# LTspice IV Getting Started Guide

LINEAR TECHNOLOGY
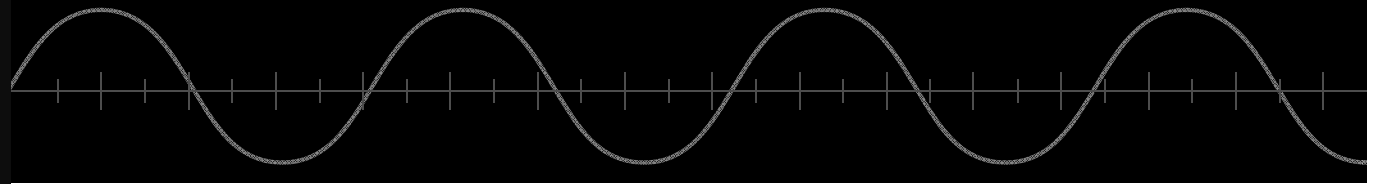
# Benefits of Using LTspice IV

◆ Stable SPICE circuit simulation with

  ◆ Unlimited number of nodes

  ◆ Schematic/symbol editor

  ◆ Waveform viewer

  ◆ Library of passive devices

Outperforms pay-for options

LTspice is also a great schematic capture

◆ Fast simulation of switching mode power supplies (SMPS)

  ◆ Steady state detection

  ◆ Turn on transient

  ◆ Step response

  ◆ Efficiency / power computations

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

◆ Advanced analysis and simulation options

  ◆ Not covered in this presentation

LINEAR TECHNOLOGY

# How Do You Get LTspice IV

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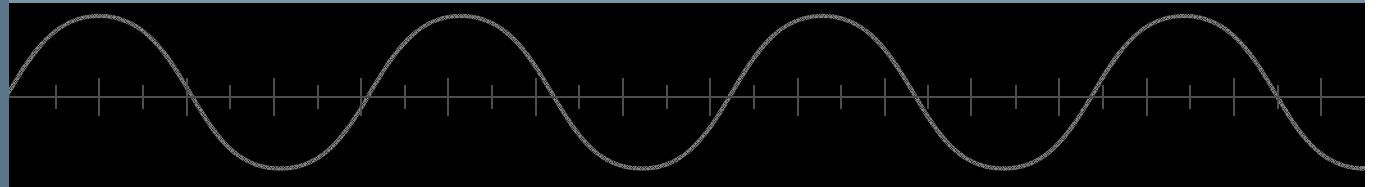
**LTspice IV**

LTspice IV is a high performance SPICE 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 LTspice IV, Macro Models for 80% of Linear Technology's switching regulators, over 200 op amp models, as well as resistors, transistors and MOSFET models.

- **Download LTspice IV (Update October 8, 2011 )**
- LTspice Users Guide
- LTspice Getting Started Guide
- Using Transformers in LTspice IV
- LTspice Demo Circuit Collection
- View upcoming LTspice seminars

Follow LTspice on Twitter!

View the LTspice Video Channel

# Getting Started

# Getting Started using LTspice IV

◆ **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 on the left side

- ◆ Follow the instructions provided

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

OVERVIEW

PACKAGING

ORDER INFO

SIMULATE

DEMO BOARDS

TECH SUPPORT

**Simulate**

LTspice IV is a powerful, fast and free simulation tool, schematic capture and waveform viewer with enhancements and models for improving the simulation of switching regulators. To download LTspice IV and other FREE simulation tools, please visit our Design Simulation page.

To launch a ready to run LTspice demonstration circuit for this part:

- **Step 1:** If you have not installed LTspice on this computer, download and install LTspice IV

Download LTspice

- **Step 2:** Once LTspice is installed, click on the link(s) below to launch the simulation

  - LTC3411 Demo Circuit - 1.25A, 4MHz, Synchronous Step-Down DC/DC Converter (3.3V to 1.0V @ 1A)

Download Demo Circuit

- **Step 3:** If LTspice IV does not automatically open after clicking the link above, you can instead run the simulation by right clicking on the link and selecting "Save Target As". After saving the file to your computer, start LTspice and open the demonstration circuit by selecting 'Open' from the 'File' menu

To explore other ready to run LTspice demonstration circuits, please visit our Demo Circuits Collection.

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

LINEAR TECHNOLOGY

# Demo Circuit

✓ Designed and Reviewed by Factory Apps Group
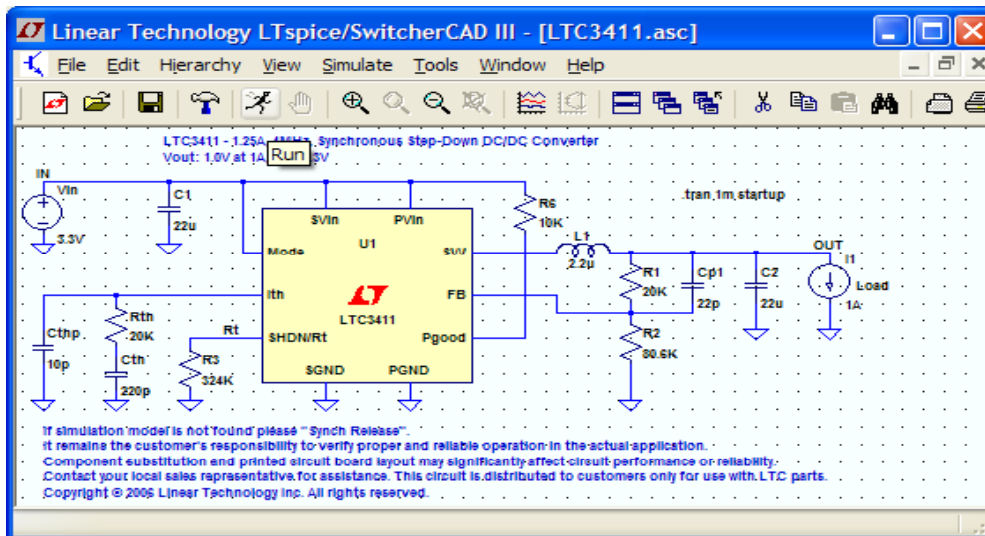


To run a demo circuit jump to the *Run and Probe a Circuit in LTspice* section

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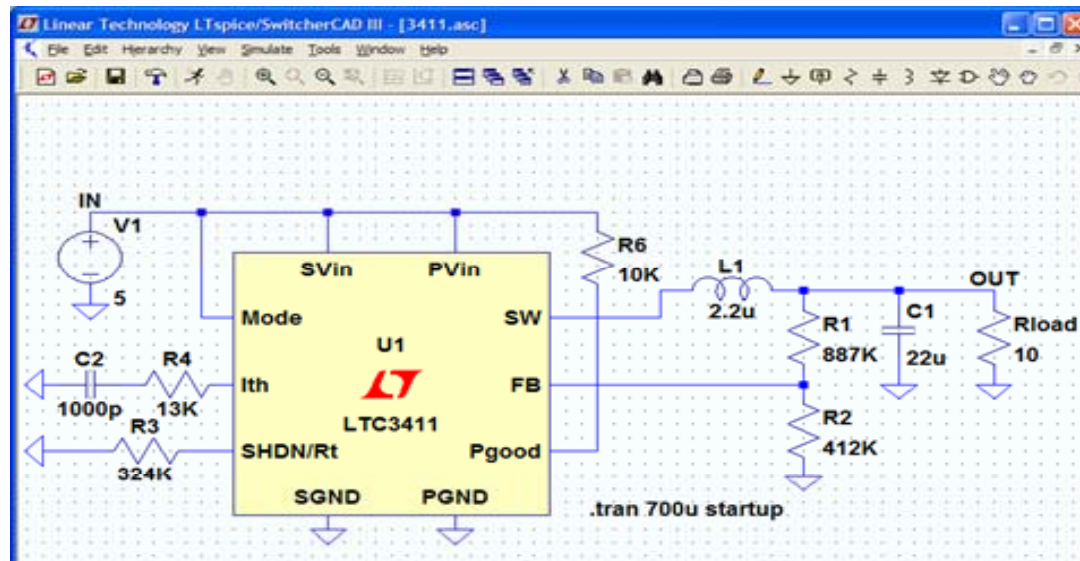- Printed circuit board layout may significantly affect circuit performance or reliability

# Getting Started using LTspice IV

◆ 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
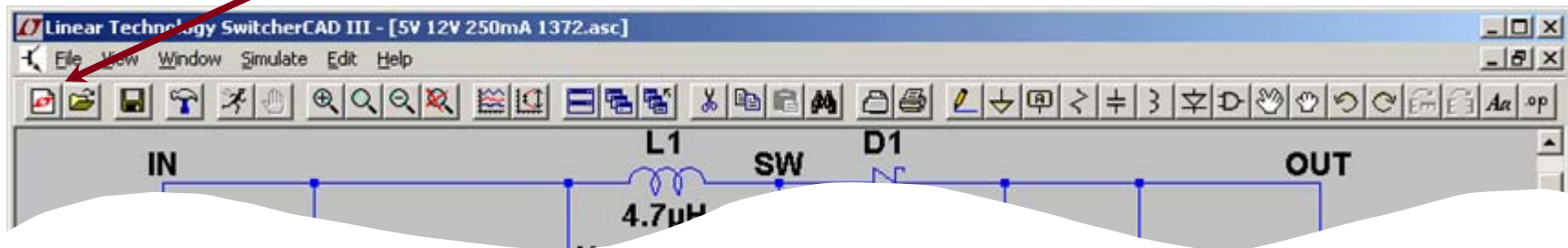
New Schematic



