Modes of Operation

LTspice IV has two basic modes of driving the simulator:

1. Use the program as a general-purpose schematic capture program with an integrated simulator. Menu commands File=>New, and File=>Open(file type .asc)

2. Feed the simulator with a handcrafted netlist or a foreign netlist generated with a different schematic capture tool. Menu command File=>Open(file type .cir)

LTspice IV is intended to be used as a general purpose schematic capture program with an integrated SPICE simulator. The idea is you draw a circuit(or start with an example circuit that's already drafted) and observe its operation in the simulator. The design process involves iterating the circuit until the desired circuit behavior is achieved in simulation. Earlier versions of LTspice included a synthesizer that would attempt to divine a SMPS design from a user-supplied specification, but that mode of operation has been obsoleted.

The schematic is ultimately converted to a textual SPICE netlist that is passed to the simulator. While the netlist is usually extracted from a graphical schematic drafted in LTspice, an imported netlist can be run directly without having a schematic. This has several uses: (i) Linear Technology's filter synthesis program, FilterCAD, can synthesize a netlist for LTspice to simulate the time domain or frequency response of a filter. (ii) it simplifies benchmarking LTspice against other SPICE programs (iii) professionals historically experienced with SPICE circuit simulators are familiar with working directly with the textual netlists because schematic capture was not integrated with SPICE simulators in older systems.

