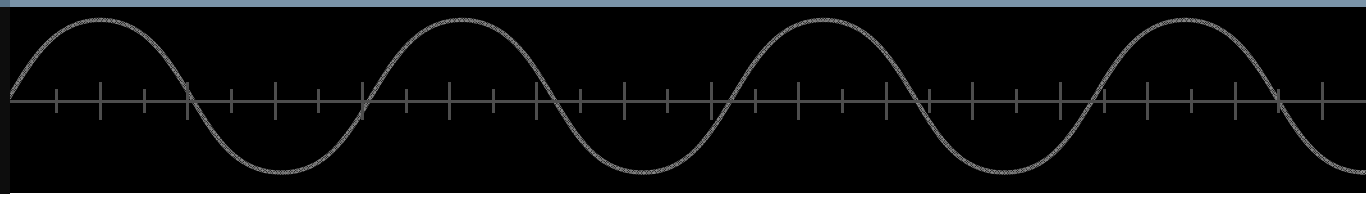
SwitcherCAD III/LTspice Getting Started Guide



LINEAR TECHNOLOGY

Benefits of Using SwitcherCAD III/LTspice

- Stable SPICE circuit simulation with
  - Unlimited number of nodes
  - Schematic/symbol editor
  - Waveform viewer
  - Library of passive devices
- Fast simulation of switching mode power supplies (SMPS)
  - Steady state detection
  - Turn on transient
  - Step response
  - Efficiency / power computations
- Advanced analysis and simulation options
  - Not covered in this presentation

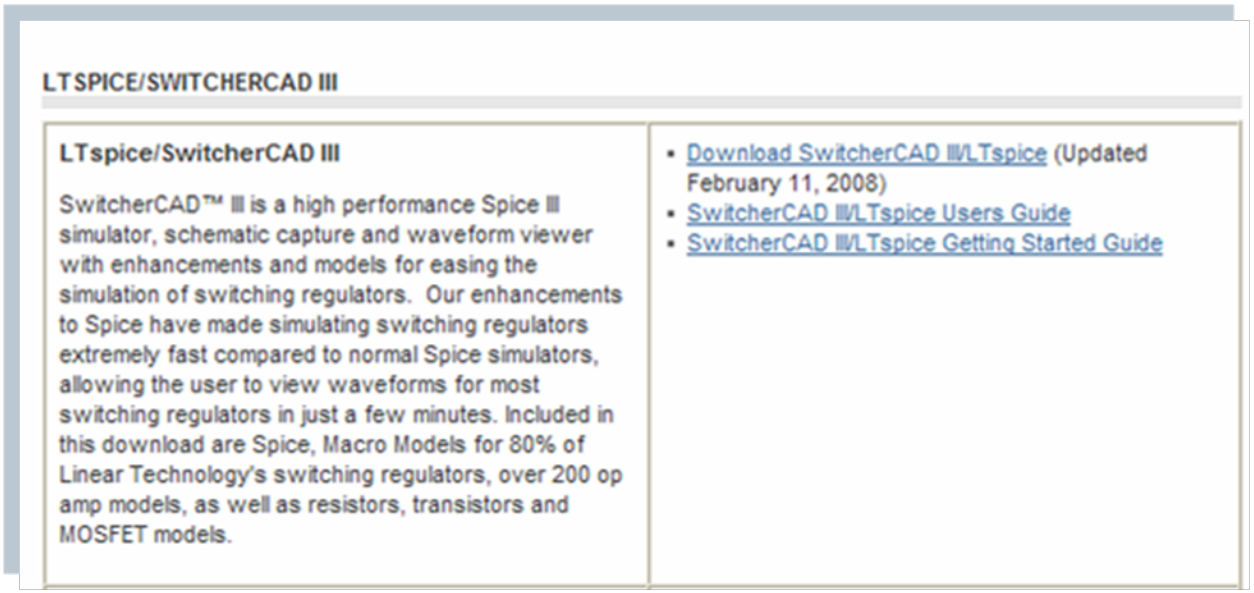
Outperforms pay-for options

LTspice is also a great schematic capture

- Over 1100 macromodels of Linear Technology products
- 500+ SMPS

How Do You Get SwitcherCAD III/LTspice

- Go to http://www.linear.com/software
- Left click on download SwitcherCAD III/LTspice
- Register for a new MyLinear account to receive updates if you have not done so already



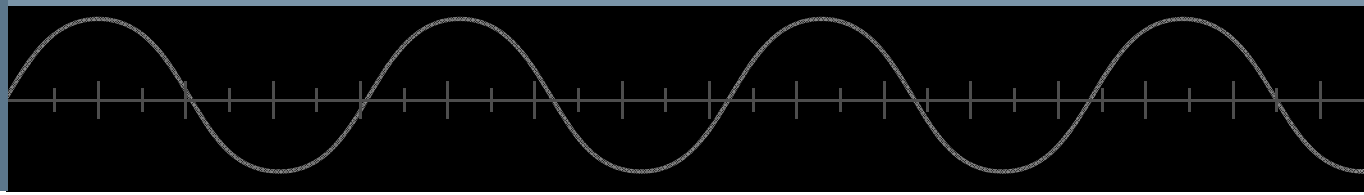
LTSPICE/SWITCHERCAD III

LTspice/SwitcherCAD III

SwitcherCAD™ III is a high performance Spice III simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulators. Our enhancements to Spice have made simulating switching regulators extremely fast compared to normal Spice simulators, allowing the user to view waveforms for most switching regulators in just a few minutes. Included in this download are Spice, Macro Models for 80% of Linear Technology's switching regulators, over 200 op amp models, as well as resistors, transistors and MOSFET models.

- Download SwitcherCAD III/LTspice (Updated February 11, 2008)
- SwitcherCAD III/LTspice Users Guide
- SwitcherCAD III/LTspice Getting Started Guide

Getting Started



Getting Started using SwitcherCAD III/LTspice

- Use one of the 100s of demo circuits available on linear.com
  - Reviewed by Linear Technology's Factory Applications Group
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point
- Use the schematic editor to create your own design
  - LTspice contains macromodels for most LTC power devices

Demo Circuits on linear.com

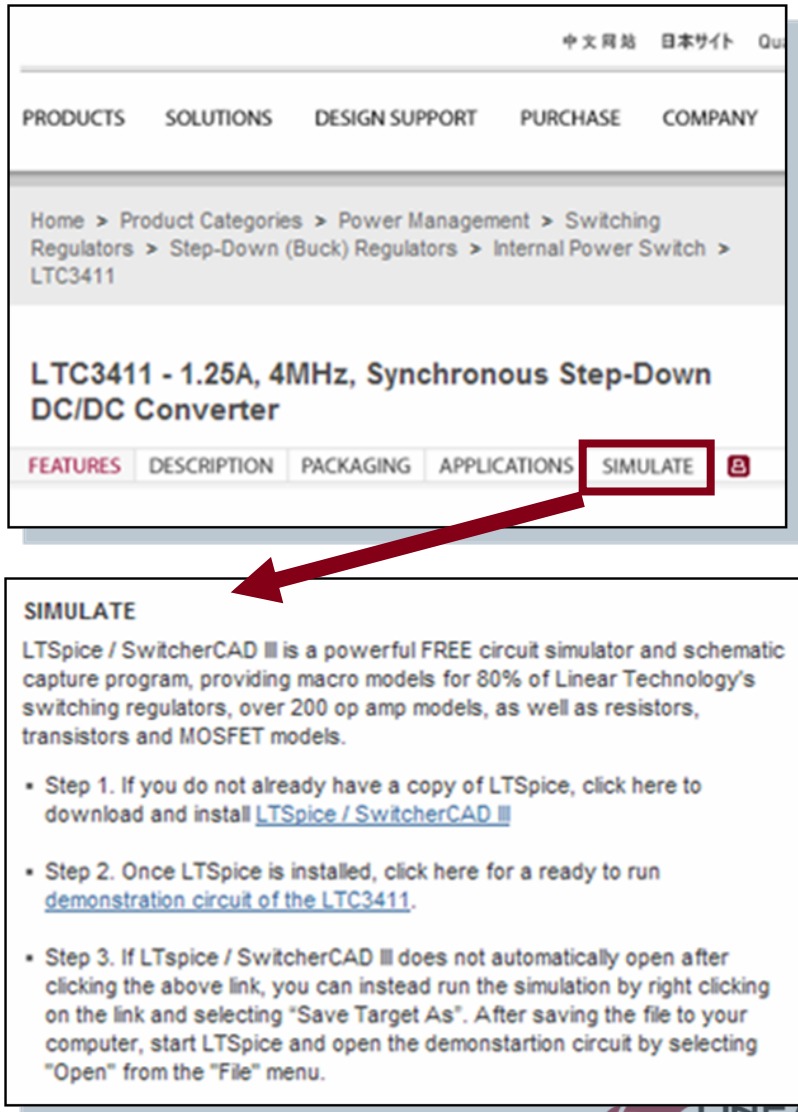
- Go to http://www.linear.com
- Enter root part number in the search box (e.g. 3411)
- Select Simulate Tab
- Follow the instructions provided

If you do not find a demo circuit of interest, use a pre-drafted test fixture – covered next

Download LTspice

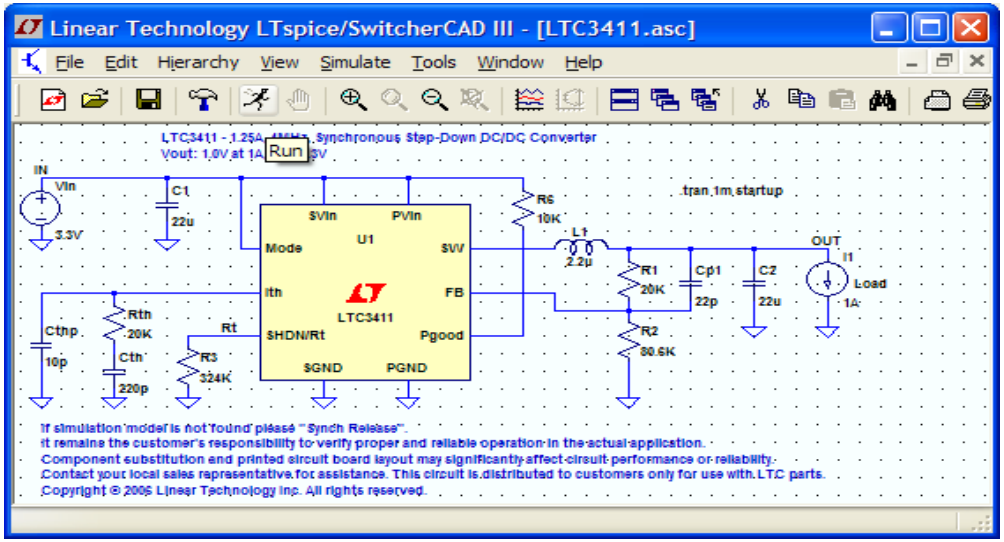
Download Demo Circuit

Complete list of demo circuits available at www.linear.com/democircuits



Demo Circuit

- ✓ Designed and Reviewed by Factory Apps Group



To run a demo circuit jump to the ***Run and Probe a Circuit in LTspice*** section

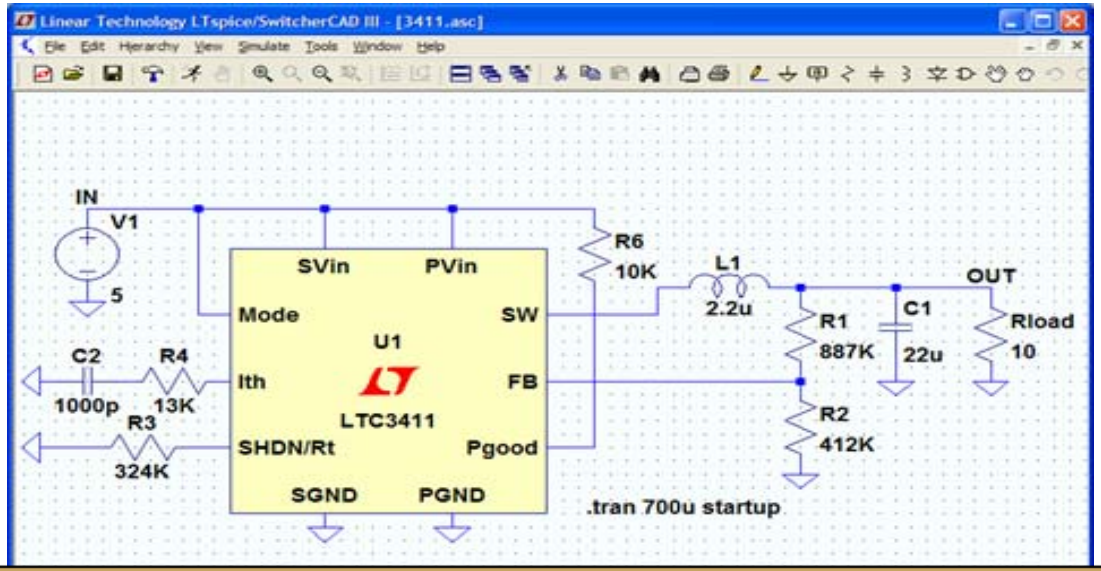
- It remains the customer's responsibility to verify proper and reliable operation in the actual application
- Printed circuit board layout may significantly affect circuit performance or reliability

Getting Started using SwitcherCAD III/LTspice

- Use one of the 100s of demo circuits available on linear.com
  - Reviewed by Linear Technology's Factory Applications Group
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point
- Use the schematic editor to create your own design
  - LTspice contains macromodels for most LTC power devices