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

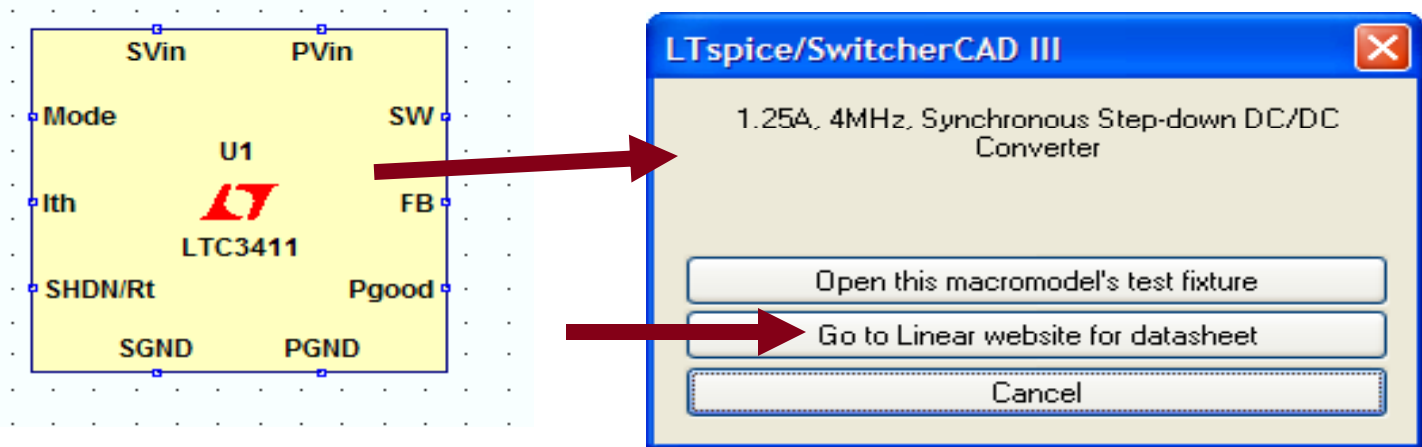# Add a Macromodel & Opening Test Fixture

Add Component
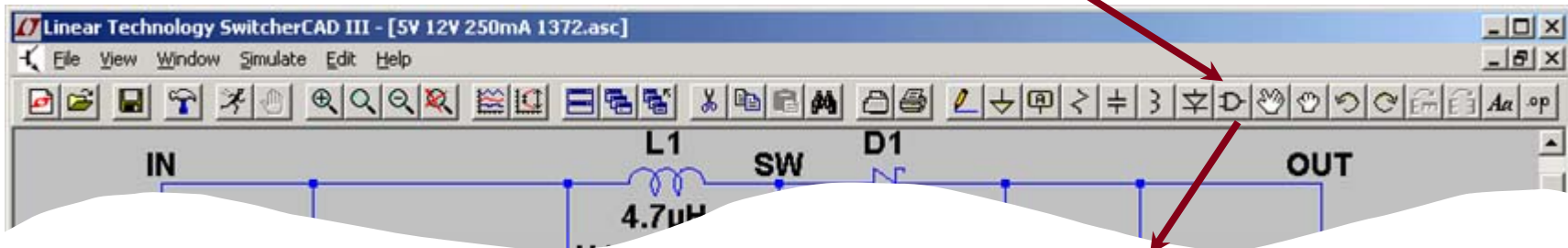


- Left click on the **Component** symbol in the Schematic Editor Toolbar

- 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
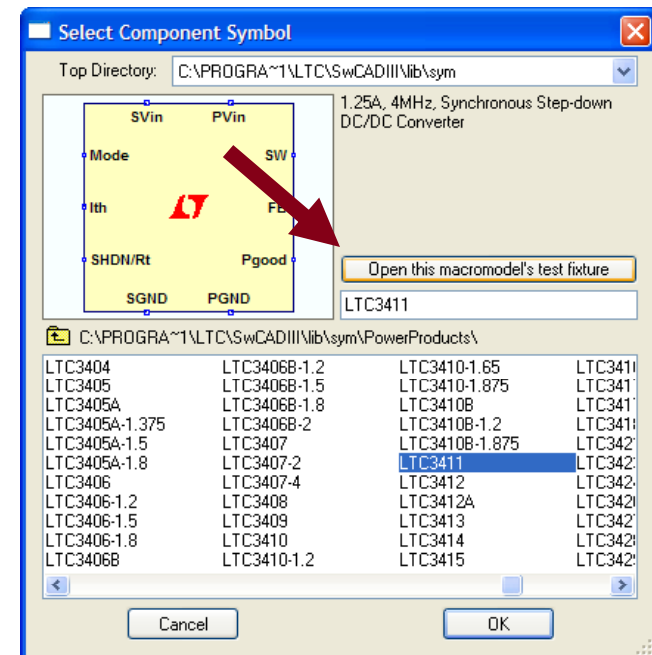
# Getting Started using LTspice IV

◆ 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
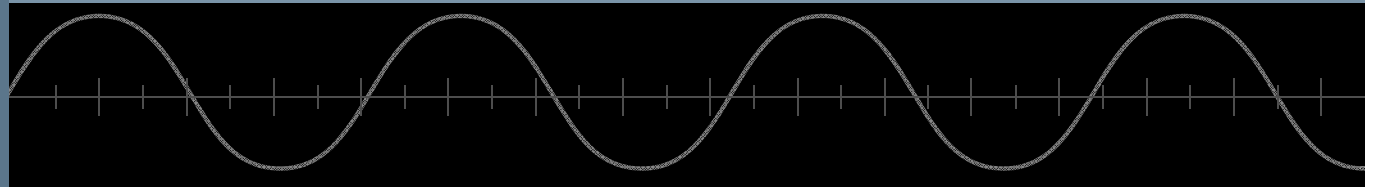
# Start with a New Schematic
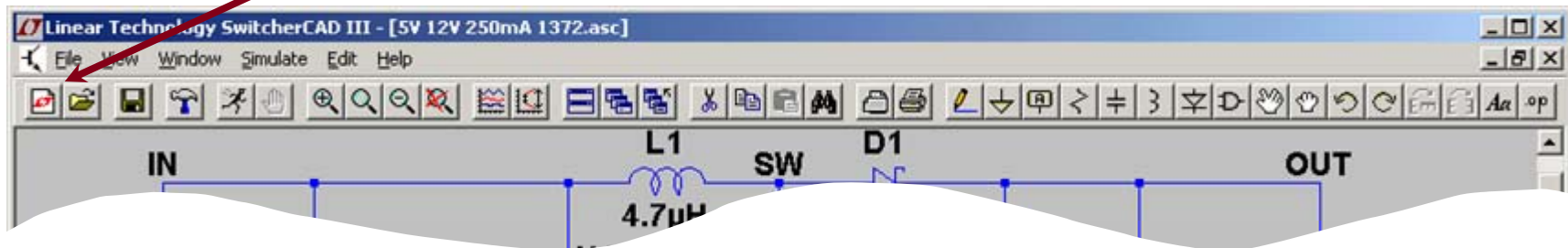
New Schematic



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

LTspice is also a great schematic capture

# Add a Linear Technology Macromodel

Add Component
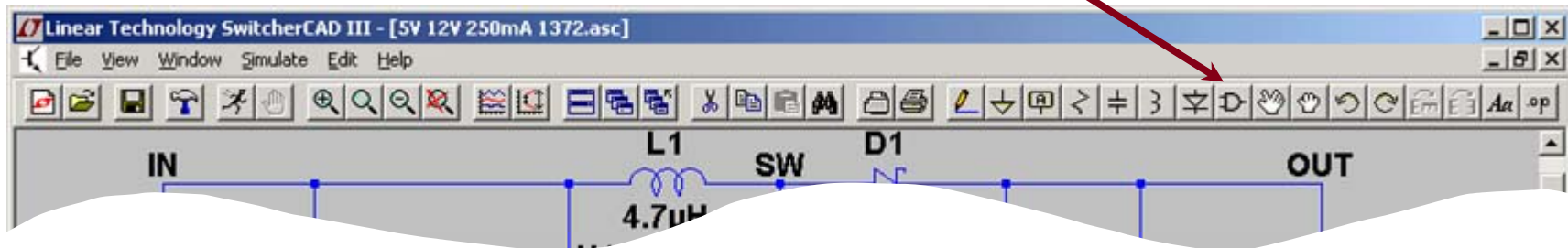


- ◆ Left click on the **Component** symbol in the Schematic Editor Toolbar
- ◆ Enter "root" part to search for the model (e.g. 3411)
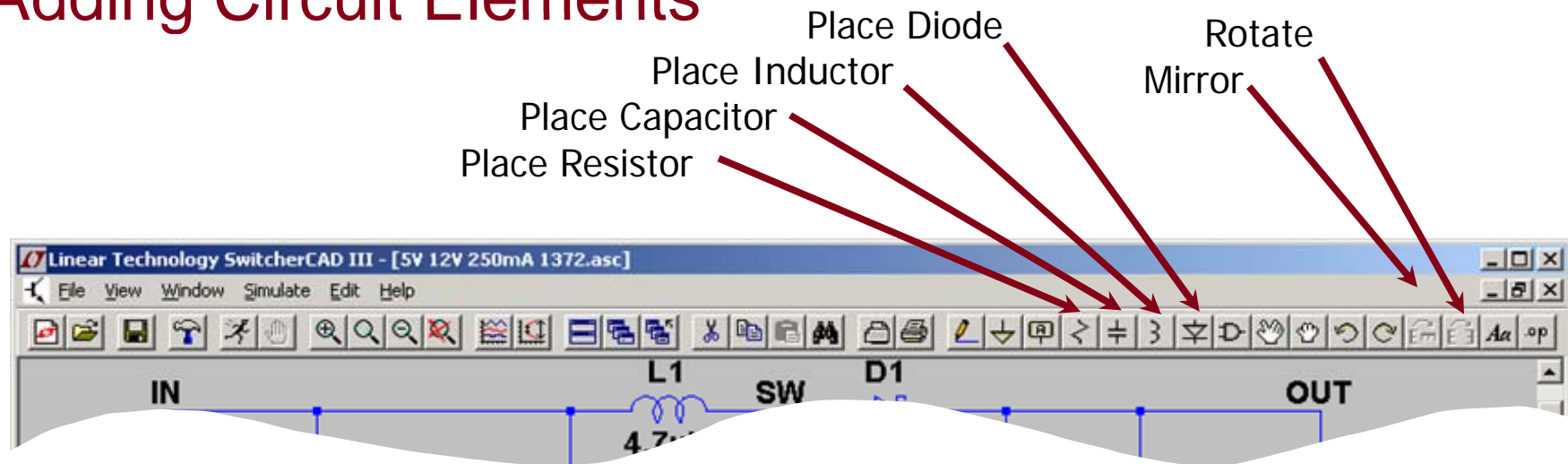- ◆ Left click on **OK**
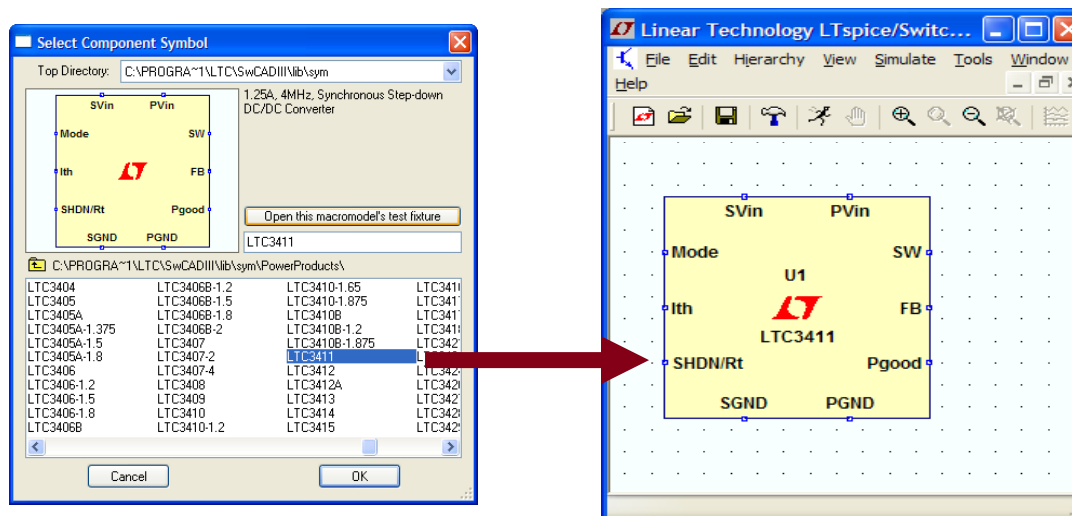
# Getting the Latest Datasheet