Example Circuits

There are several resources of example circuits for LTspice IV. There is a directory typically installed at C:\Program Files\LTC\LTspiceIV\examples\Educational that gives numbers non-commercial examples of SPICE simulations that illustrate different analysis types, methods or program features. In the directory C:\Program Files\LTC\ LTspiceIV \examples\jigs there is an example simulation for every Linear Technology device with a macromodel in LTspice IV. Note that these jig circuits are often only test jigs for the macromodel, not necessarily recommended reference designs. Most importantly, your Linear Technology office can probably give you design support specific for your application needs.
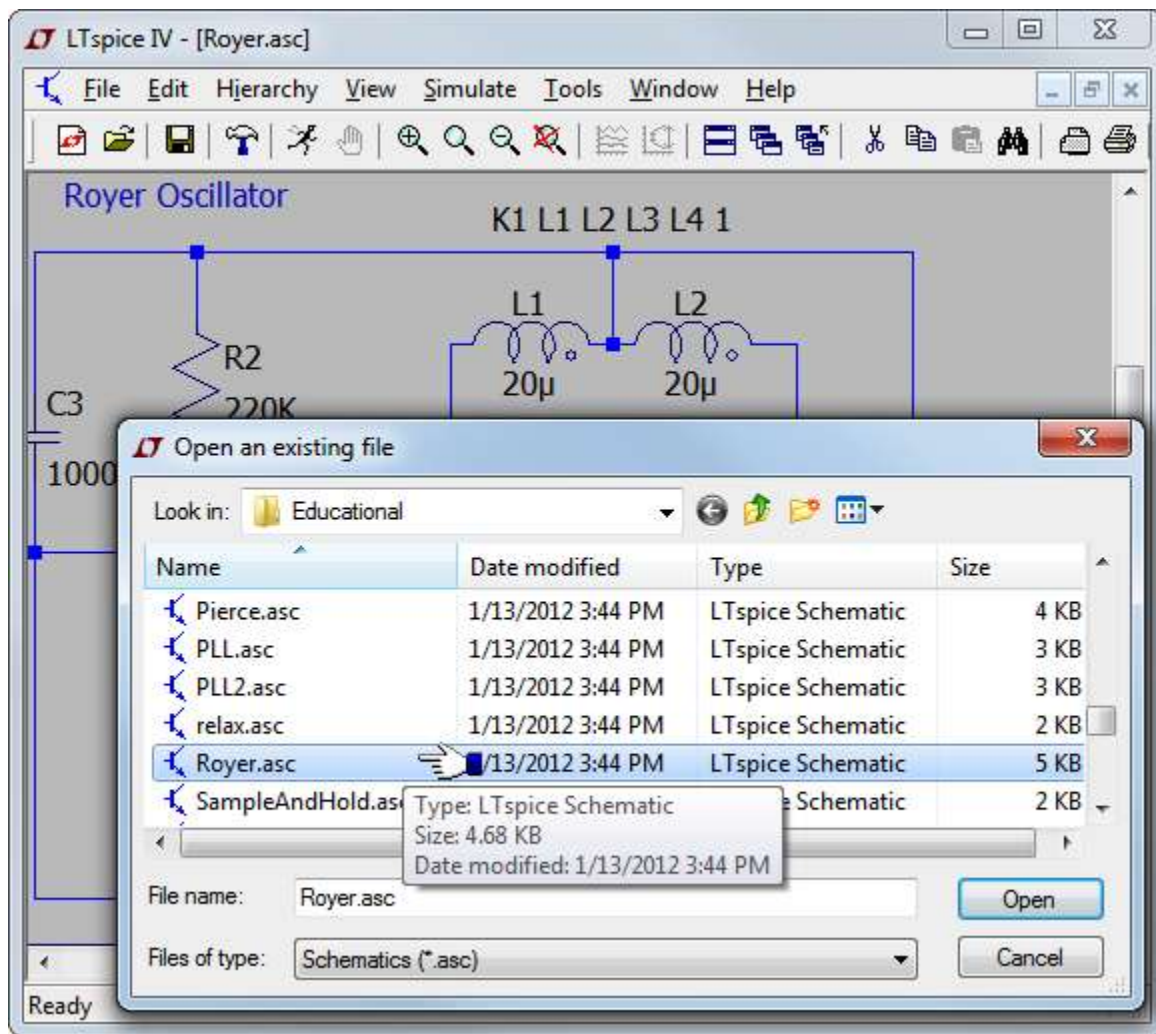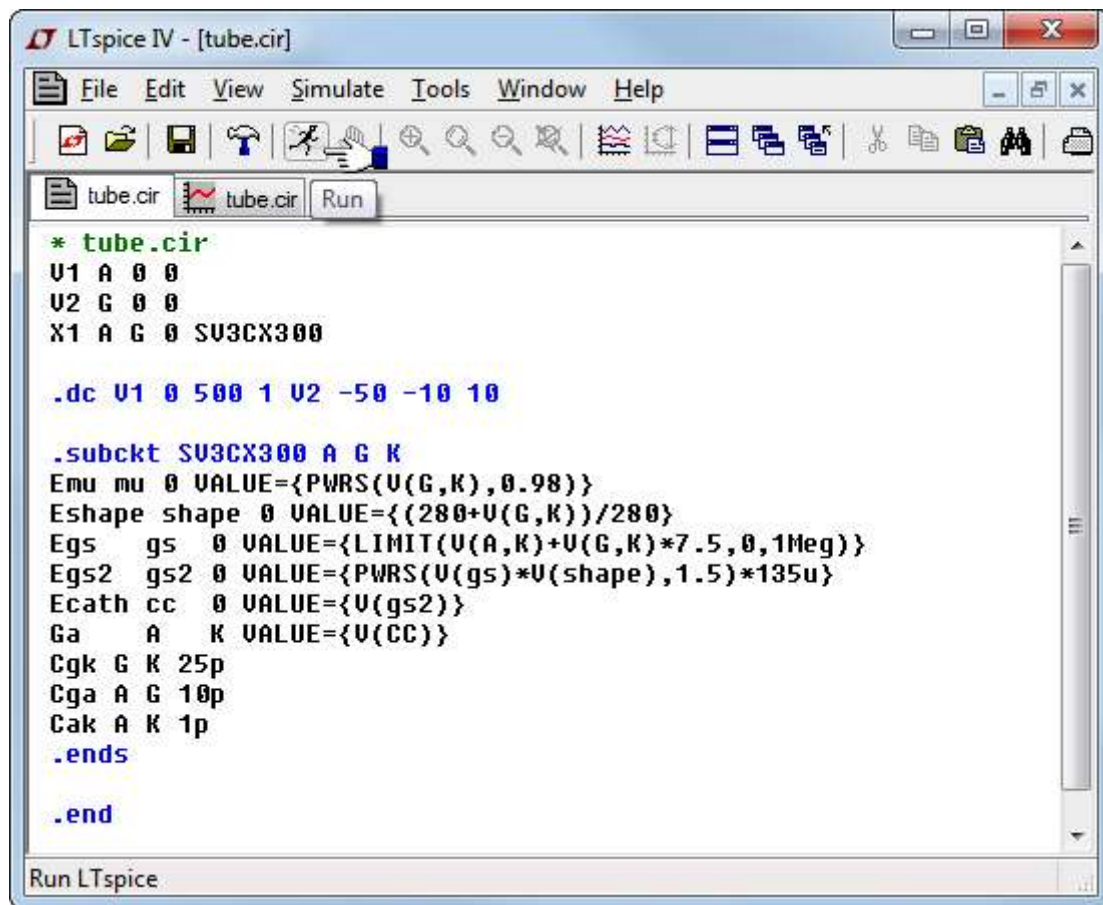
General Purpose Schematic Driven SPICE

