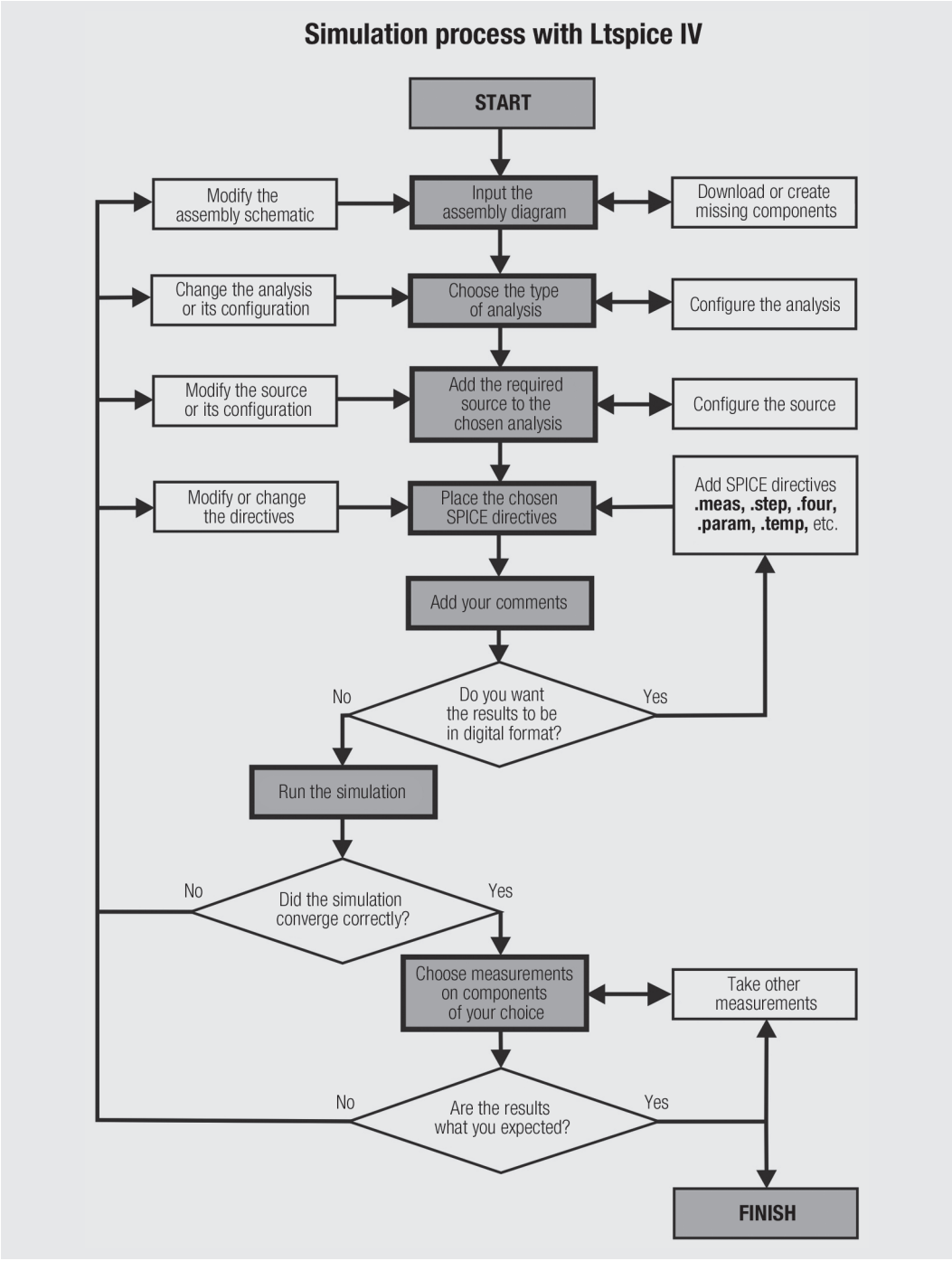
Simulation process with Ltspice IV
START
Input the assembly diagram
Modify the assembly schematic
Download or create missing components
Choose the type of analysis
Change the analysis or its configuration
Configure the analysis
Add the required source to the chosen analysis
Modify the source or its configuration
Configure the source
Place the chosen SPICE directives
Modify or change the directives
Add SPICE directives .meas, .step, .four, .param, .temp, etc.
Add your comments
Do you want the results to be in digital format?
No
Yes
Run the simulation
Did the simulation converge correctly?
No
Yes
Choose measurements on components of your choice
Take other measurements
Are the results what you expected?
No
Yes
FINISH

Figure 1.1

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.