◆ 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
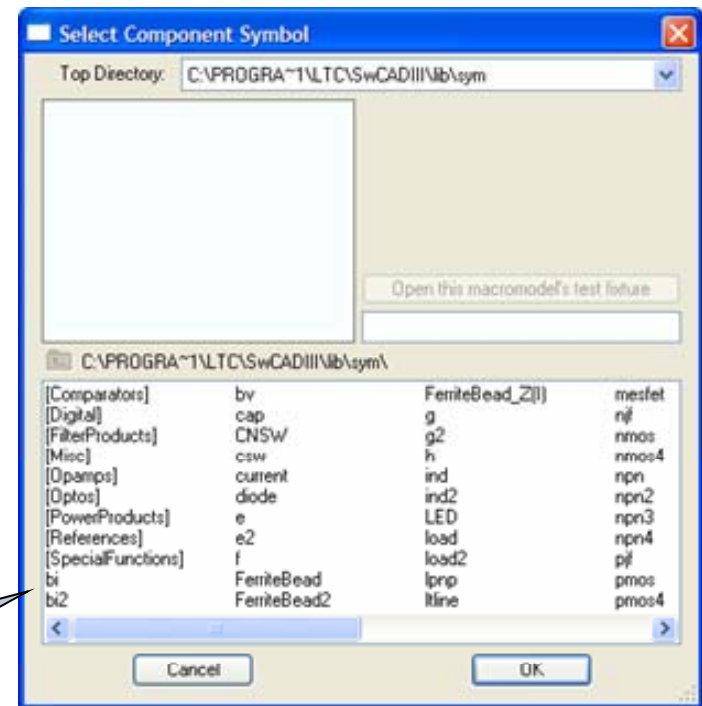
# Adding Circuit Elements

Place Diode

Rotate

Place Inductor

Mirror

Place Capacitor

Place Resistor



- 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
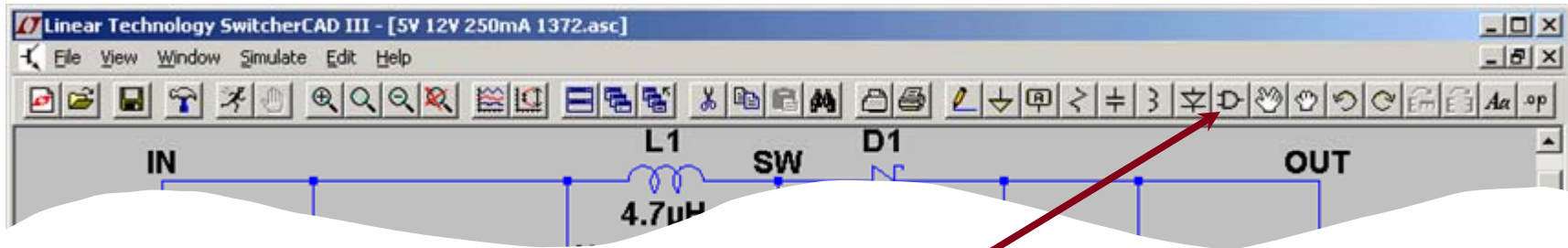


- ◆ Left click on the **Component** symbol in the Schematic Editor Toolbar

- ◆ 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
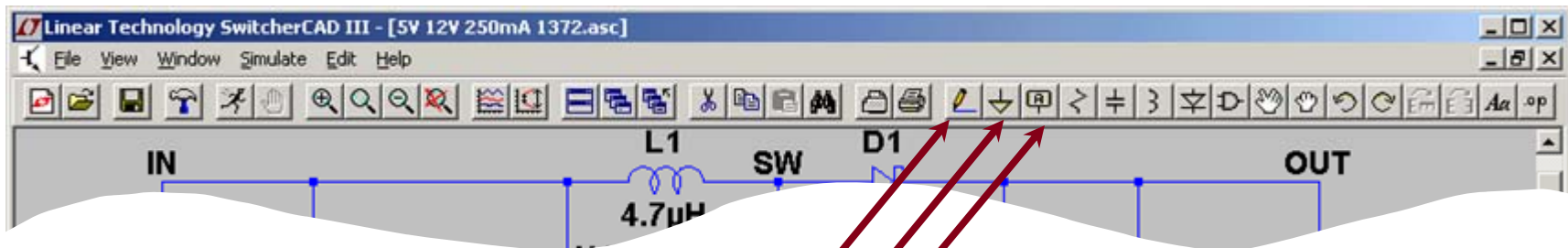
- ◆ Left click to place it

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



Draw Wire
Place Ground
Label Node

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

Delete
Duplicate
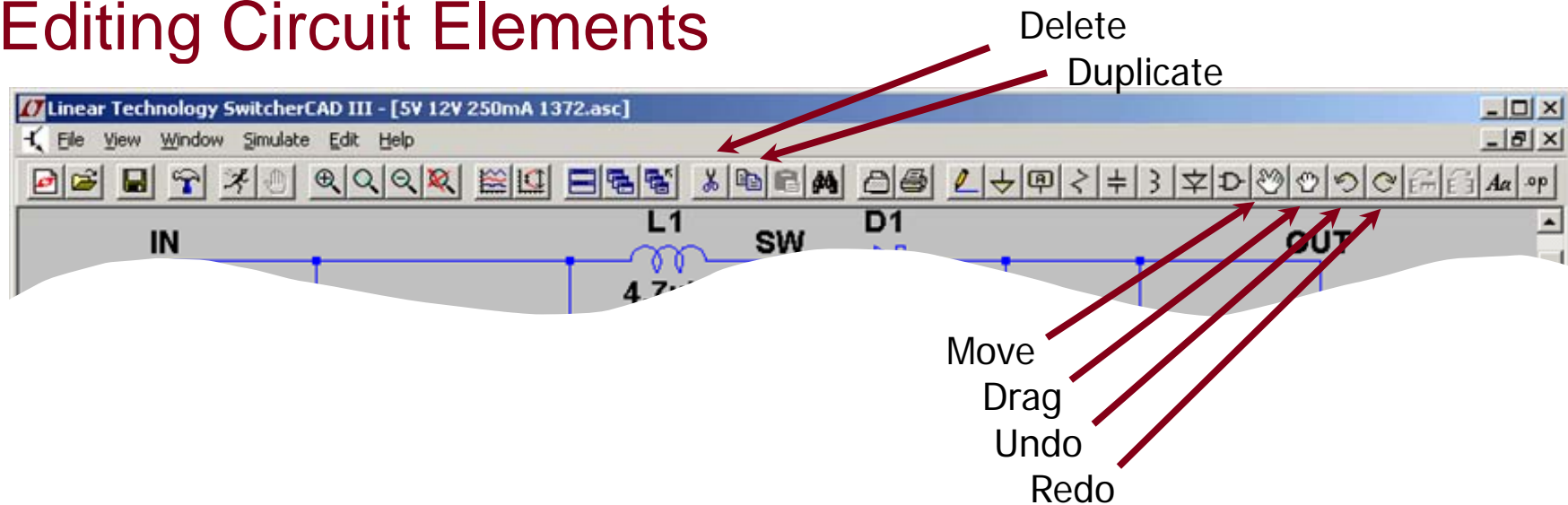
Move
Drag
Undo
Redo

◆ **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

LINEAR TECHNOLOGY

# Editing Circuit Elements Attributes

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

**Resistor - R6**

Manufacturer: --------  
Part Number: --------  

[ OK ]  
[ Select Resistor ]  [ Cancel ]  

Resistor Properties  
Resistance[Ω]: `10K`  
Tolerance[%]: [ ]  
Power Rating[W]: [ ]  

**Inductor - L1**

Manufacturer: Coilcraft  
Part Number: DO1608P-222  

[ OK ]  
[ Select Inductor ]  [ Cancel ]  
Show Phase Dot ☐  

Inductor Properties  
Inductance[H]: `2.2u`  
Peak Current[A]: `2.3`  
Series Resistance[Ω]: `0.06`  
Parallel Resistance[Ω]: `55000`  
Parallel Capacitance[F]: `1.8p`  
(Series resistance defaults to 1mΩ)  

**Capacitor - Cp1**

Manufacturer: --------  
Part Number: --------  
Type: --------  

[ OK ]  
[ Select Capacitor ]  [ Cancel ]  