You are free to use LTspice IV as a general-purpose schematic
capture/SPICE program. This is useful not only for SMPS design, but
many aspects of analog engineering. The example circuits typically
installed in the directory C:\Program
Files\LTC\LTspiceIV\examples\Educational\ illustrate various LTspice
capabilities.
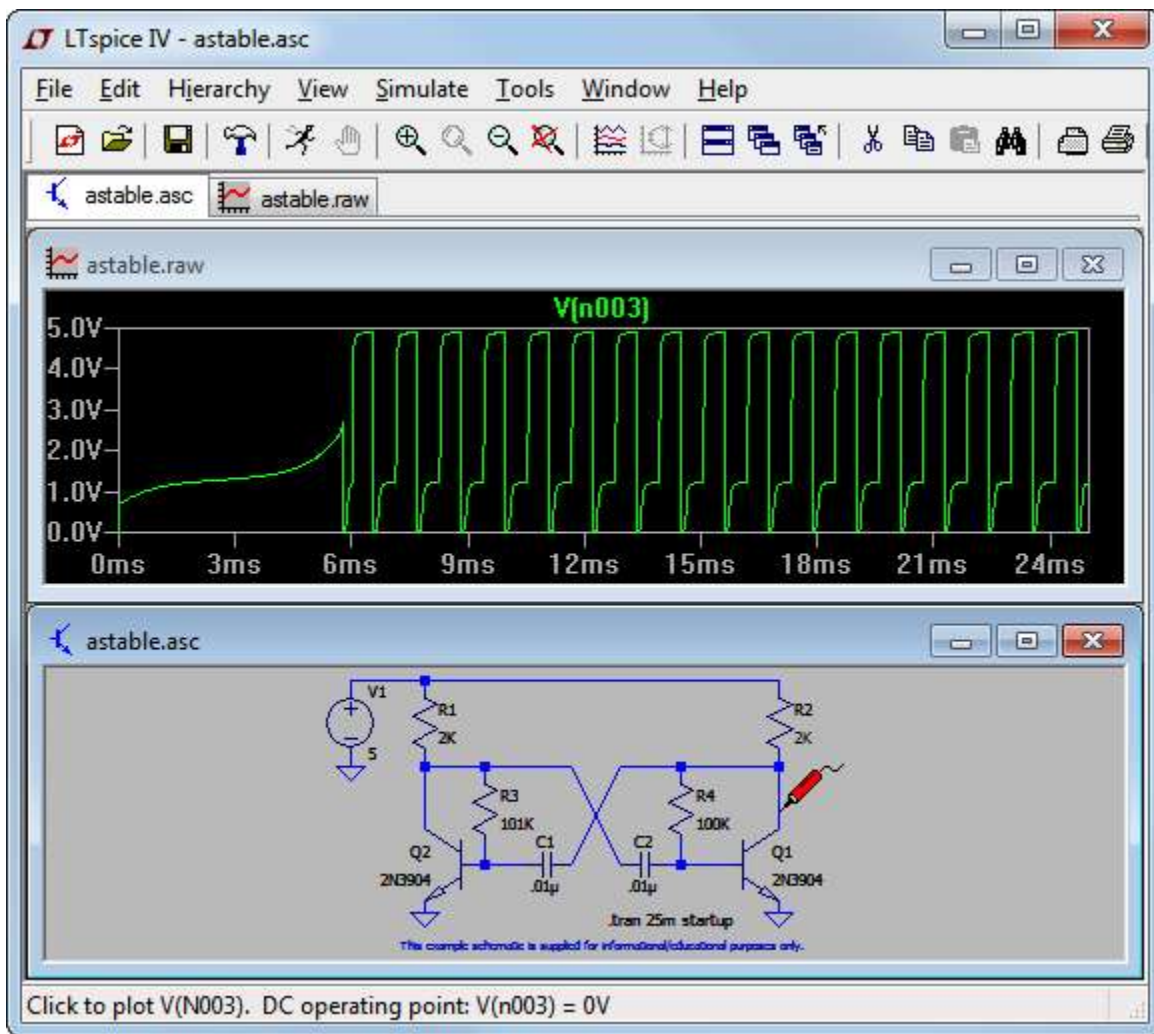
## Externally Generated Netlists