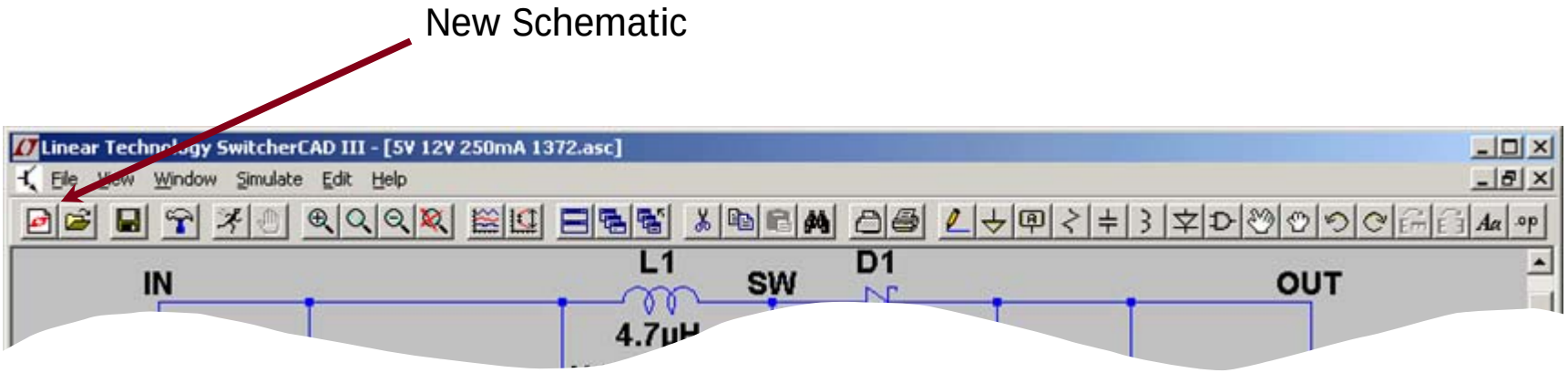
Pre-Drafted Test Fixture (JIG)

- Used for testing models during development
- Provides a draft starting point
  - Not reviewed by Linear Technology's factory applications team



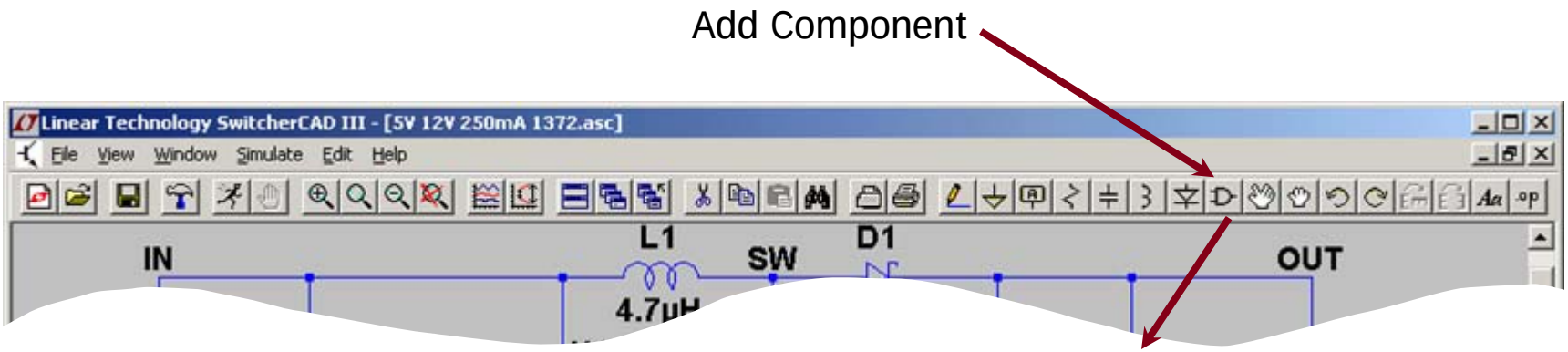
- It remains the customer's responsibility to verify proper and reliable operation in the actual application
- Printed circuit board layout may significantly affect circuit performance or reliability

Start with a New Schematic

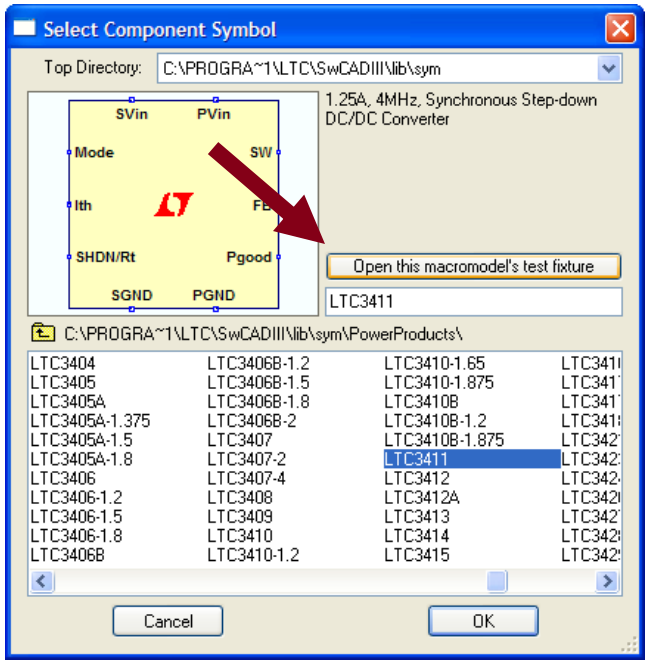


- Left click on the **New Schematic** symbol in the Schematic Editor Toolbar

Add a Macromodel & Opening Test Fixture



- Left click on the **Component** symbol in the Schematic Editor Toolbar
- Enter “root” part to search for the model (e.g. 3411)
- Left click on **Open this macromodel’s test fixture**

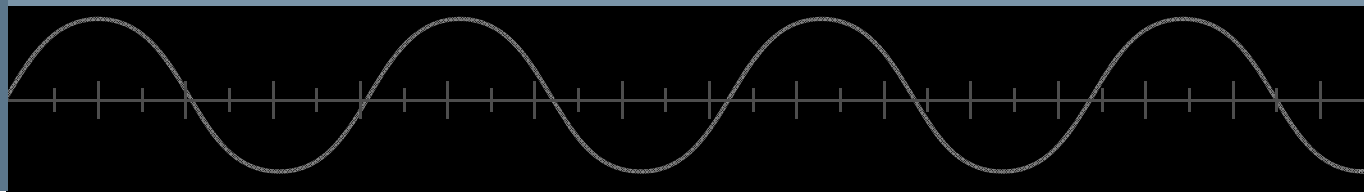


To run a test fixture, jump to the ***Run and Probe a Circuit in LTspice*** section

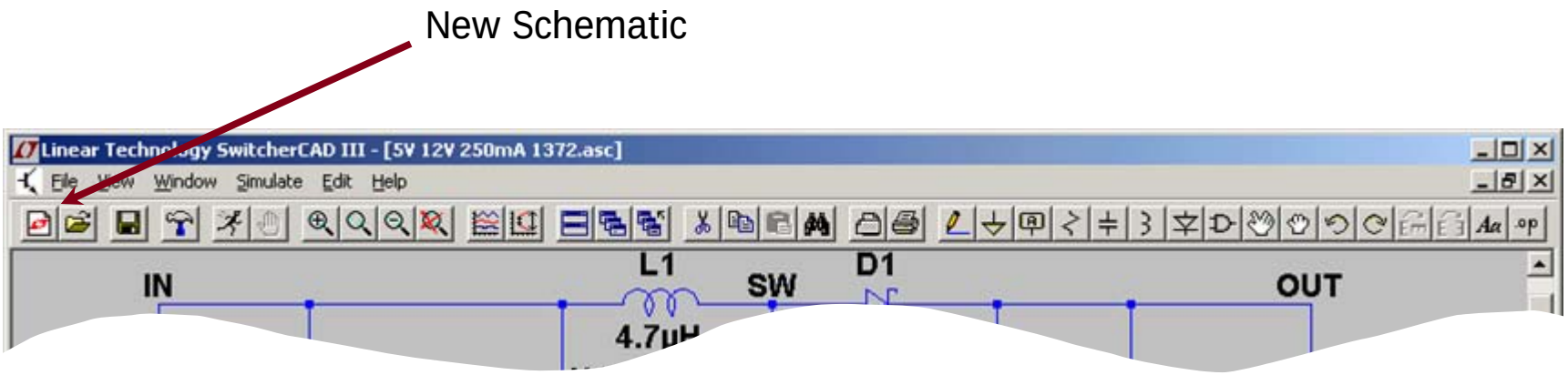
Getting Started using SwitcherCAD III/LTspice

- Use one of the 100s of demo circuits available on linear.com
  - Reviewed by Linear Technology's Factory Applications Group
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point
- Use the schematic editor to create your own design
  - LTspice contains macromodels for most LTC power devices

Draft a Design Using the Schematic Editor



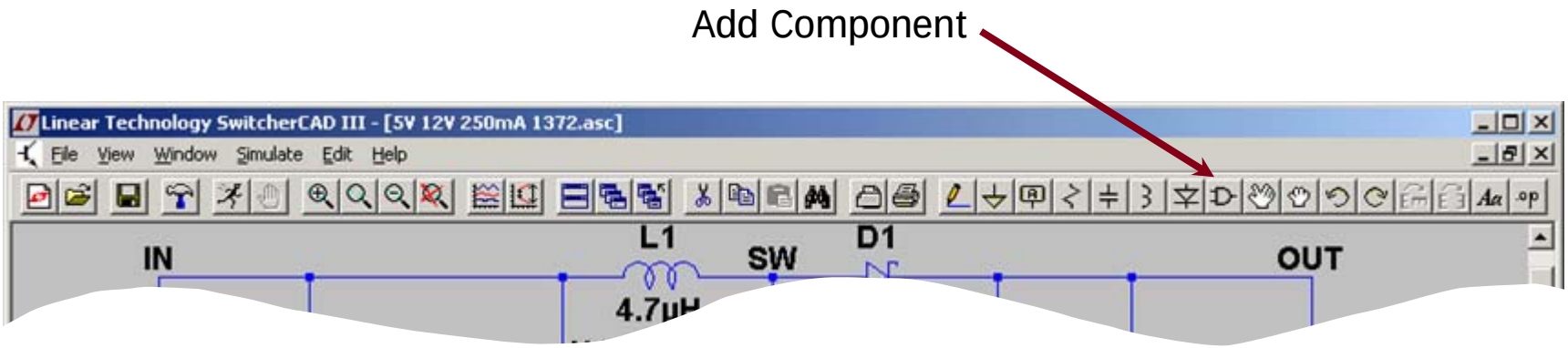
Start with a New Schematic



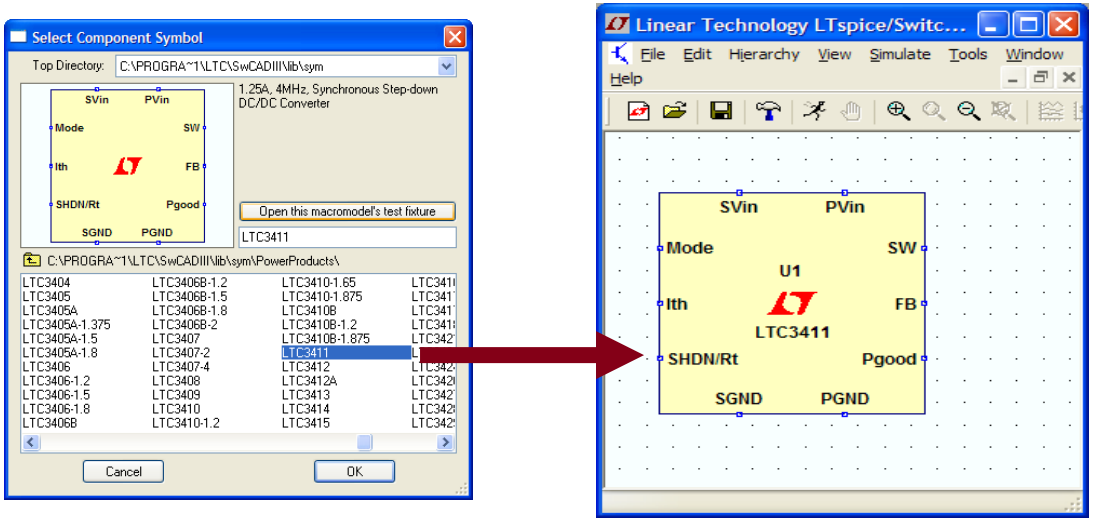
- Left click on the **New Schematic** symbol in the Schematic Editor Toolbar

LTspice is also a great schematic capture

Add a Linear Technology Macromodel



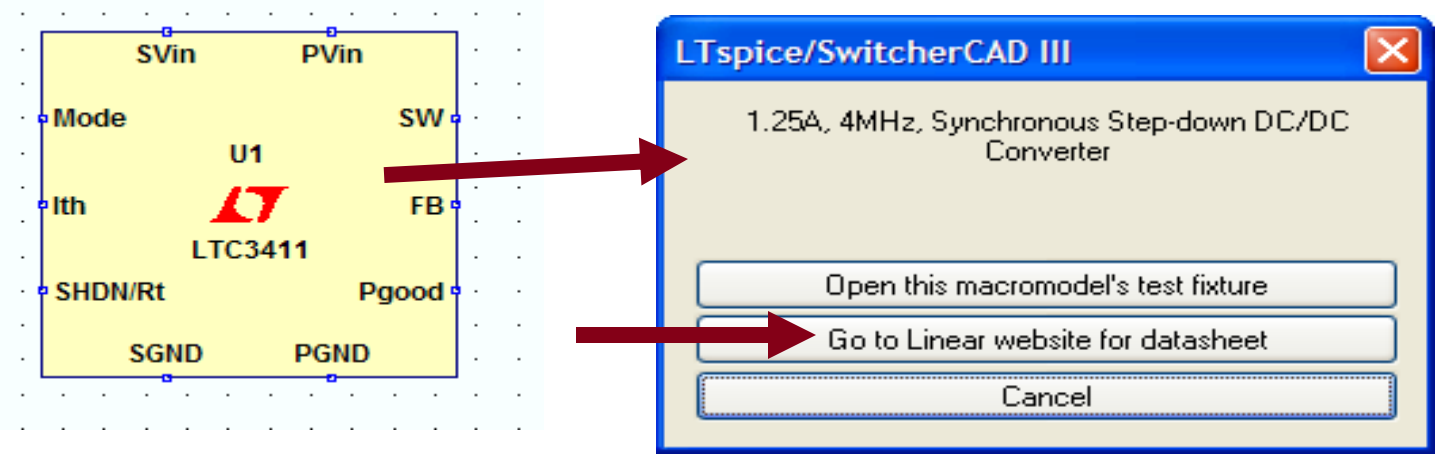
- Left click on the **Component** symbol in the Schematic Editor Toolbar
- Enter “root” part to search for the model (e.g. 3411)
- Left click on **OK**