Capacitor Properties  
Capacitance[F]: `22p`  
Voltage Rating[V]: [ ]  
RMS Current Rating[A]: [ ]  
Equiv. Series Resistance[Ω]: [ ]  
Equiv. Series Inductance[H]: [ ]  
Equiv. Parallel Resistance[Ω]: [ ]  
Equiv. Parallel Capacitance[F]: [ ]  
Mean Time Between Failures[hr]: [ ]  
Parts Per Package: [ ]  

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

**LINEAR** TECHNOLOGY

# 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 = $10^{-15}$

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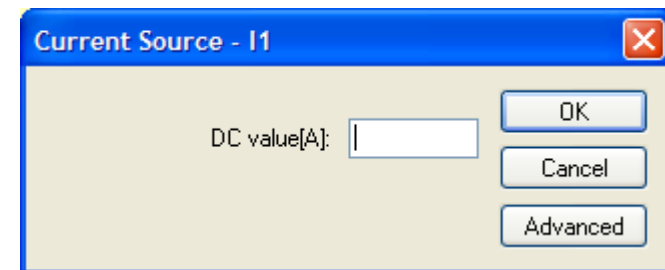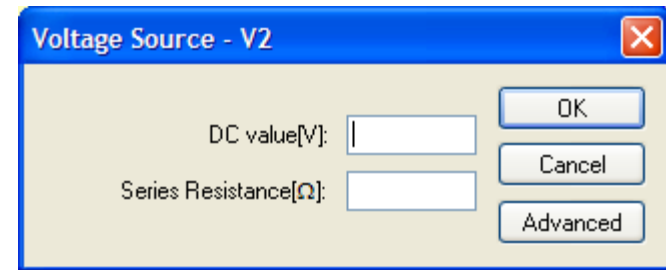- Enter *1* for 1 Farad, not *1F*

LINEAR
TECHNOLOGY

# 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
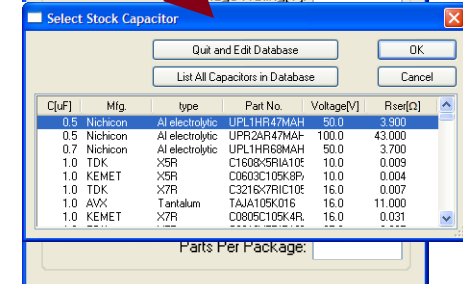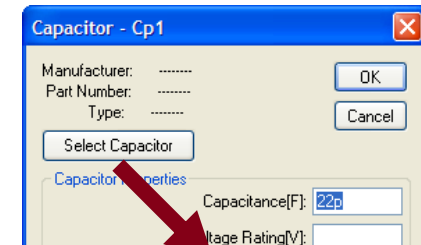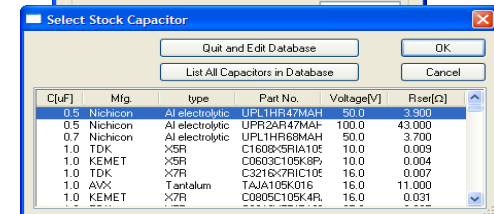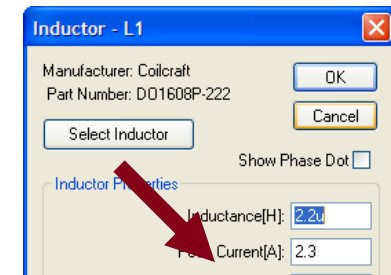
# Editing Voltage Sources and Loads

◆ 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

Place Circuit Element
Place Diode
Place Inductor
Place Capacitor
Place Resistor
Label Node
Place Ground
Draw Wire



Move
Drag
Undo
Redo
Rotate
Mirror
Place Comment
Place SPICE directive

Delete
Duplicate
Paste b/t Schematics
Find
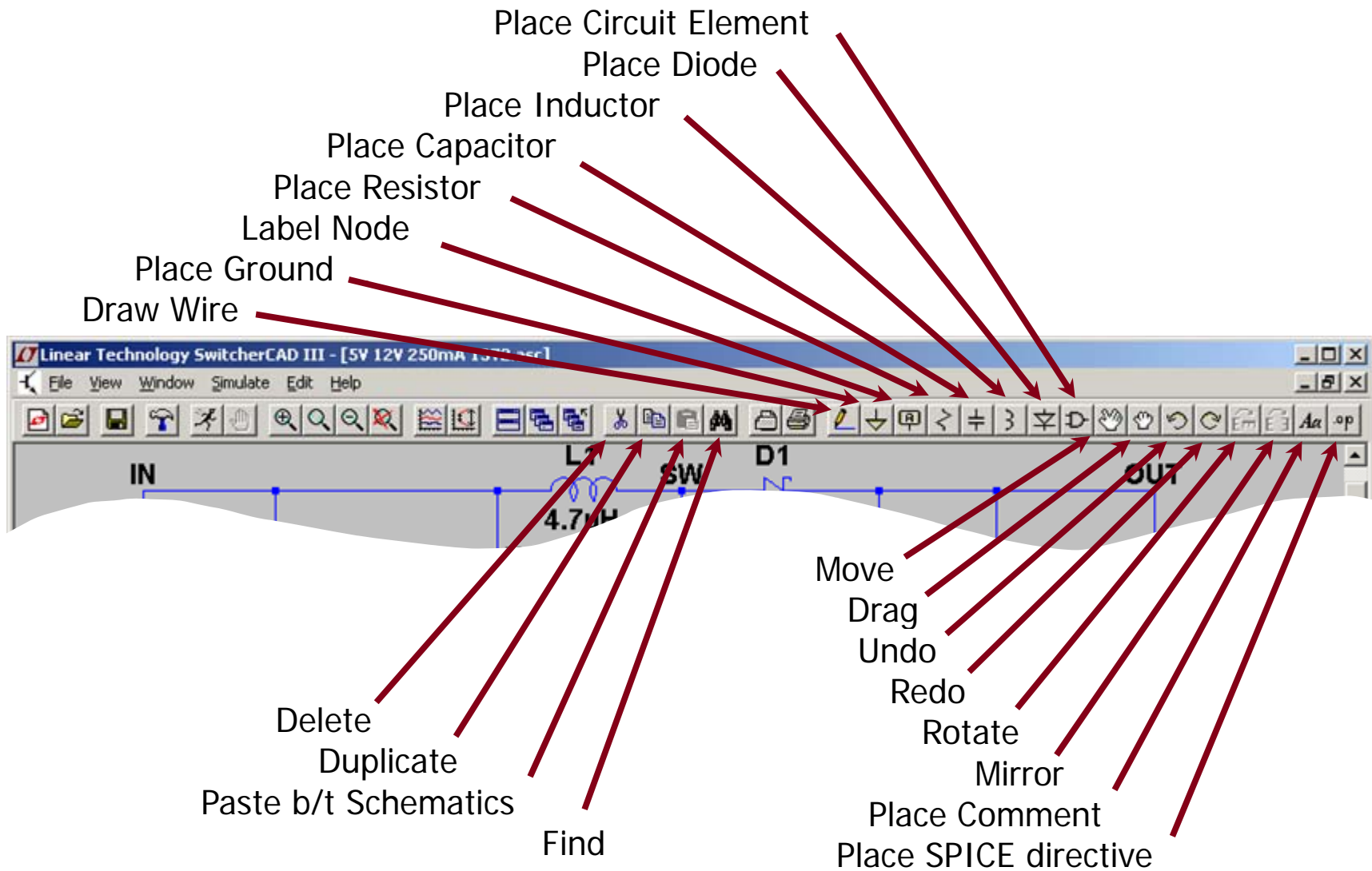
# Run and Probe a Circuit
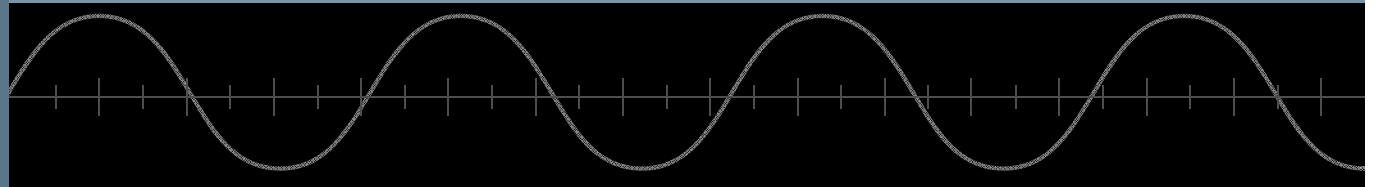
# Simulation Commands

◆ 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

More information on simulation and dot commands are available in LTspice IV User Guide