You can open netlists generated either by hand or by other schematic capture programs. These files usually have a filename extension of ".cir", but ".net" and ".sp"; are understood. The ASCII editor used for netlist files supports unlimited file size and unlimited undo/redo. The menu command Tools=>Color Preferences can be used to adjust the colors used in the ASCII editor.

```
LTspice IV - [tube.cir]
File  Edit  View  Simulate  Tools  Window  Help

 tube.cir    tube.cir   Run

* tube.cir
V1 A 0 0
V2 G 0 0
X1 A G 0 SV3CX300

.dc V1 0 500 1 V2 -50 -10 10

.subckt SV3CX300 A G K
Emu mu 0 VALUE={PWRS(V(G,K),0.98)}
Eshape shape 0 VALUE={(280+V(G,K))/280}
Egs    gs  0 VALUE={LIMIT(V(A,K)+V(G,K)*7.5,0,1Meg)}
Egs2  gs2 0 VALUE={PWRS(V(gs)*V(shape),1.5)*135u}
Ecath cc  0 VALUE={V(gs2)}
Ga     A   K VALUE={V(CC)}
Cgk G K 25p
Cga A G 10p
Cak A K 1p
.ends

.end

Run LTspice
```

Efficiency Report

It is possible to obtain an efficiency report from a DC-DC converter
from a time domain .tran analysis that contains the keyword "steady".
After a steady state simulation, an efficiency report can be made
visible on the schematic as a block of comment text:
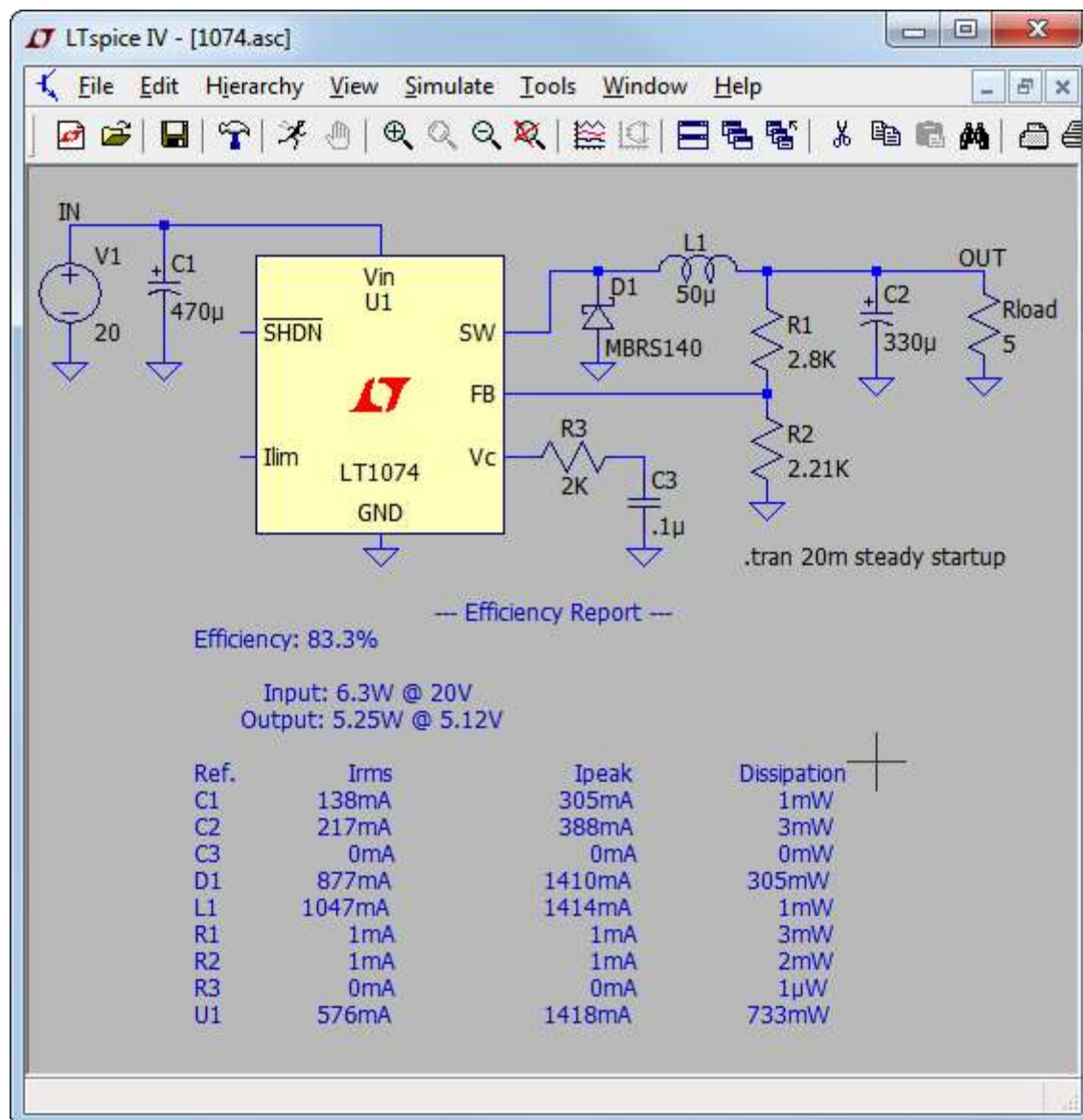
The efficiency of the DC-DC converter is derived in the following manner. In order to identify the input and output, there must be exactly one voltage source and one current source. The voltage source is assumed to be the input while the current source is assumed to be the output. The circuit is run until steady state is sensed by the simulator. This requires the SMPS macromodels to be written with information on how to detect steady state. Usually this is detected by noting when the error amp current, averaged over a clock cycle, diminishes to a small value for several cycles. Then at a clock edge, the energy stored in each reactance is noted and the simulation is run for another ten clock cycles but now integrating the dissipation in every device. At the clock edge of the last cycle, the energy stored in every reactance is noted again and the simulation is stopped. The efficiency is reported as the ratio of output power delivered to the load by the input power sourced by the input voltage after making an adjustment for the change in energy stored in the reactances. Since the dissipation of each device was also noted, it is possible to look