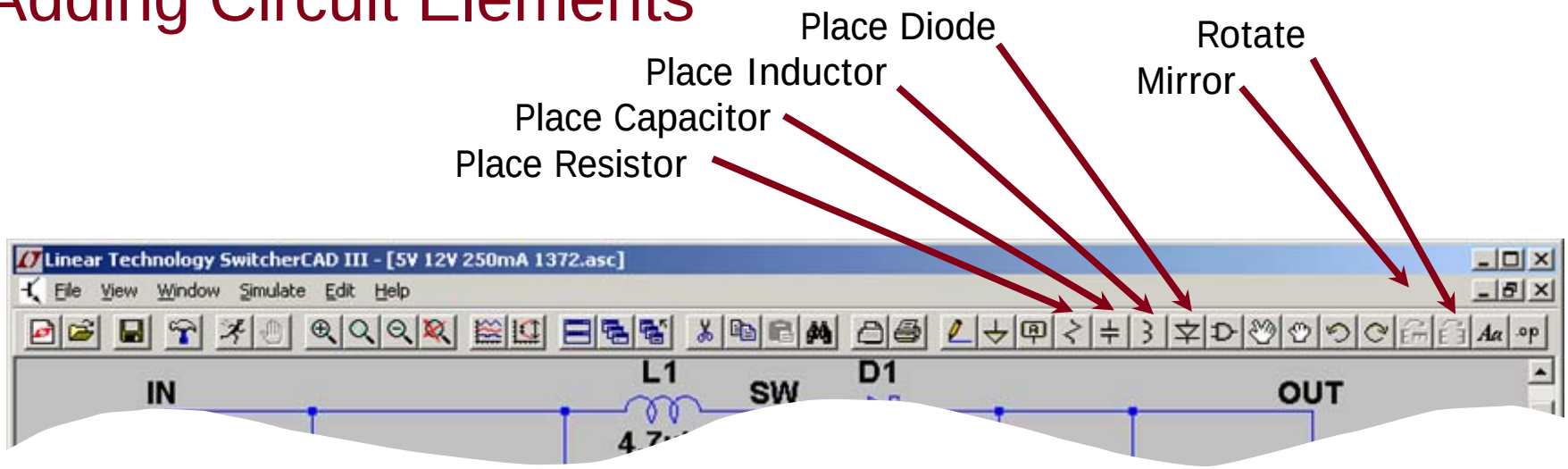
Getting the Latest Datasheet

- Use the macromodel's shortcuts to download the *Datasheet* as a reference for your design
  - Hold Ctrl key and right click (*Ctrl – right click*) over the LT macromodel's symbol
  - Left click on Go to Linear website for datasheet on the dialog box that appears

You can also open the macromodel's test fixture as a draft starting point



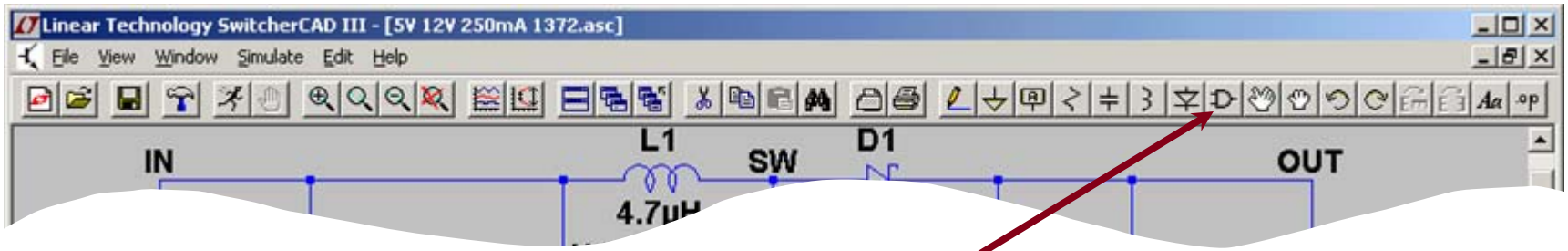
Adding Circuit Elements



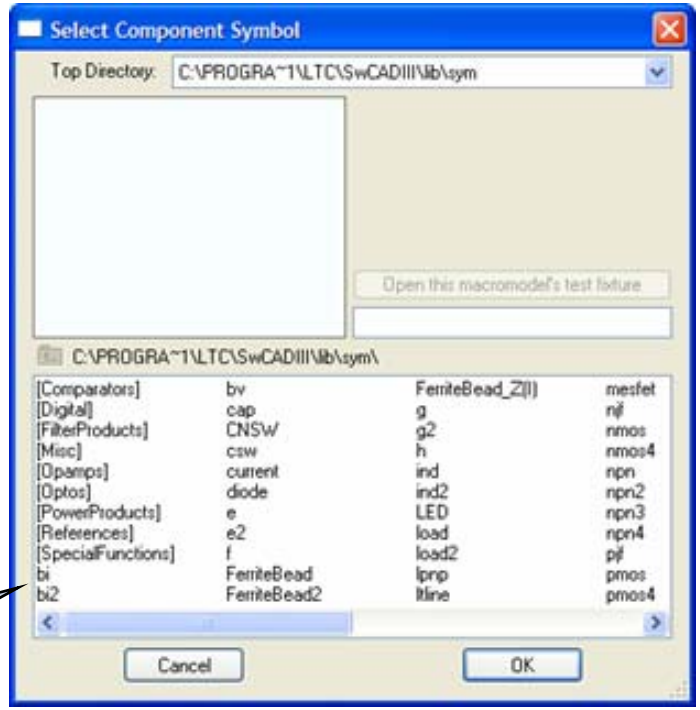
- Left click on the desired *component* in the Schematic Editor Toolbar
- Left click on **Rotate** or **Mirror** to adjust orientation
  - Alternate you can also use Ctrl – R and Ctrl – M key shortcuts
- Move the mouse to the position you want to place it
- Left click to place it

To cancel or quit a component type, click the right mouse button

Adding Sources, Loads & Additional Circuit Elements



- Left click on the **Component** symbol in the Schematic Editor Toolbar
- Search directory structure for desired circuit element (e.g. load and voltage)
- Left click on **OK**
- Move the mouse to the position you want to place it
- Left click to place it

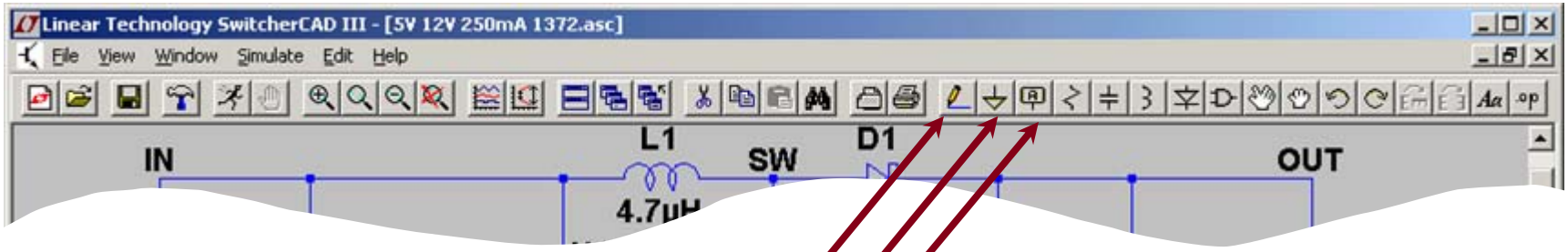


Additional Circuit Elements Like Sources and Loads

Highlights of Additional Circuit Elements

- Left click on the **Component** symbol in the Schematic Editor Toolbar for a directory of additional circuit elements:
  - Arbitrary behavioral source
  - Voltage dependent voltage
  - Current dependent current
  - Voltage dependent current
  - Current dependent voltage
  - Independent current source
  - JFET transistor
  - Mutual inductance
  - MOSFET transistor
  - Lossy transmission line
  - Bipolar transistor
  - Voltage controlled switch
  - Lossless transmission line
  - Uniform RC-line
  - Independent voltage source
  - Current controlled switch
  - Subcircuit
  - MESFET transistor
  - …many more

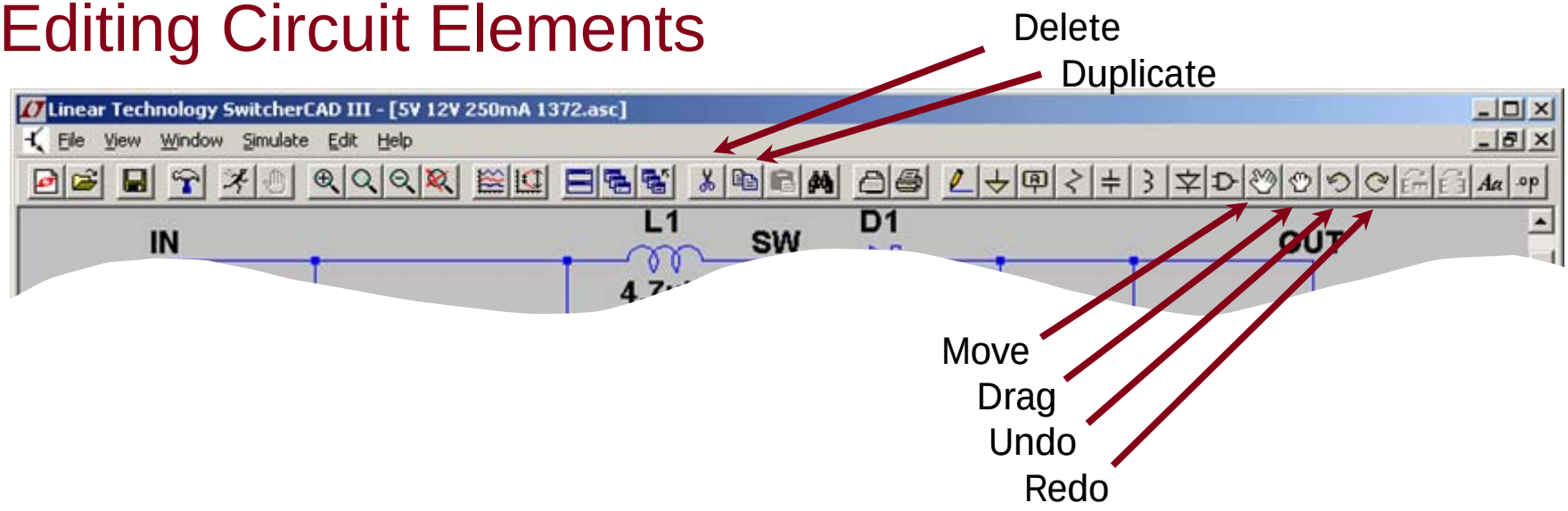
Drawing Lines and Labeling Nodes



Do not forget to place a ground in your design, it is required for simulation!

- Lines
  - Left click on the Draw Wire in the Schematic Editor Toolbar
  - Left click a blue box (terminal)
  - Define the line's path with a left click over intermediate points
  - Left click on another blue box (terminal)

Editing Circuit Elements

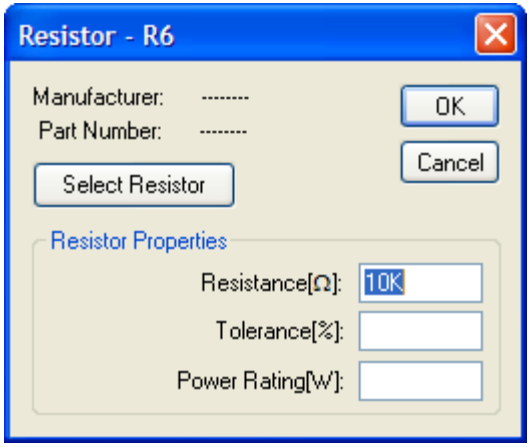


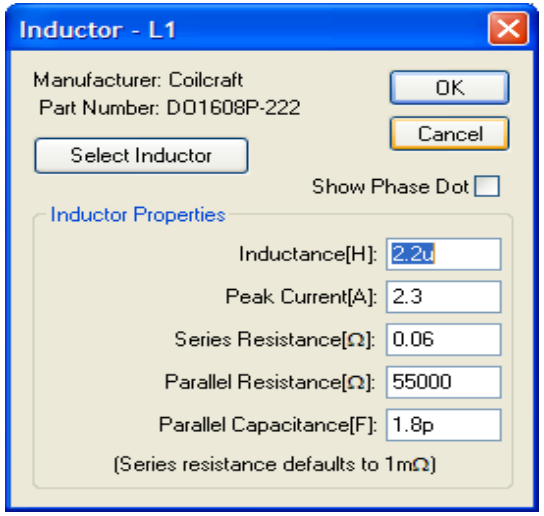
- Left click on the desired editing option
- Left click on the circuit element