◆ Simulation commands are placed on the schematic as text

- Called dot commands

LINEAR TECHNOLOGY

# 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
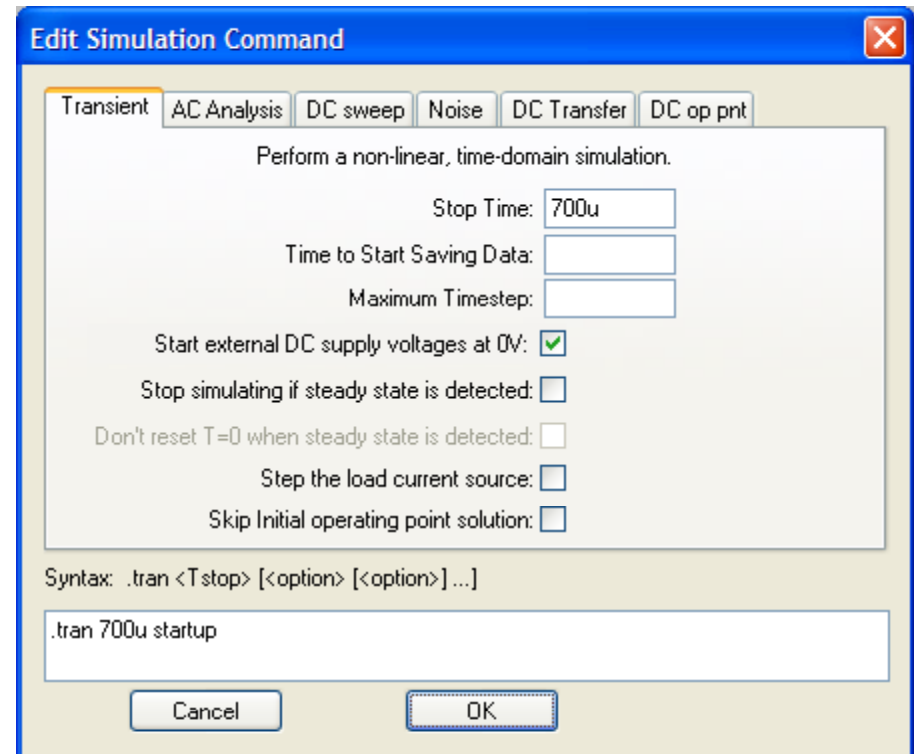
# Running a Circuit
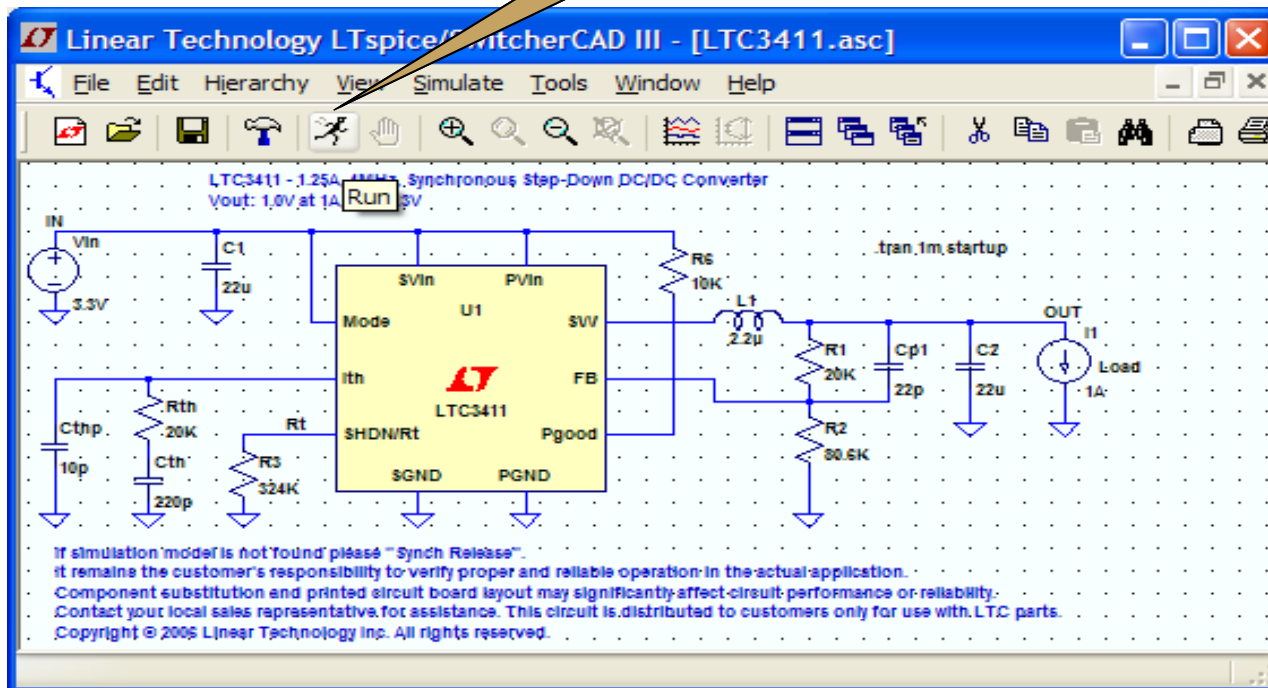


If model is not found please Sync Release
under Help menu to update LTspice

# Probing a Circuit & Waveform Viewer

◆ **Left click on any wire to plot the voltage on the waveform viewer**
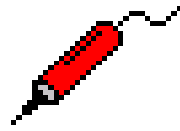
*Voltage probe cursor*

◆ **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*
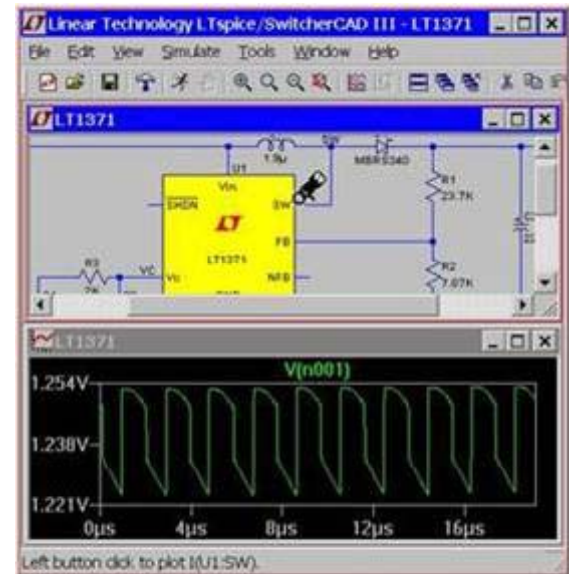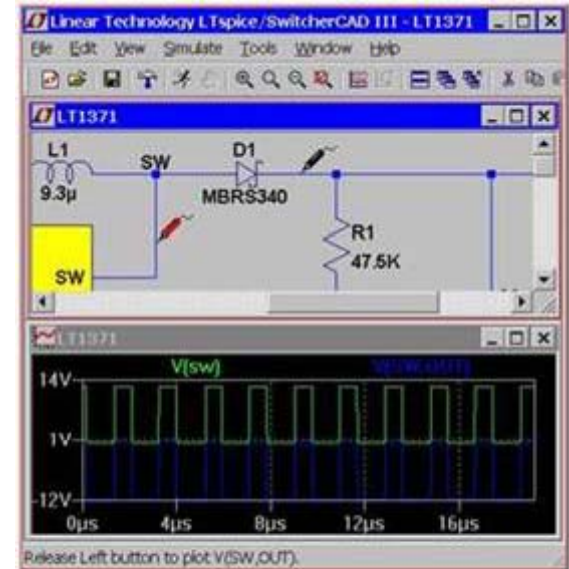
# Probing a Demo Circuit and Test Fixture

◆ 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
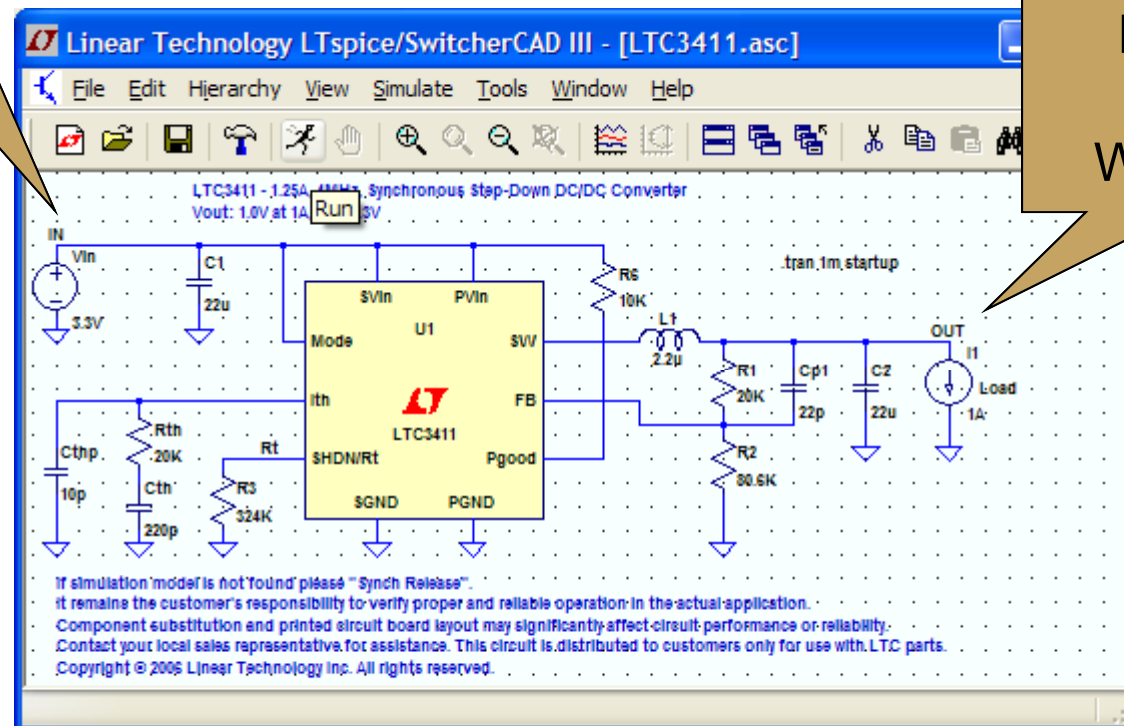
Left Click Here for Output Waveform