how close the energy checksum is to zero.

You can usually compute efficiency of SMPS circuits you draft yourself by using checking the "Stop simulating if steady state is detected" on the Edit Simulation Command editor. After the simulation, use the menu command View=>Efficiency Report.

Automatic detection of steady state doesn't always work. Sometimes the criteria for steady state detection is too strict and sometimes too lenient. You then either adjust the option parameter sstol or simply interactively set the limits for the efficiency integration.

Command Line Switches

The following table summarizes the command line switches understood by the LTspice executable(scad3.exe):

Flag	Description
-ascii	Use ASCII .raw files. Seriously degrades program performance.
-b	Run in batch mode. E.g. "scad3.exe -b deck.cir" will leave the data in file deck.raw
-big	Start as a maximized window.
-encrypt	Encrypt a model library. For 3rd parties wishing to allow people to use libraries without revealing implementation details. Not used by Linear Technology Corporation models.
-FastAccess	Batch conversion of a binary .raw file to Fast Access format.
-ini <path>	Specify an .ini file to use other than % WINDIR%\scad3.ini
-max	Synonym for -big
-netlist	Batch conversion of a schematic to a netlist.
-nowine	Prevent use of WINE(Linux) workarounds.
-PCBnetlist	Batch conversion of a schematic to a PCB format netlist.
-registry	Force LTspice to store user preferences, MRU, etc. in the registry instead of the %

	`WINDIR%\scad3.ini file.`
-Run	Start simulating the schematic opened on the command line without pressing the Run button.
-SOI	Allow MOSFET's to have up to 7 nodes even in subcircuit expansion.
-uninstall	Executes one step of the uninstallation process.
-wine	Force use of WINE(Linux) workarounds.