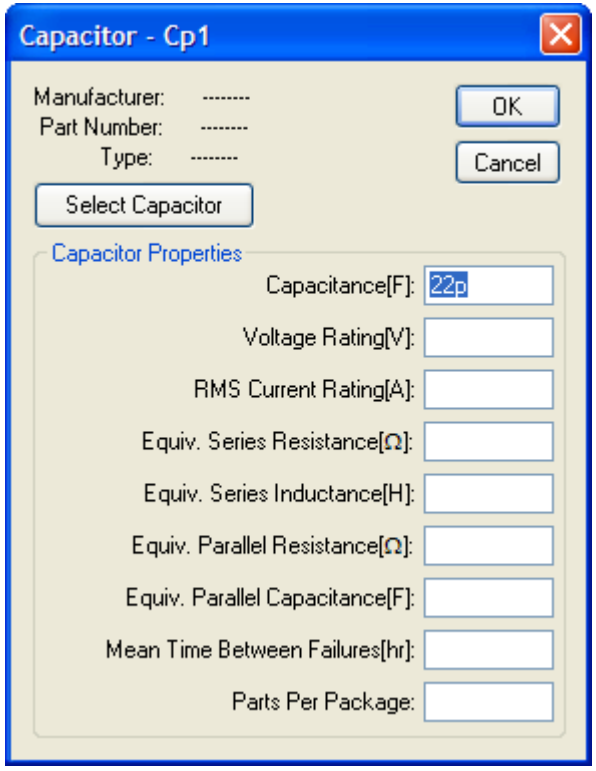
To organize your layout, use the ***Drag*** option to move circuit elements around and to adjust lines between terminals

Editing Circuit Elements Attributes

- Right click on the component ***symbol*** to modify attributes







- Right click on the text next to the component to edit the visible attribute and label
  - Pointer will turn into a text caret

Use Labels to Specify Units in Circuit Elements Attributes

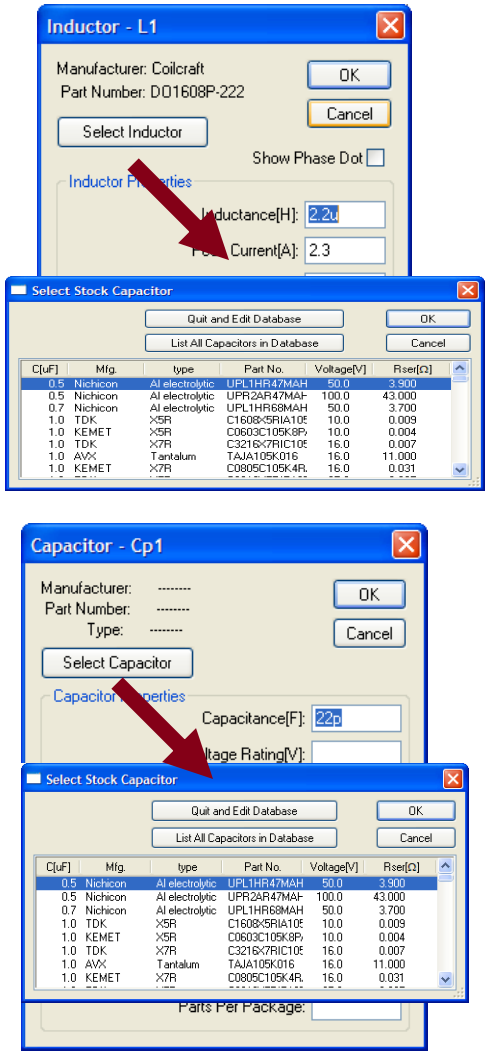
- K = k = kilo = 10^3
- MEG = meg = 10^6
- G = g = giga = 10^9
- T = t = terra = 10^{12}
- M = m = milli = 10-3
- U = u = micro = 10-6
- N = n = nano = 10-9
- P = p = pico = 10-12
- F = f = femto = 10-15

Hints

- Use ***MEG*** to specify 10^6, not *M*
- Enter ***1*** for 1 Farad, not *1F*

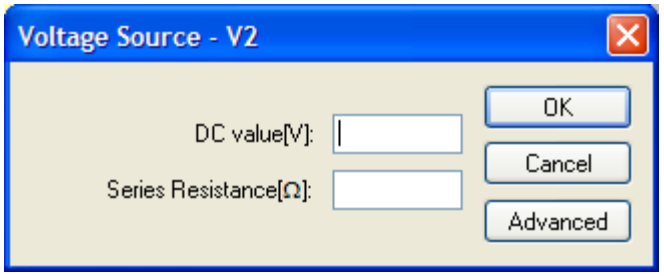
Circuit Elements Database

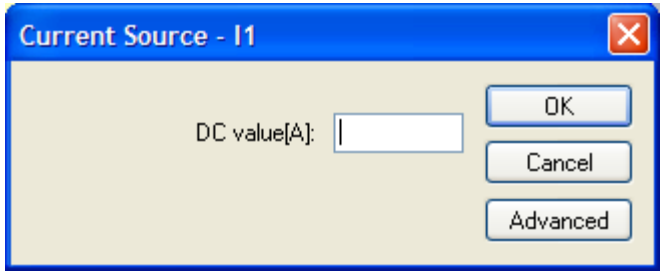
- Some components have an available database of manufacturers' attributes
  - Resistors, capacitors, inductors, diodes,
  - Bipolar transistors, MOSFET transistors, JFET transistors
  - Independent voltage and current sources
- To configure a component to a manufacture's attributes
  - Right click on the component *symbol*
  - Left click on **Select…** or **Pick New…**
  - Left click on a selected device
  - Left click on OK



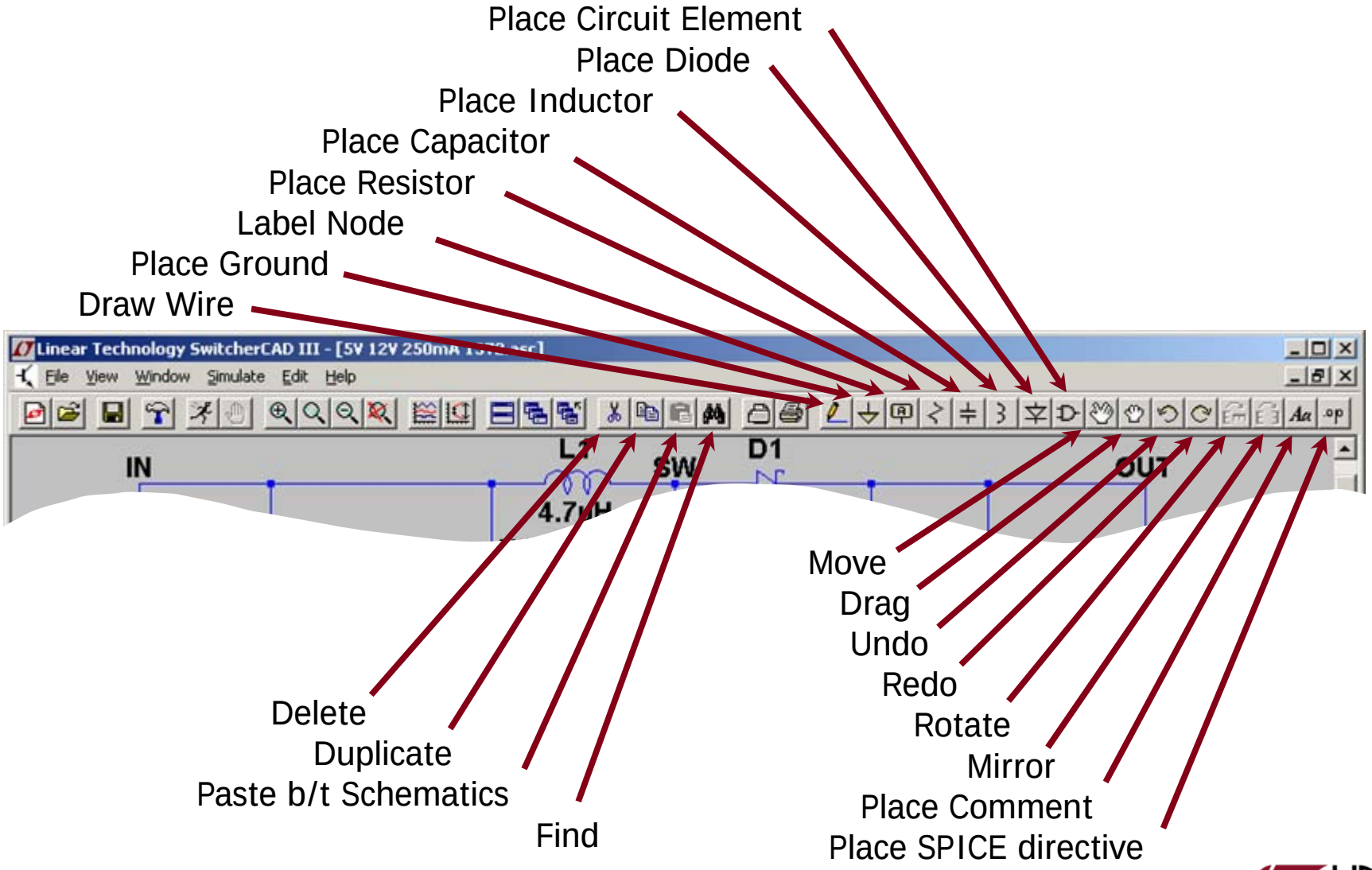
Editing Voltage Sources and Loads

- Voltage Source
  - Right click the voltage ***symbol***
  - Enter ***DC voltage value*** and (optional) Series Resistance
  - Left click on OK
- Load (current)
  - Right click on the load ***symbol***
  - Enter ***DC current value***
  - Left click on OK

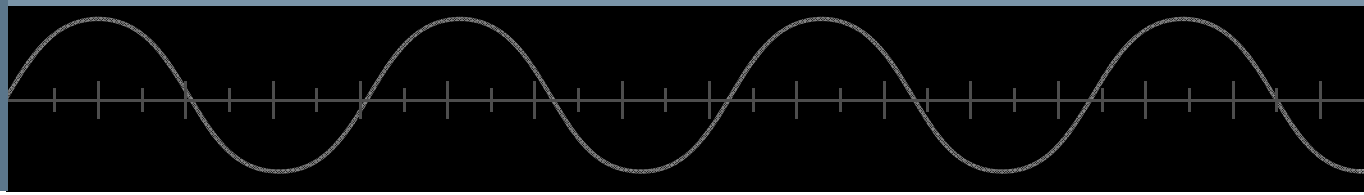




Summary of Schematic Editor Toolbar



Run and Probe a Circuit



Simulation Commands

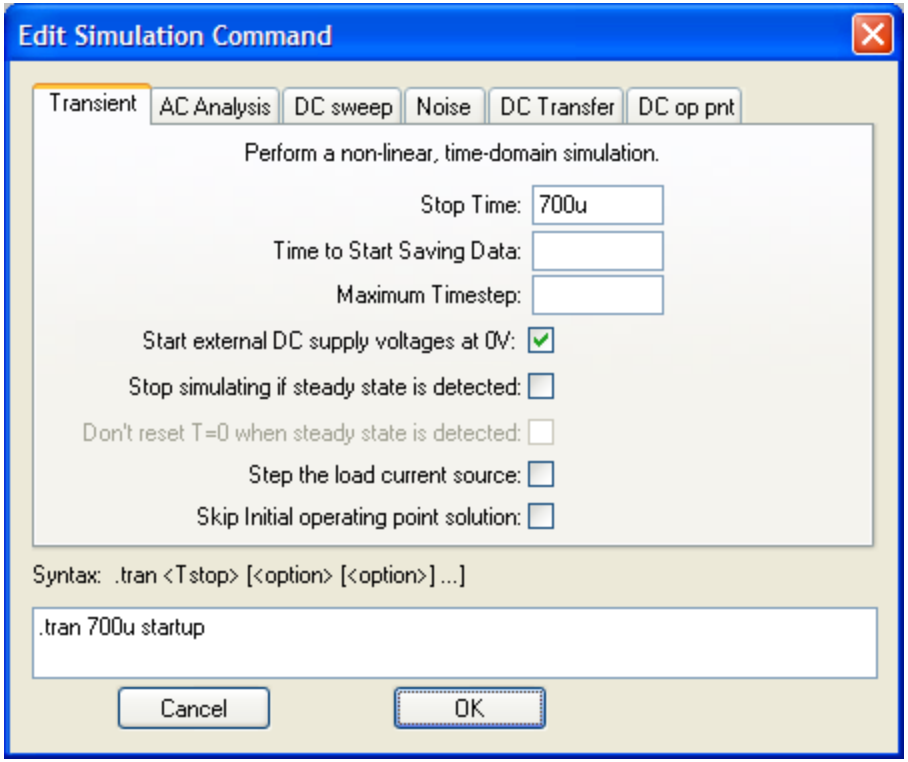
- To run a simulation, specify the type of analysis to be performed
- There are six different types of analyses:
  - Transient analysis
  - Small signal AC
  - DC sweep
  - Noise
  - DC transfer function
  - DC operating point
- Simulation commands are placed on the schematic as text
  - Called dot commands

More information on simulation and dot commands are available in SwitcherCAD III/LTspice User Guide