# 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
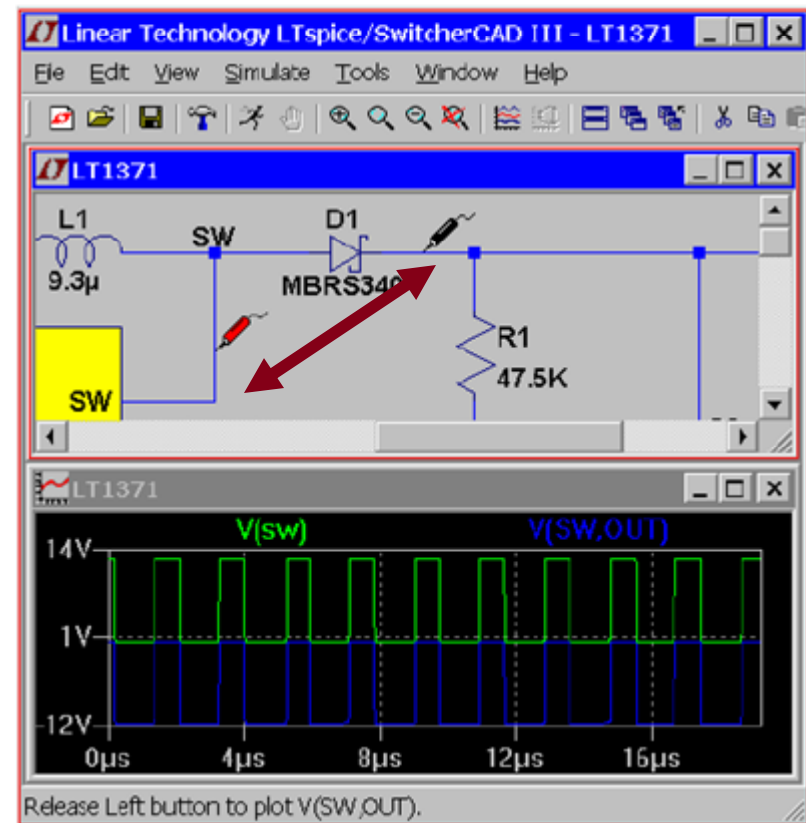
# Plot Planes

◆ 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
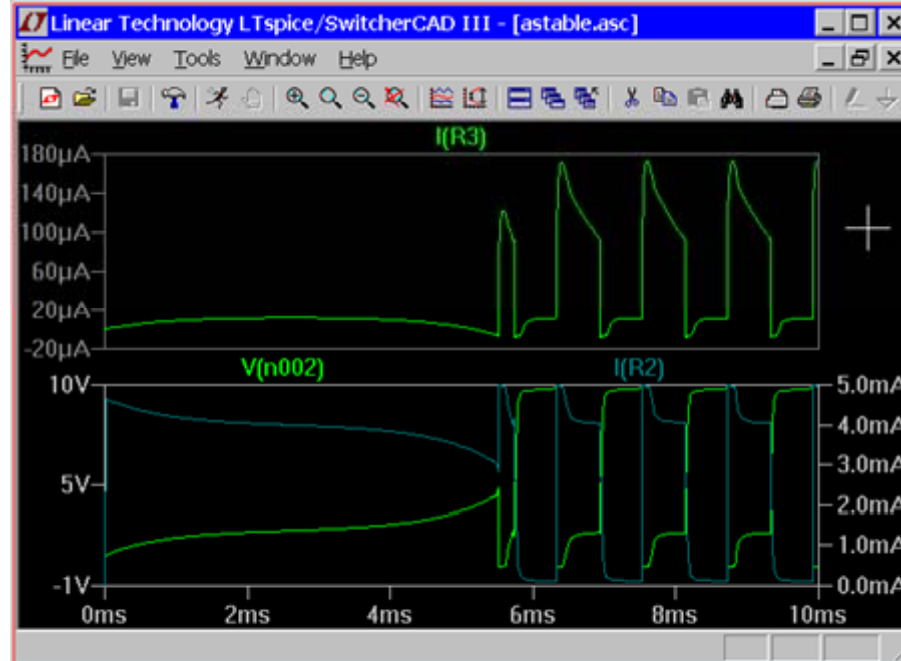
# Zooming In and Out in the Waveform Viewer

◆ **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**
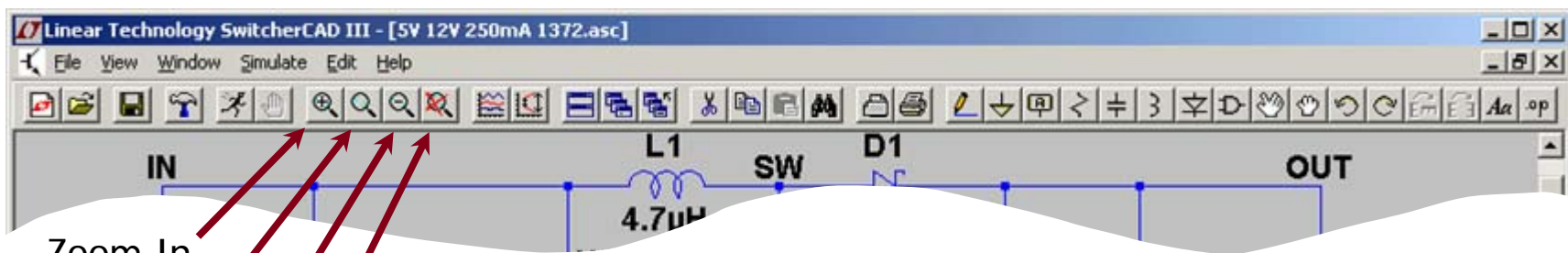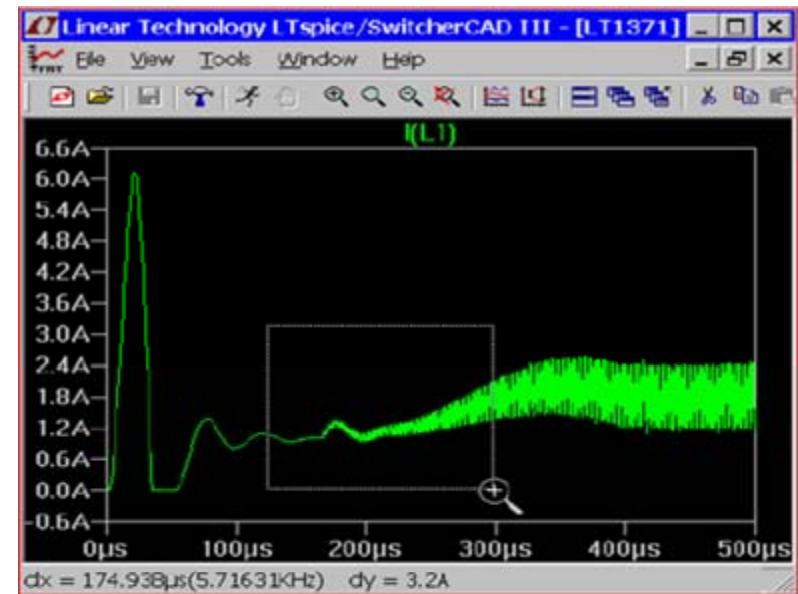


Zoom In
Pan
Zoom Out
Autoscale

# 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
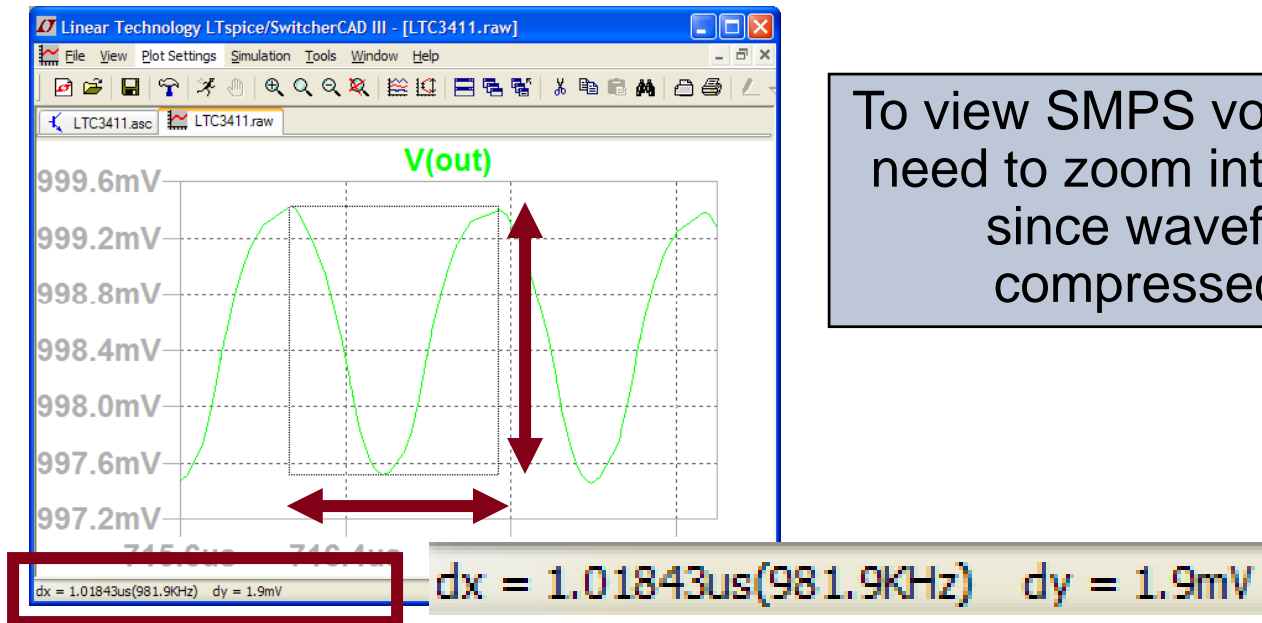


To view SMPS voltage ripple you will need to zoom into a narrow section since waveform is initially compressed to full range