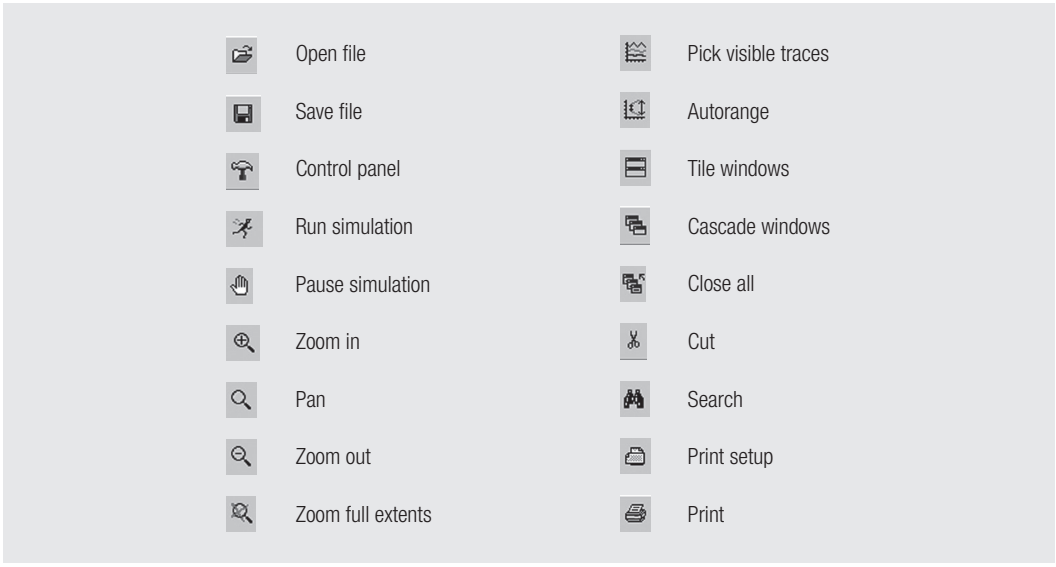

Figure 3.4

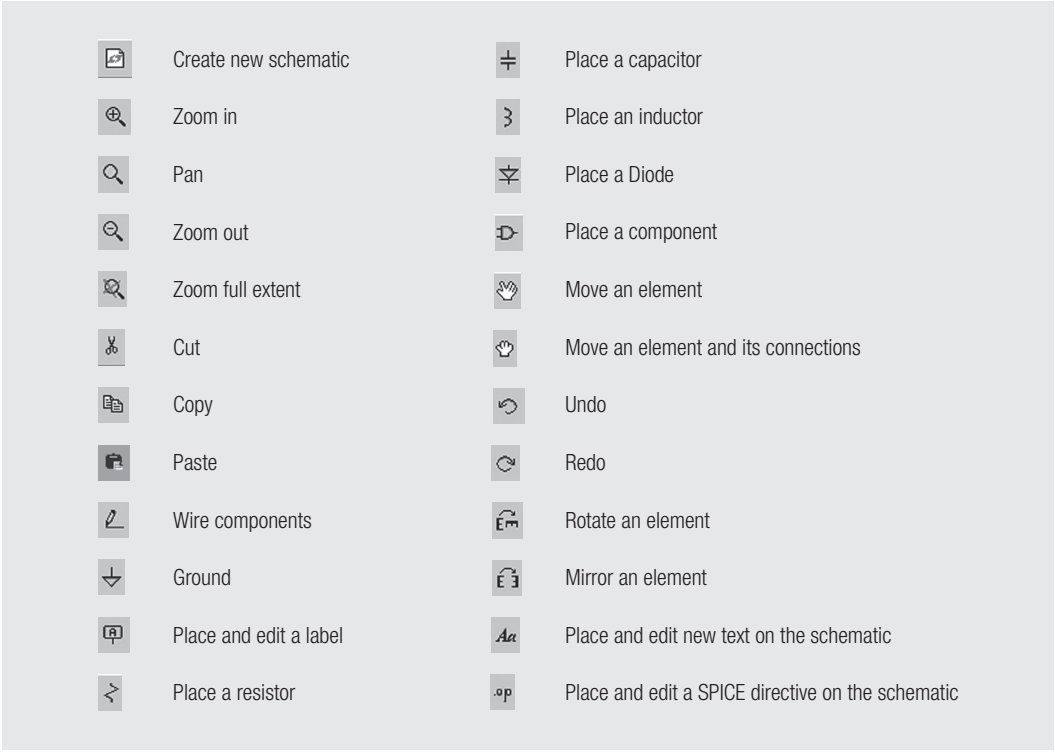

Figure 3.5

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).

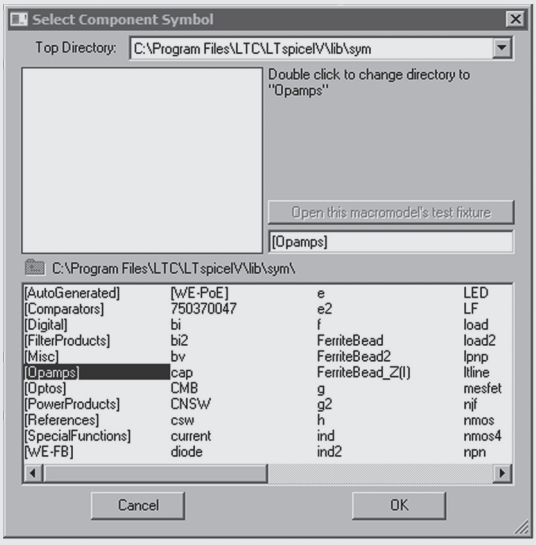

Figure 3.6

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.