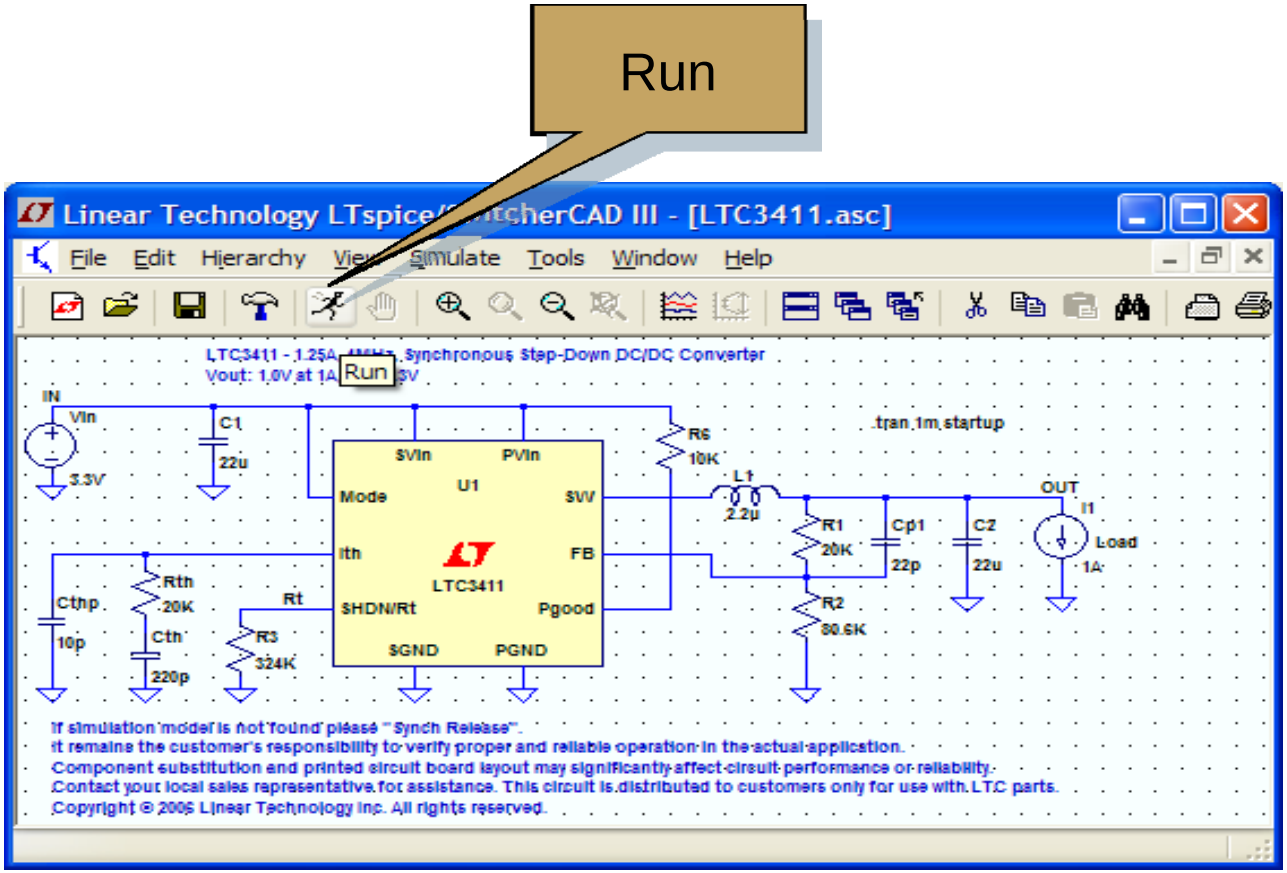
Editing Simulation Commands

- Left click on **Simulation** menu
- Left click on **Edit Simulation Cmd**
- As a starting point in a simulation
  - Left click on **Transient** tab
  - Enter a ***Stop Time***
    - You may need to adjust this again later
- Select **OK**

Demo Circuits and **Test Fixtures** have predefined Simulations Commands



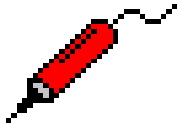
Running a Circuit



If model is not found please Sync Release under Help menu to update LTspice

Probing a Circuit & Waveform Viewer

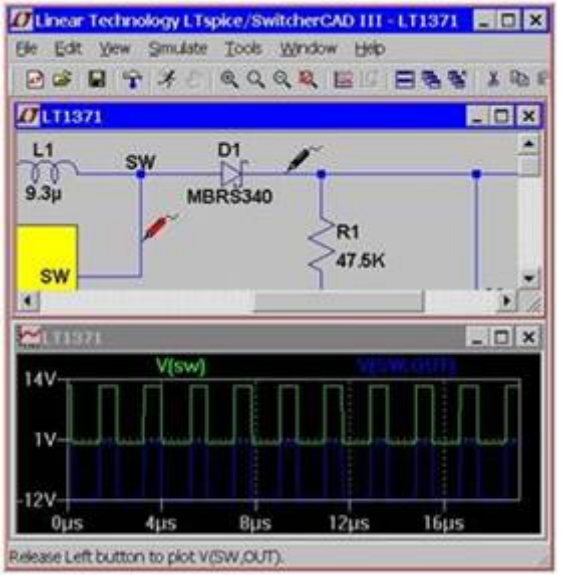
- Left click on any wire to plot the voltage on the waveform viewer

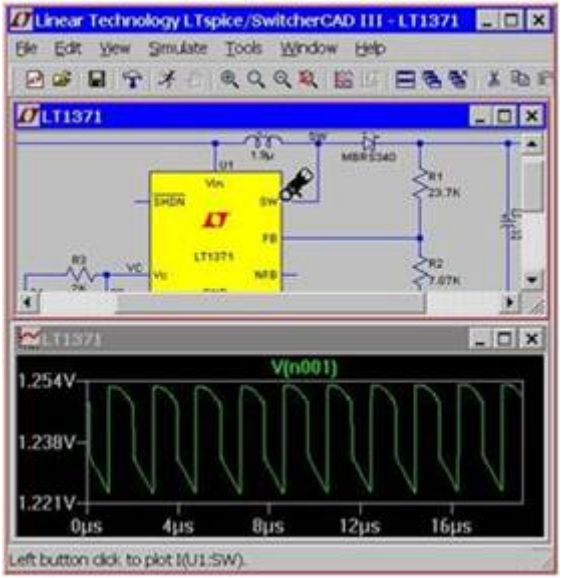


Voltage probe cursor

- Left click on the body of the component to plot the current on the waveform viewer
  - Convention of positive current is in the direction into the pin

Current probe cursor

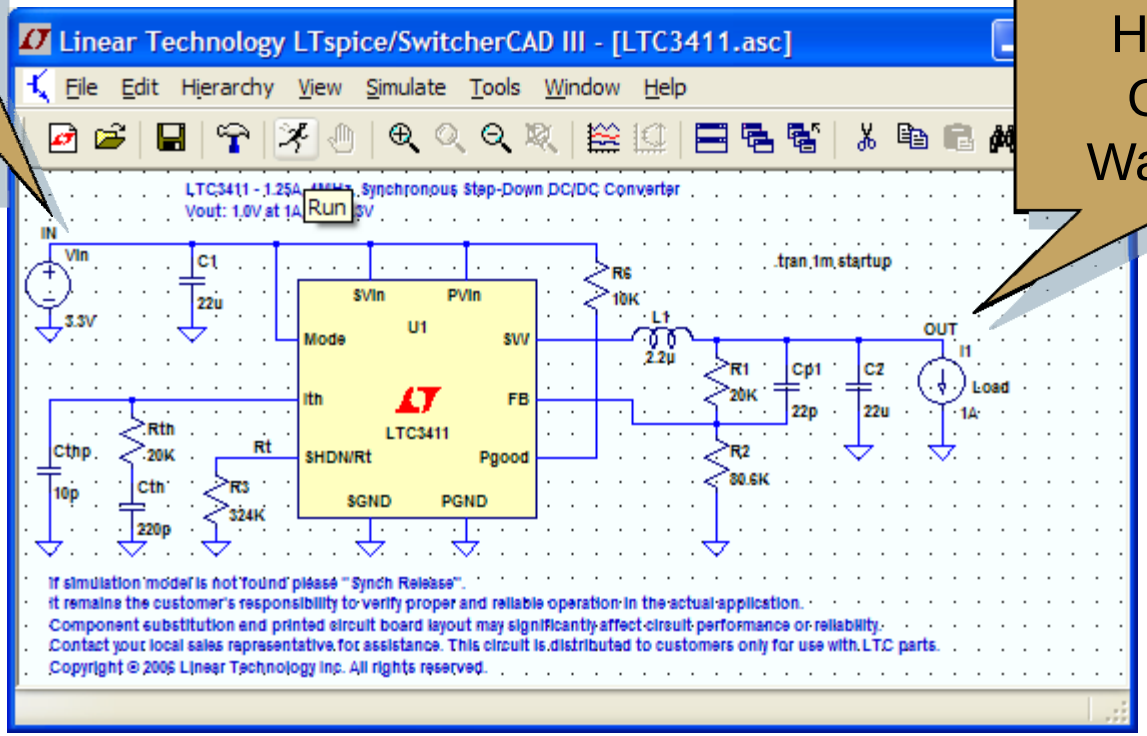




Probing a Demo Circuit and Test Fixture

- Demo Circuits and Test Fixtures have INs and OUTs clearly labeled to help you quickly select them
- To view the waveform left click on IN and OUT

Left Click Here for Input Waveform

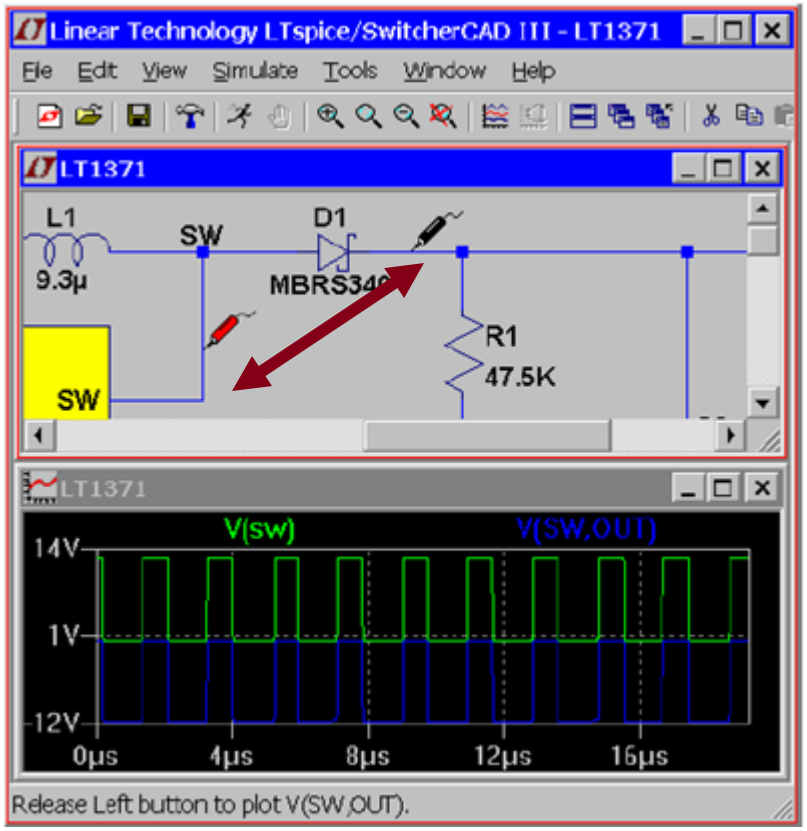


Left Click Here for Output Waveform

Voltage Differences Across Nodes

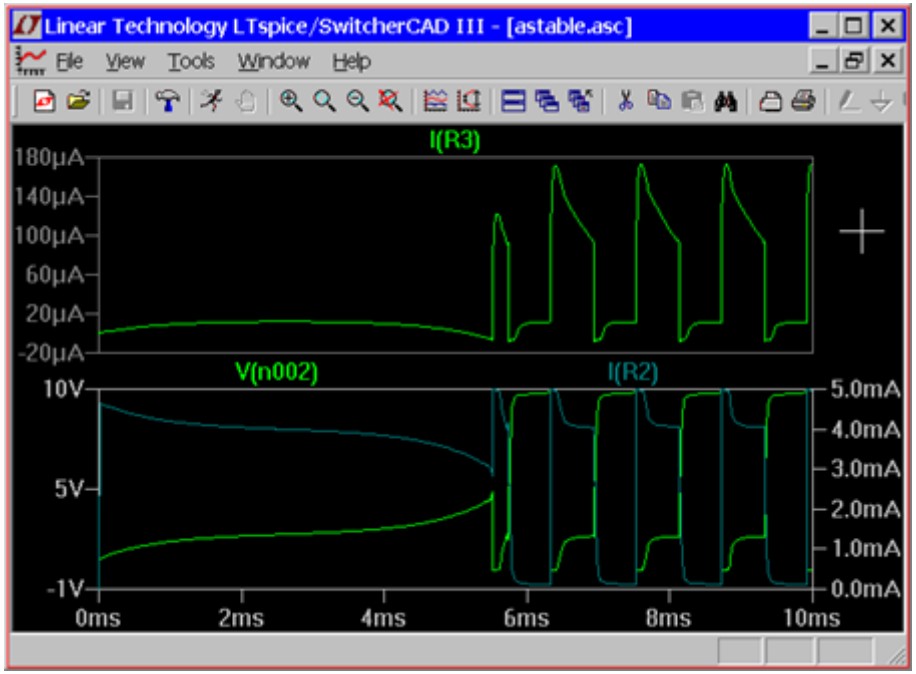
- Left click and hold on one node and drag the mouse to another node
  - Red voltage probe at the first node
  - Black probe on the second