dx = 1.01843us(981.9KHz)   dy = 1.9mV

# Average/RMS Current or Voltage Calculations

◆ Hold down Ctrl and left click on the **I** or **V** ***trace label*** in the waveform viewer
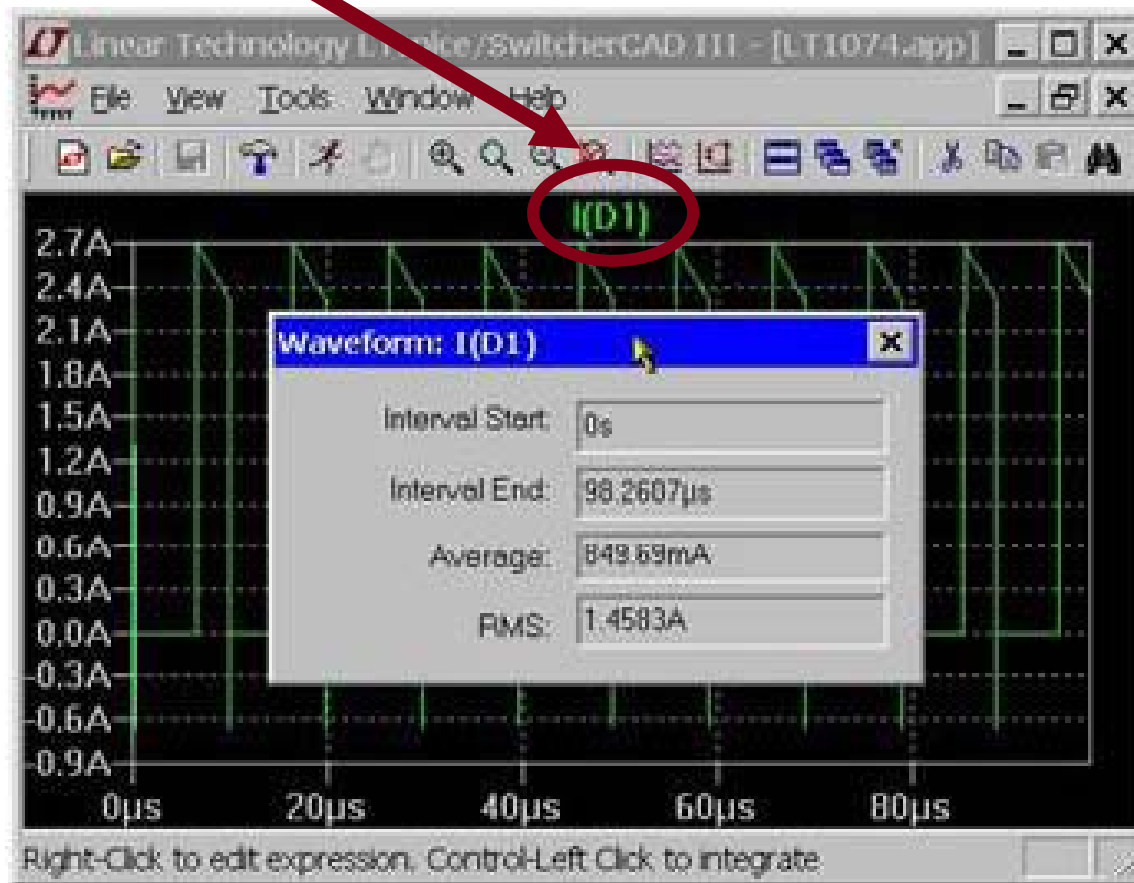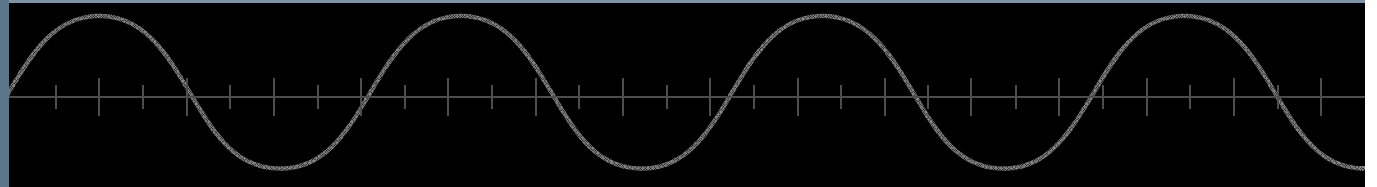
# Instantaneous & Average Power Dissipation

◆ **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
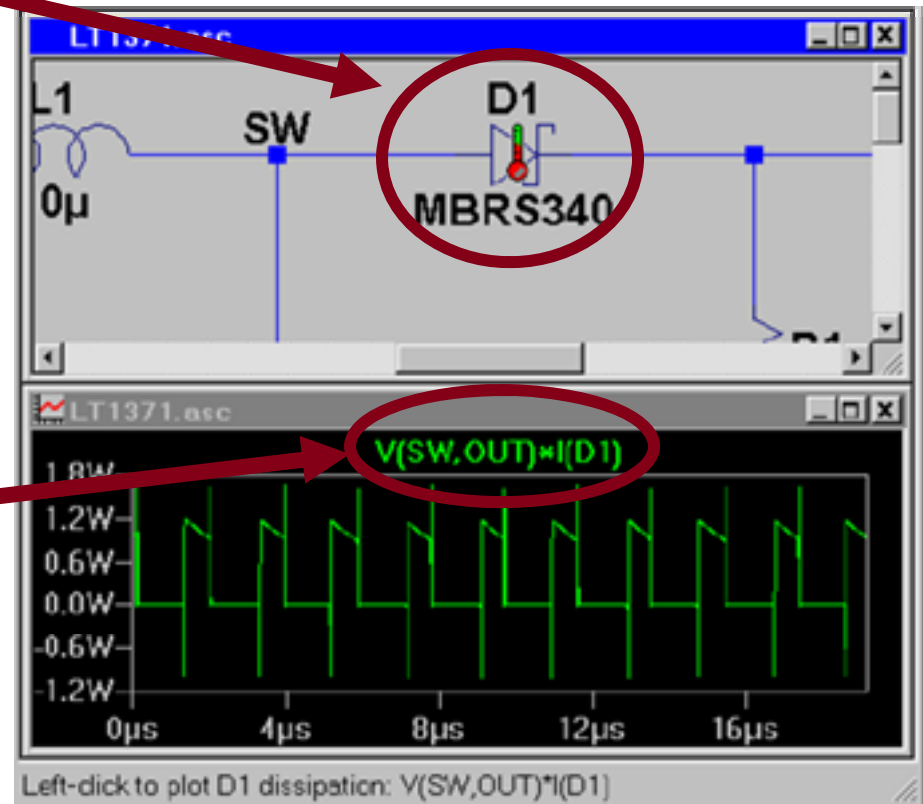
# Generating a BOM and Efficiency Report

# Bill of Materials (BOM)

- ◆ Left click on **View** menu
- ◆ Left click on **Bill of Materials**
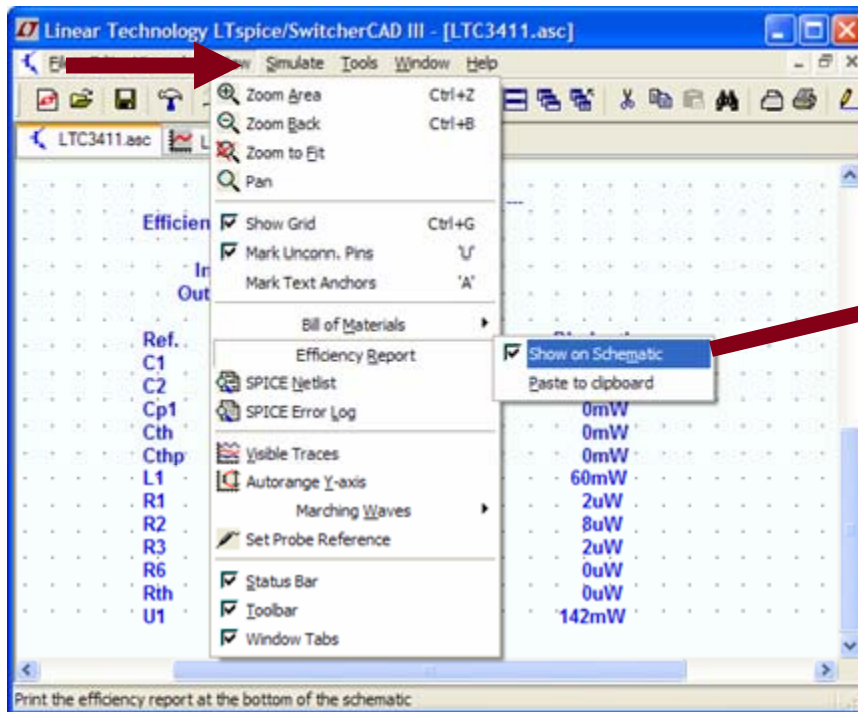
# Computing Efficiency of SMPS Circuits

- ◆ 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
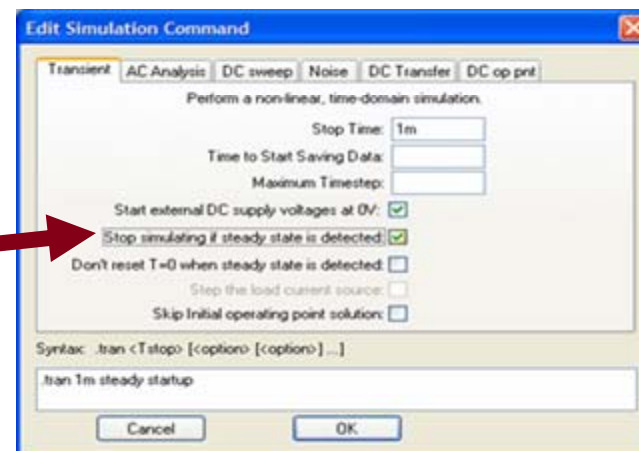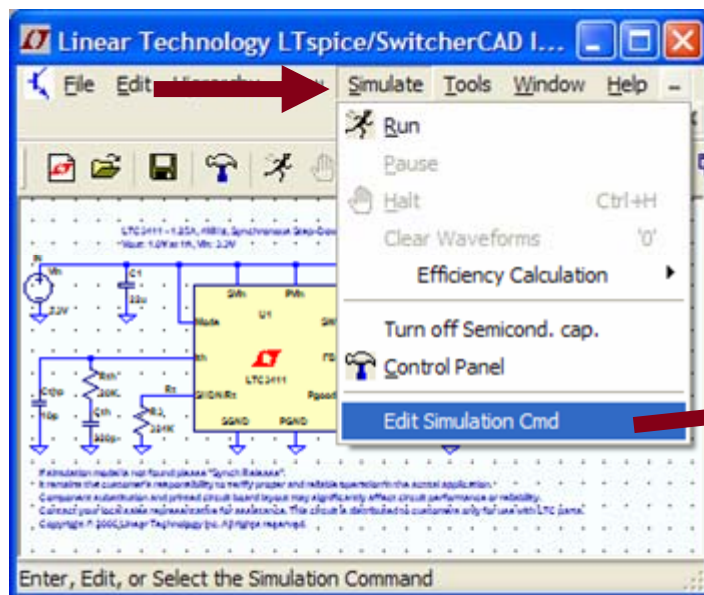
# Viewing Efficiency Report