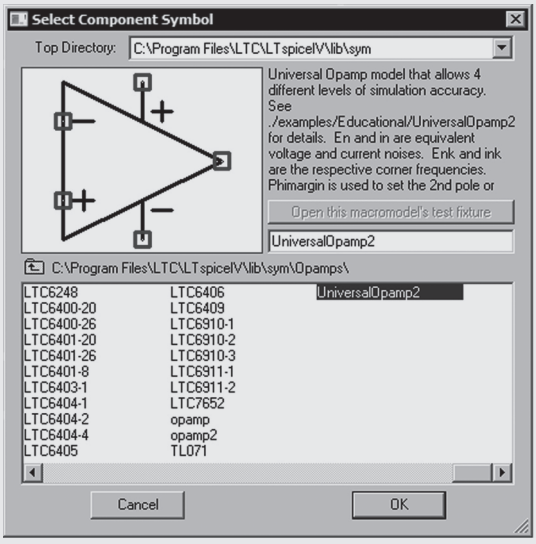

Figure 3.7

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.

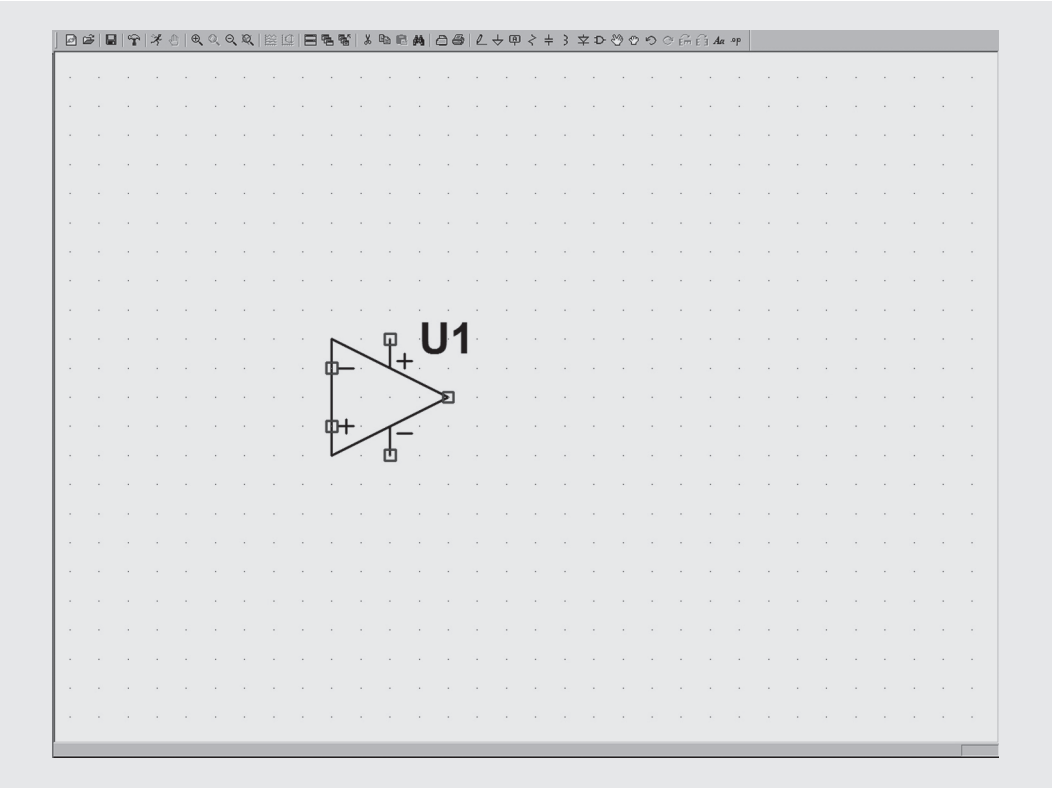

Figure 3.8

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.

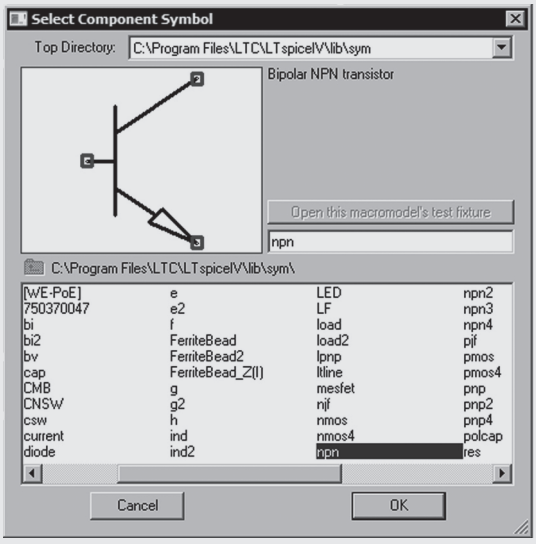

Figure 3.9

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.