Differential voltages are displayed in the waveform viewer



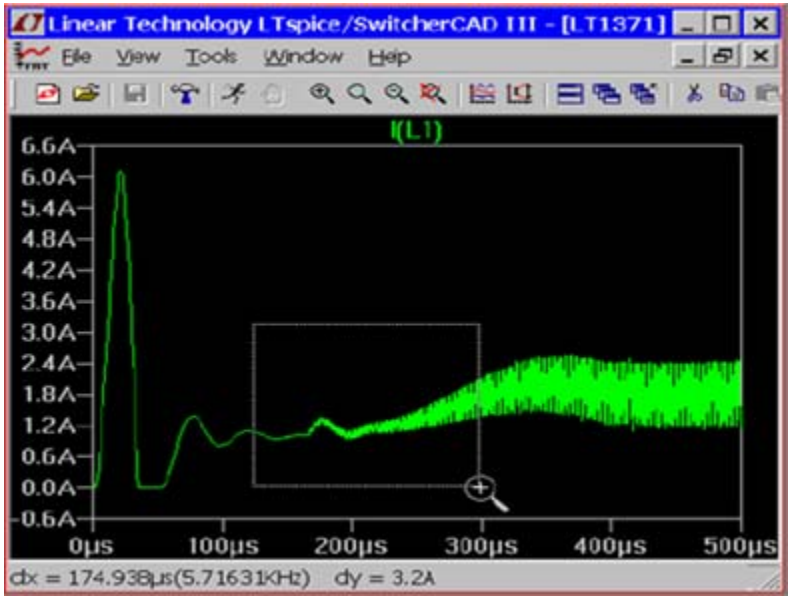
Plot Planes

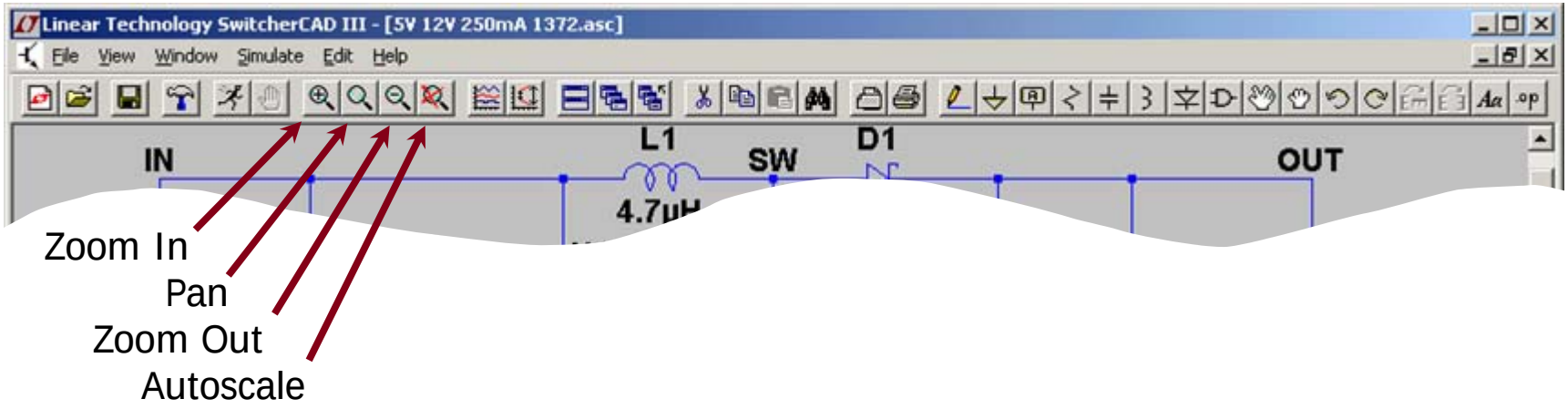
- Multiple plot panes can be displayed on one window to allow better separation between traces permitting different traces to be independently autoscaled
  - Right click in the waveform pane
  - Select Add Plot Pane
  - Left click and hold to drag a label to a new plot pane



Zooming In and Out in the Waveform Viewer

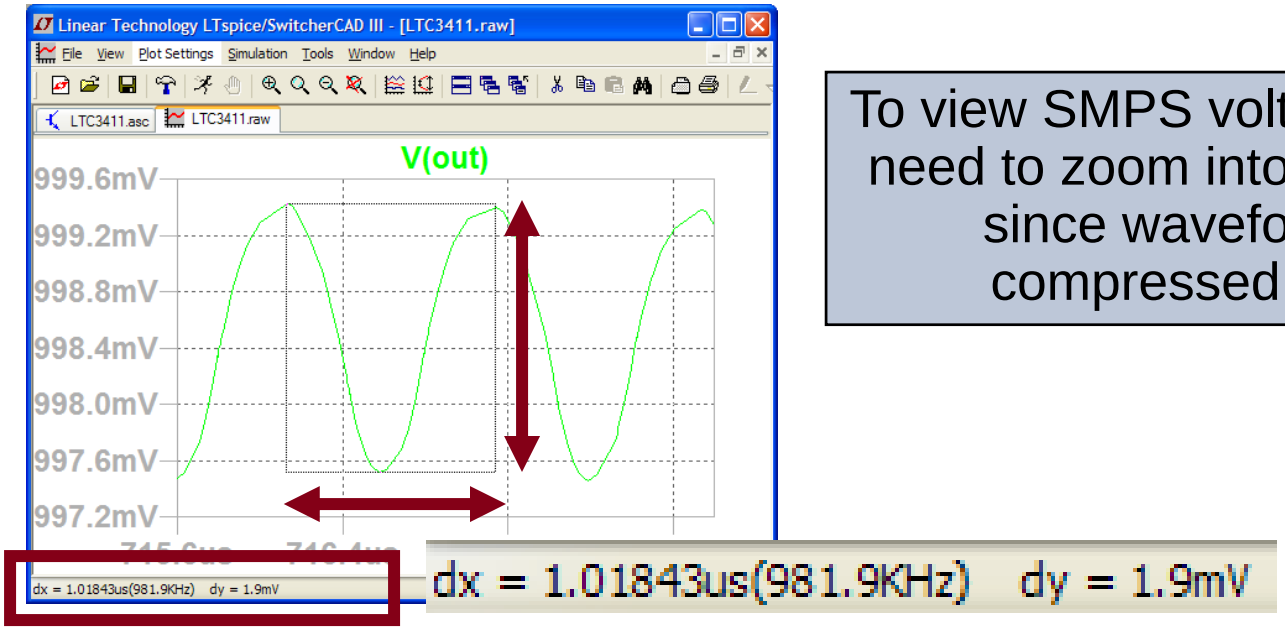
- To zoom in
  - Left click and hold as you drag a box about the region you wish to zoom in then release
- To zoom out
  - Right click and select **Zoom to Fit** or **Zoom Back**





Measuring V_{Ripple}, I_{Ripple} and Time (Frequency)

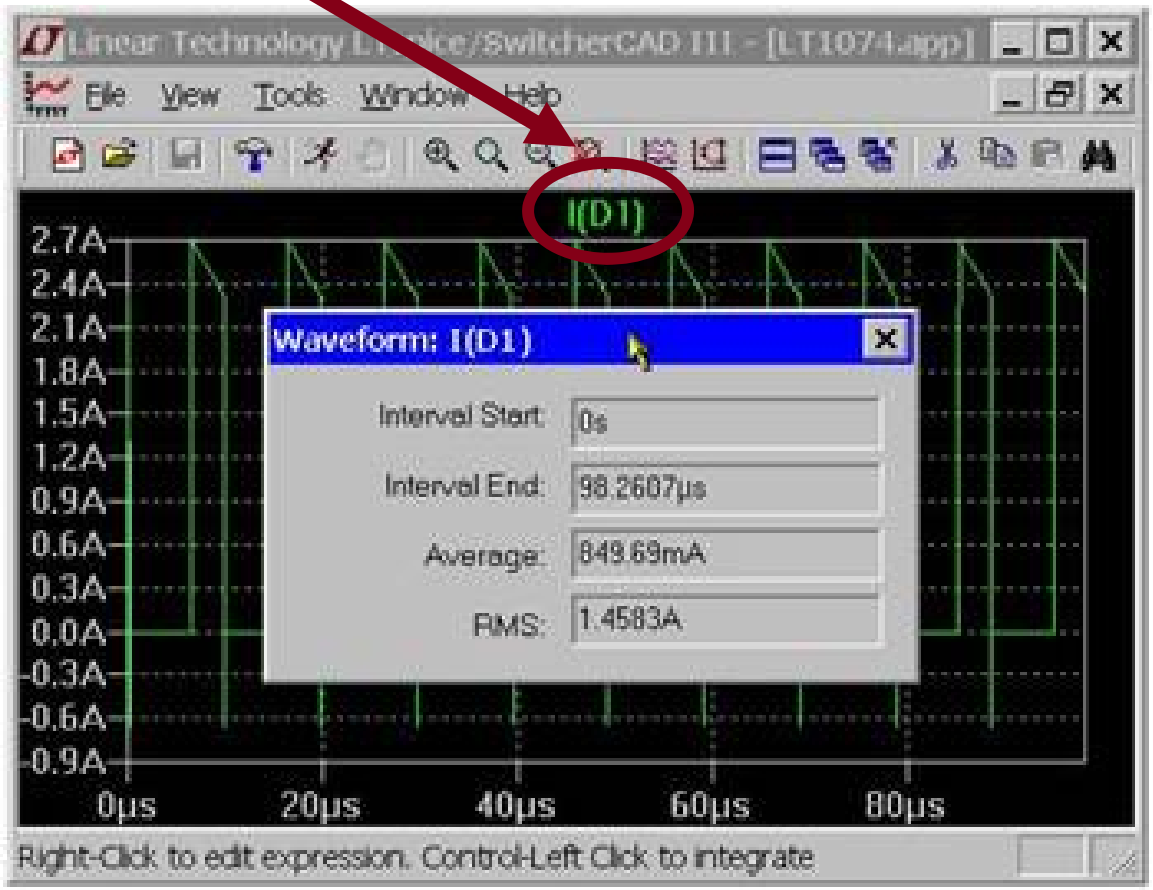
- Drag a box about the region you wish to measure (peak to peak over a period)
  - Left click and *hold* to drag a box over the portion of interest
- View the lower left hand side of the screen
  - To avoid resizing, shrink your box before you let go of the left mouse click or use the Undo command in the Edit menu



To view SMPS voltage ripple you will need to zoom into a narrow section since waveform is initially compressed to full range

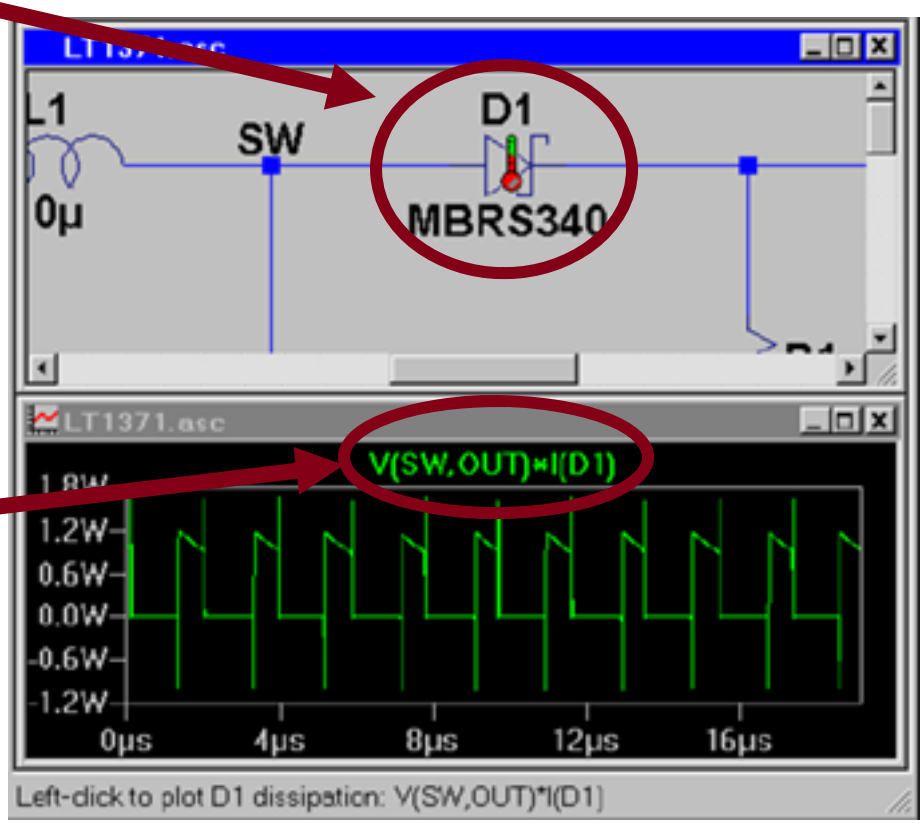
Average/RMS Current or Voltage Calculations

- Hold down Ctrl and left click on the I or V ***trace label*** in the waveform viewer

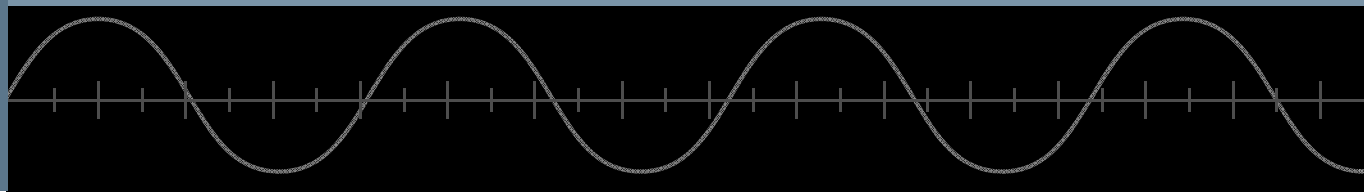


Instantaneous & Average Power Dissipation

- Instantaneous Power Dissipation
  - Hold down the ALT key and left click on the symbol of the component
  - Pointer will change to a thermometer
  - Plotted in units of Watts
- Average Power Dissipation
  - Hold down the Ctrl key and left click on the ***trace label*** power dissipation waveform



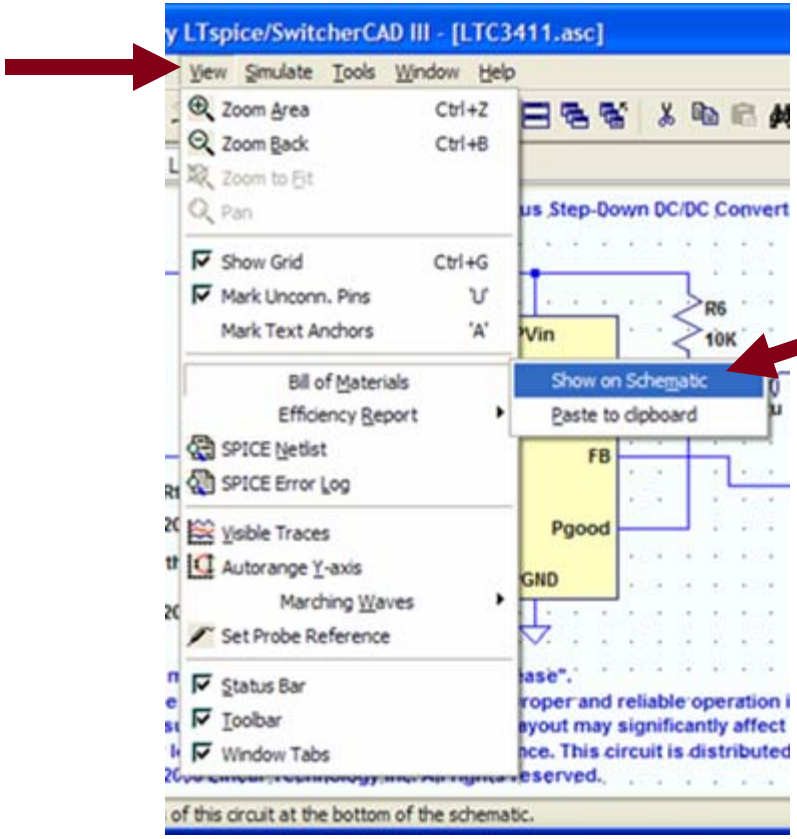
Generating a BOM and Efficiency Report

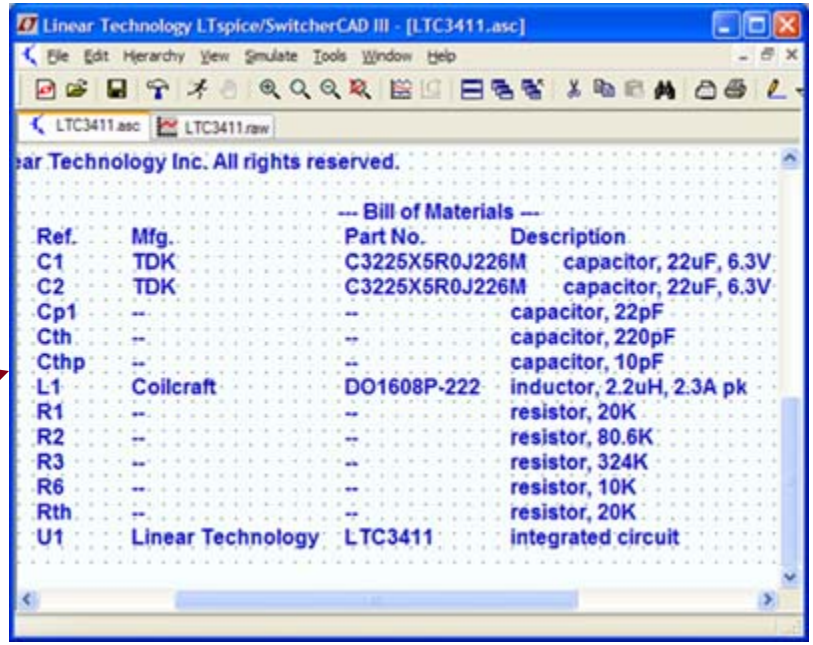


LINEAR TECHNOLOGY

Bill of Materials (BOM)

- Left click on **View** menu
- Left click on **Bill of Materials**

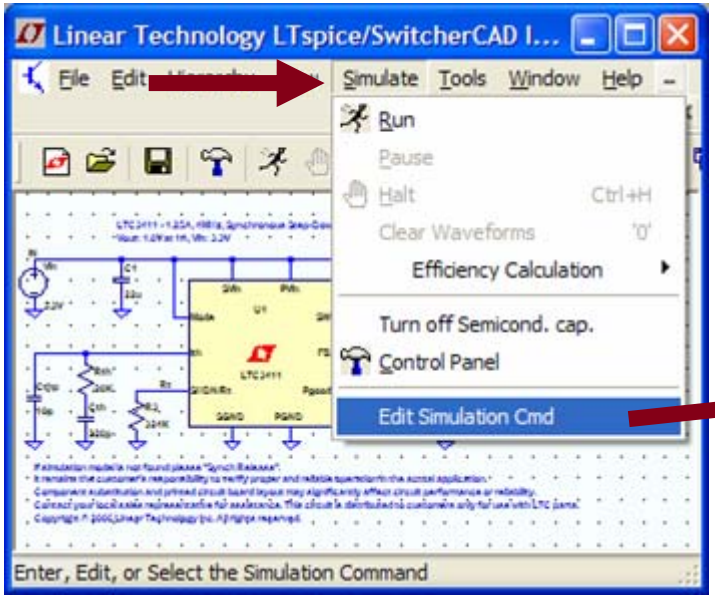


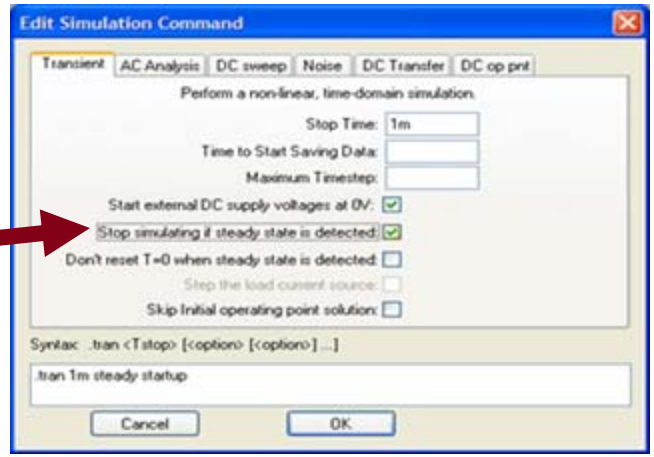


Computing Efficiency of SMPS Circuits

- Left click on **Simulate** menu
- Left click on **Edit Simulation Cmd**
- Left click on **Stop simulating if steady state is detected**
  - Automatically detect the steady state by checking the internal state of the macromodels
- Rerun simulation

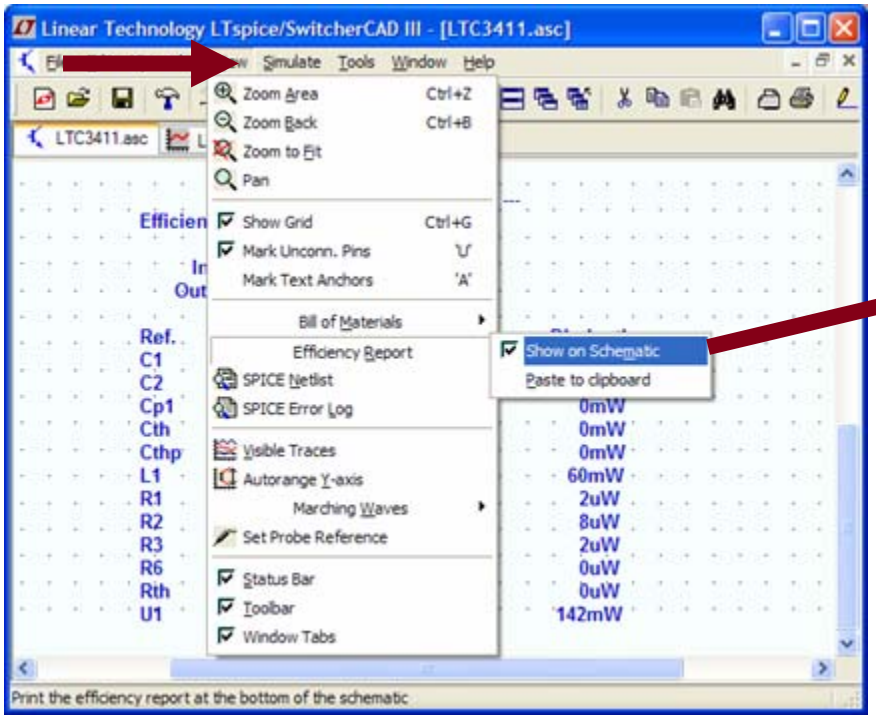
Automatic detection of steady state may not work – steady state detection may be too strict or lenient

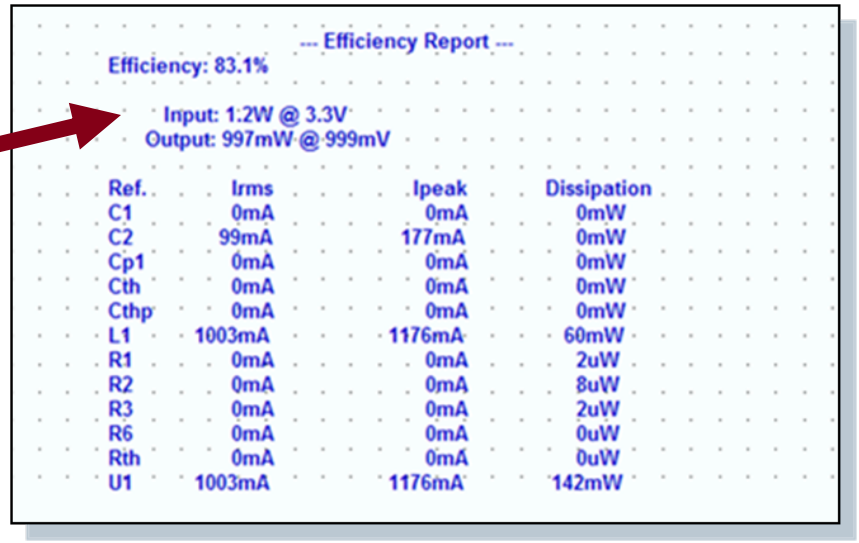




Viewing Efficiency Report

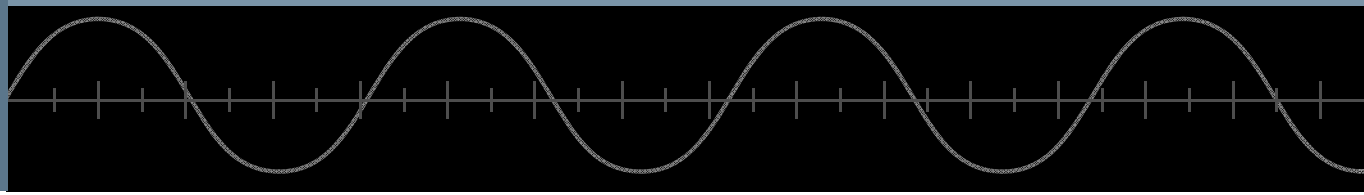
- Left click on **Simulate** menu
- Left click on **Efficiency Report**





Simulate a Transient Response in a SMPS

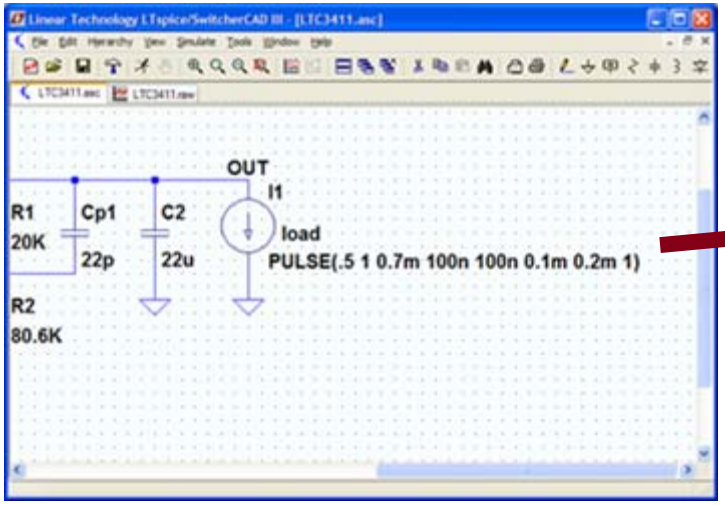
Advanced Topic

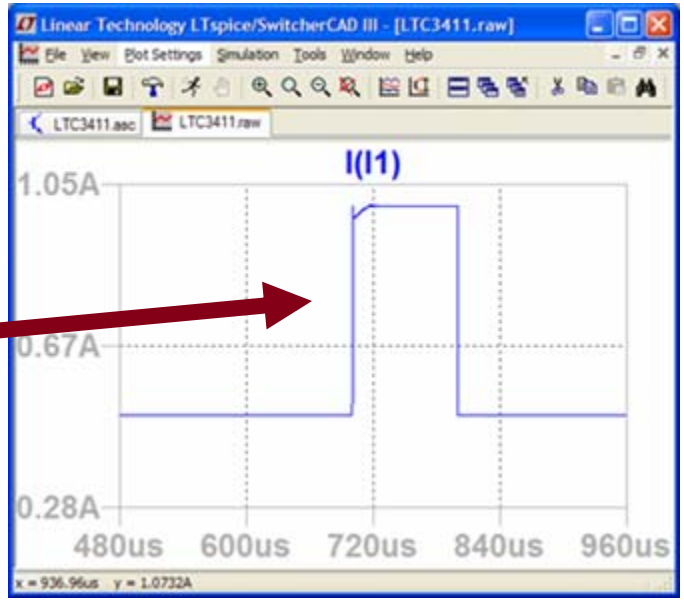


LINEAR TECHNOLOGY

Use a Pulsed Function as a Transient Response Load

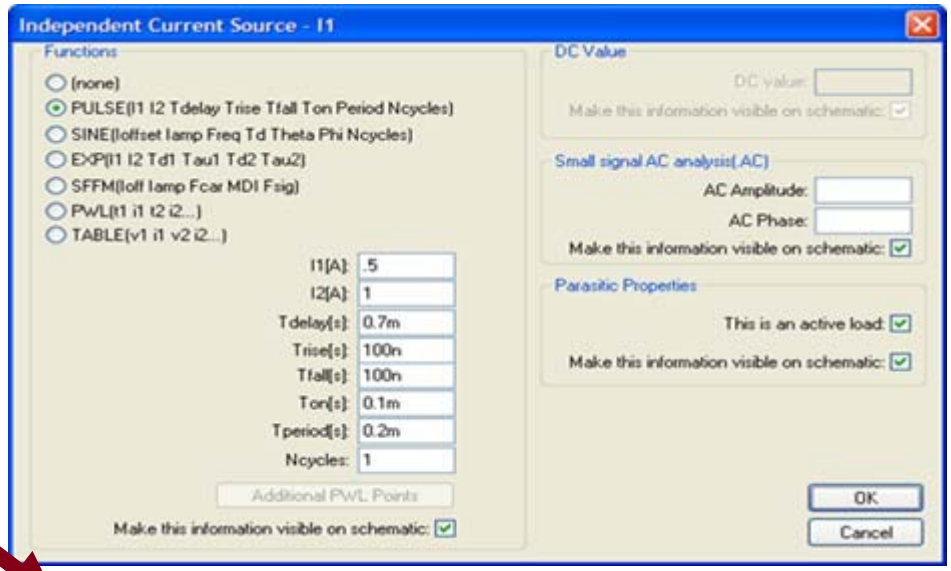
- Insert a current source load
  - Left click on the Component symbol in the Schematic Editor Toolbar
  - Select load (or load2) circuit element and configure as pulsed
  - Left click on OK
- Configure load as a pulsed function (covered next)
  - Steps current from initial to pulsed value and back
- Run and review results





Configuring Load as a Pulse Function

- Right click on the load (or load2) component
- Select Pulse
- Modify the Attributes
  - I1 = Initial value
  - I2 = Pulsed Value
  - Tdelay = Delay
  - Tr = Rise time
  - Tf = Fall time
  - Ton = On time
  - Tperiod = Period
  - Ncycles = Number of cycles
    - Omit for free running

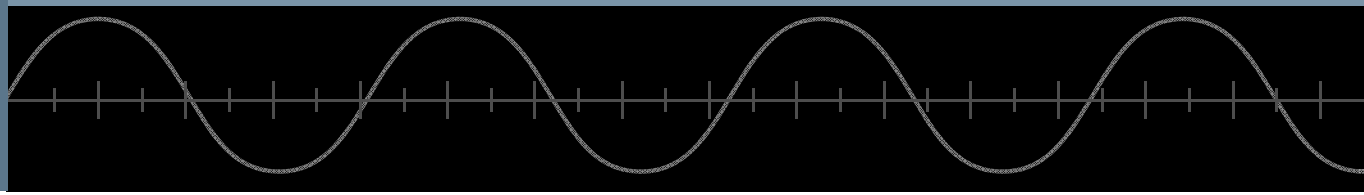


Tdelay needs to be adequate so that the device is in steady state and out of startup before the load step occurs

You may need to un-click **Stop simulating if steady state is detected** and specify an end time in Edit Simulation Cmd under the Simulate menu

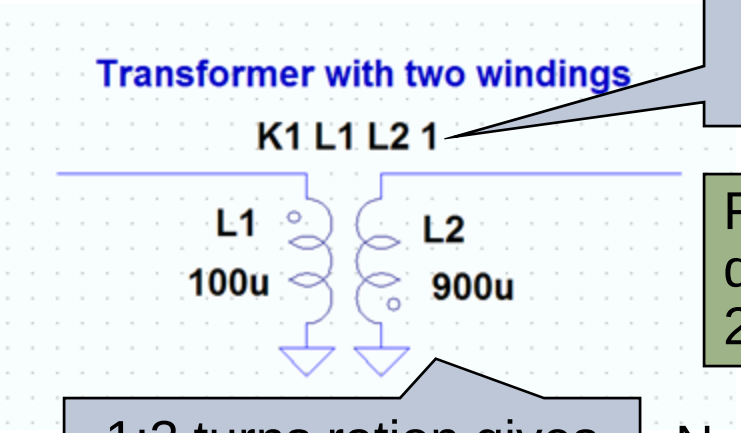
Simulate a Transformer

Advanced Topic



Simulating a Transformer

- Draw each winding of the transformer as an individual inductor
- Couple inductors with a mutual inductance statement
  - Add a SPICE directive of the form K1 L1 L2 L3 ... 1 to the schematic
    - Left Click on **Edit** then **SPICE Directive**
    - Inductors in a mutual inductance will be drawn with a phasing dot
  - Start initially with a mutual coupling coefficient equal to 1



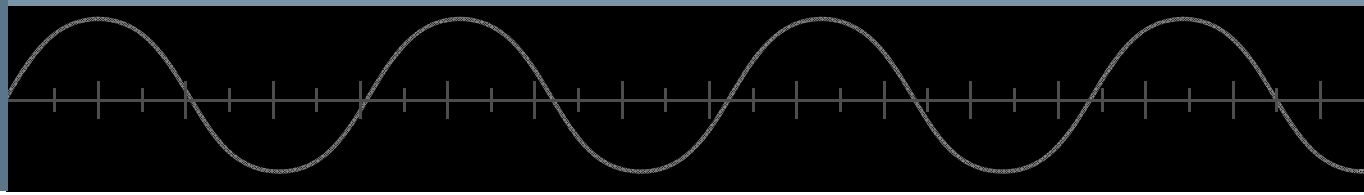
K statement coupling the windings

For more information check out LTC1871 demo circuit and page 23-24 of September 2006 LT Magazine at www.linear.com

1:3 turns ration gives a 1:9 inductance ratio

Note: winding inductance ratio is the square of the turns ratio

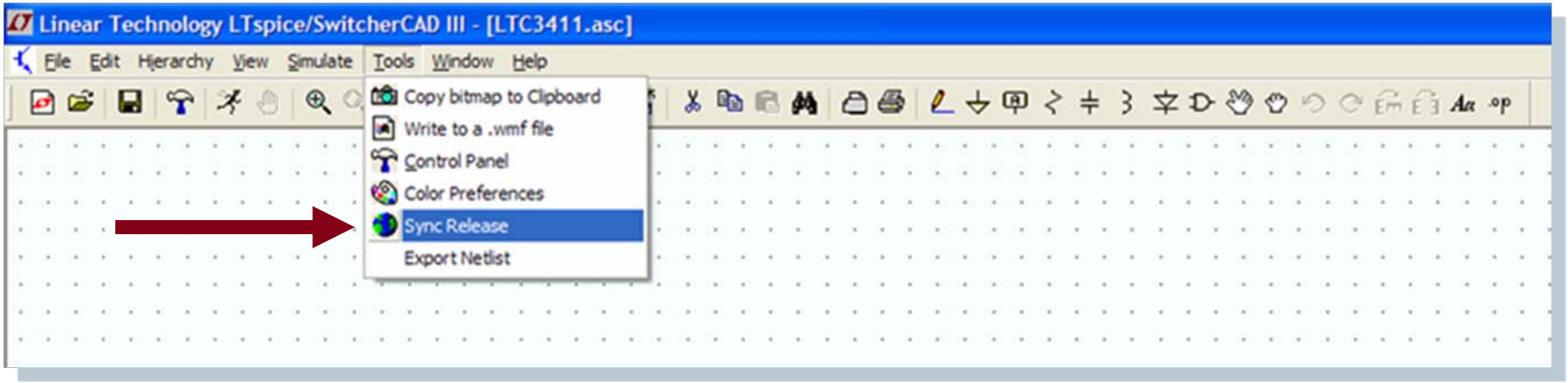
Additional Information and Support



Reminder to Periodically Sync Release

- Update your release of LTspice to get the latest
  - Software updates
  - Models and examples

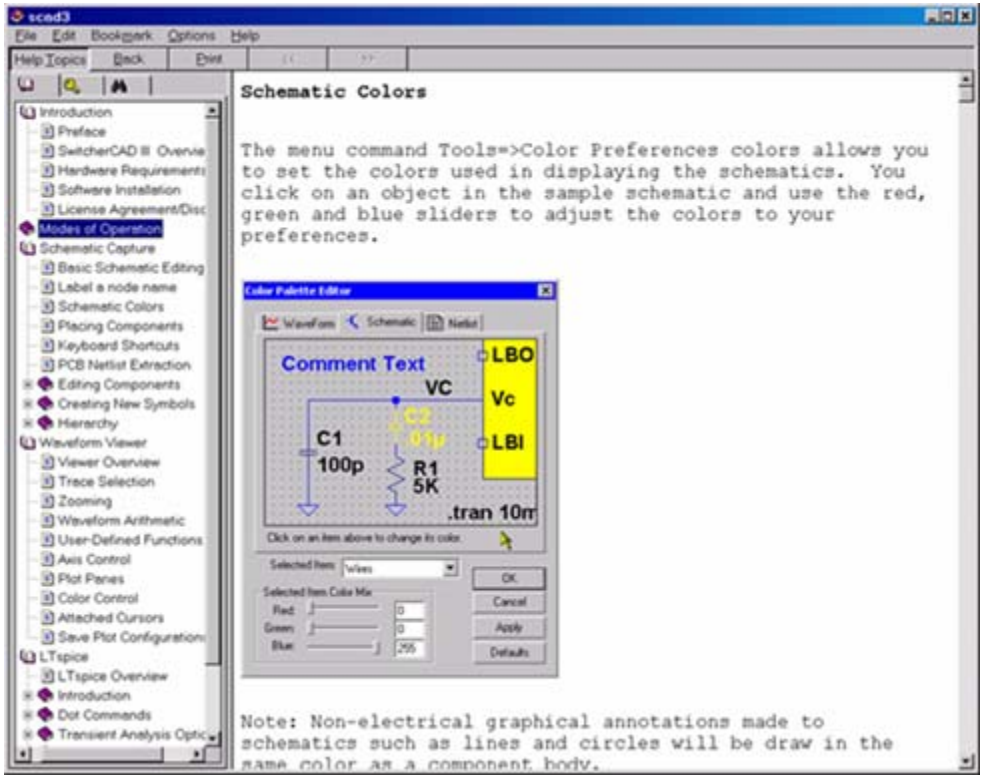
Sign up for *Linear Insider* via MyLinear (www.linear.com) for email news and updates



List of changes are available in the changelog.txt that is located in your LTspice root directory (C:\Program Files\LTC\SwCADIII)

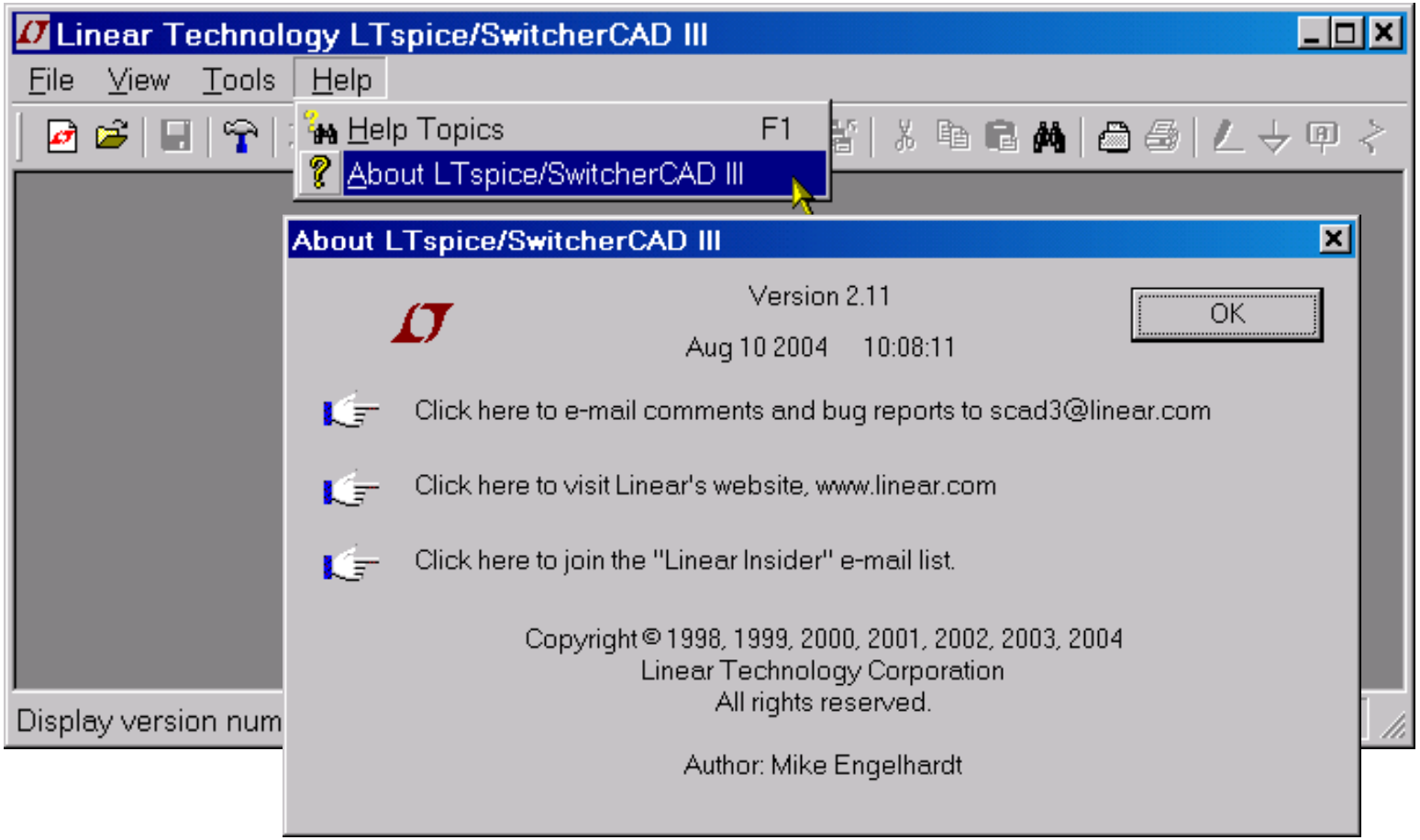
Built-in Help System

- Left Click on **Help** menu and then **Help Topics**



To print out a hardcopy, download user guide at
http://LTspice.linear.com/software/scad3.pdf

Emailing Comments and Signing up for Linear Insider



Customer Support

- *Linear Technology customers* can obtain support by
  - Calling your local field applications engineer
    - http://www.linear.com/contact/
  - Calling +1 (408) 432 – 1900 for factory application support
- *Additional support* (not related to Linear Technology circuits or models support)
  - Built-in help topics & User Manual
  - Independent LTspice users' group (search messages)

Simulation with the supplied models is fully supported

All bug reports are appreciated and will be resolved

Independent LTspice Users' Group

- The group has a section of ***files*** and ***messages*** with additional tutorials, libraries, and examples
  - http://groups.yahoo.com/group/LTspice/
- Join LTspice Users' Group
  - Email LTspice-subscribe@yahoogroups.com
    - Subject=Subscribe