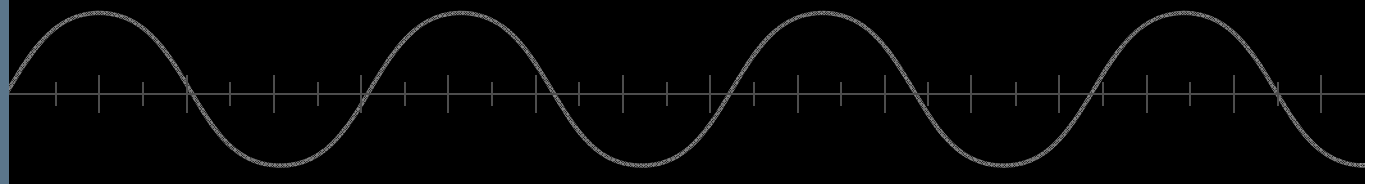◆ Left click on **Simulate** menu

◆ Left click on **Efficiency Report**

# Simulate a Transient Response in a SMPS

## Advanced Topic
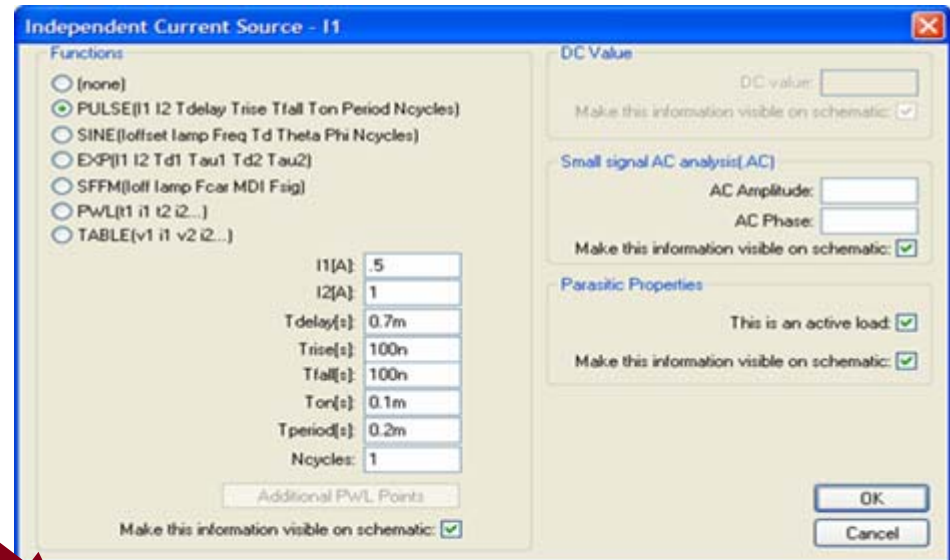
# Use a Pulsed Function as a Transient Response Load

◆ 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
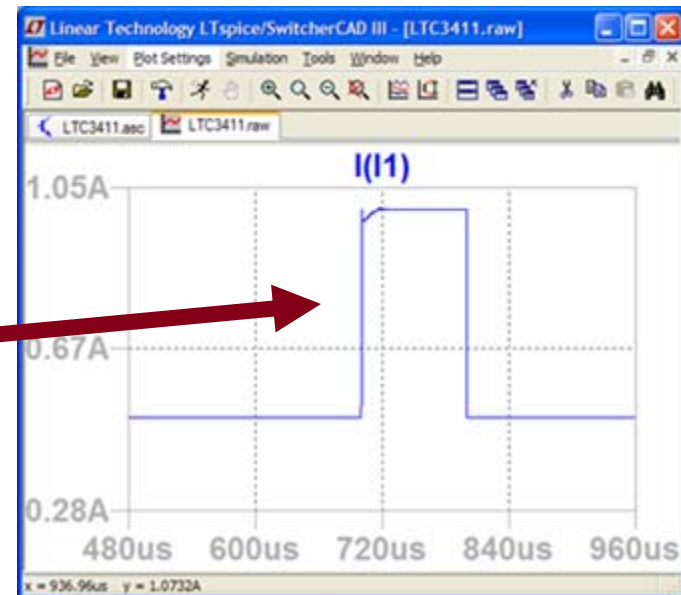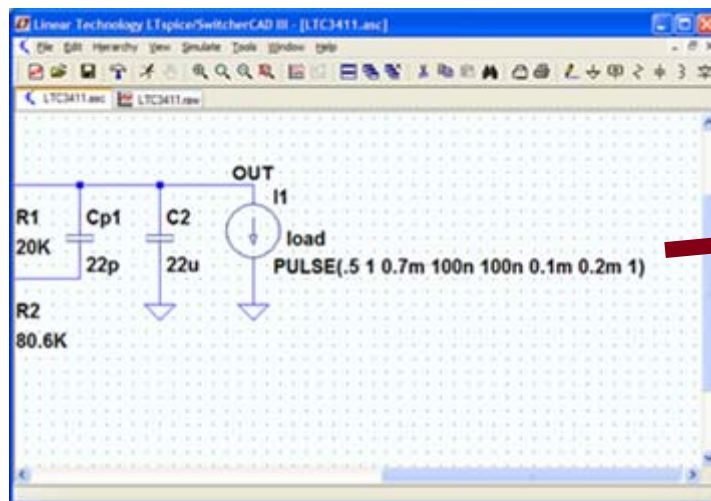
# Configuring Load as a Pulse Function

- ◆ 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

**Independent Current Source - I1**

Functions
- ○ (none)
- ⦿ PULSE(I1 I2 Tdelay Trise Tfall Ton Period Ncycles)
- ○ SINE(Ioffset Iamp Freq Td Theta Phi Ncycles)
- ○ EXP(I1 I2 Td1 Tau1 Td2 Tau2)
- ○ SFFM(Ioff Iamp Fcar MDI Fsig)
- ○ PWL(t1 i1 t2 i2...)
- ○ TABLE(v1 i1 v2 i2...)

| | |
|---|---|
| I1[A]: | .5 |
| I2[A]: | 1 |
| Tdelay[s]: | 0.7m |
| Trise[s]: | 100n |
| Tfall[s]: | 100n |
| Ton[s]: | 0.1m |
| Tperiod[s]: | 0.2m |
| Ncycles: | 1 |

Additional PWL Points

Make this information visible on schematic: ☑

DC Value
DC value:
Make this information visible on schematic: ☑

Small signal AC analysis(.AC)
AC Amplitude:
AC Phase:
Make this information visible on schematic: ☑

Parasitic Properties
This is an active load: ☑
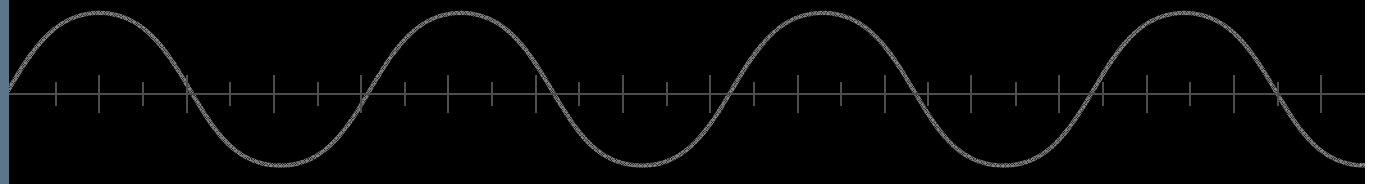Make this information visible on schematic: ☑

OK
Cancel

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

**LINEAR** TECHNOLOGY

# 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

Transformer with two windings

K1 L1 L2 1

L1
100u

L2
900u
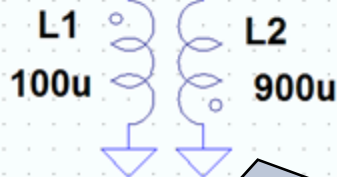
K statement coupling the windings

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

**LINEAR** TECHNOLOGY

# Additional Information and Support
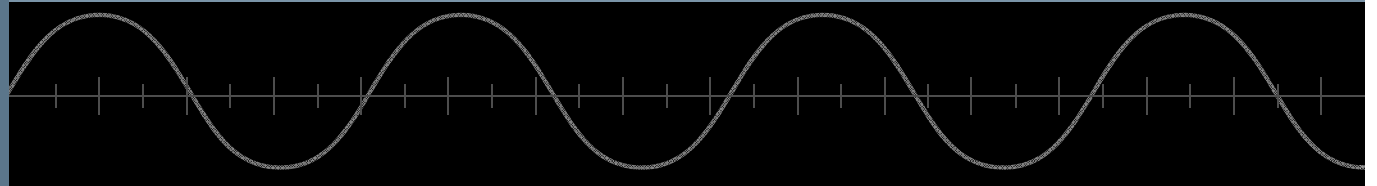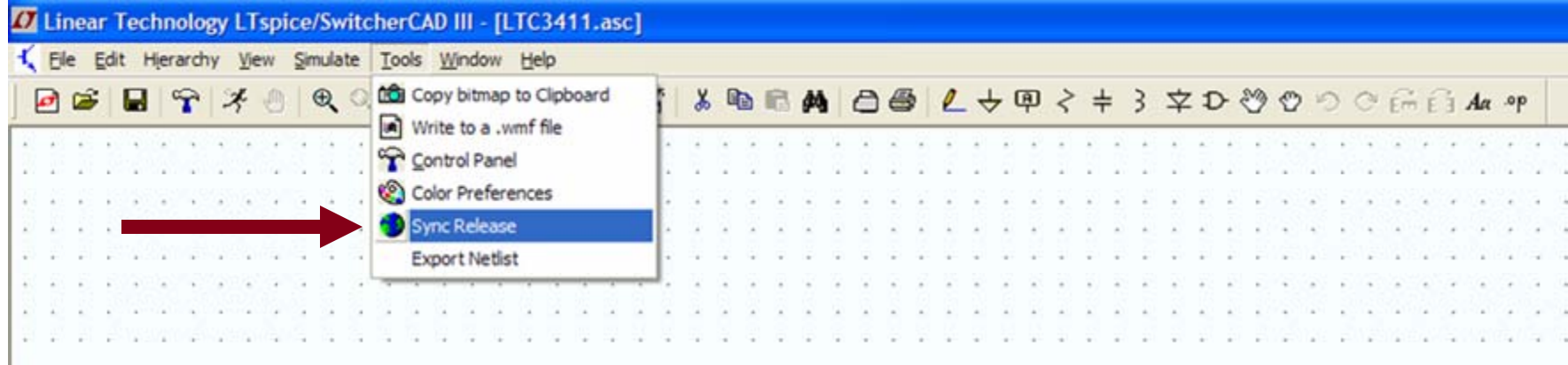
LINEAR
TECHNOLOGY

# Reminder to Periodically Sync Release

◆ **Update your release of LTspice to get the latest**

- ◆ Software updates

- ◆ Models and examples

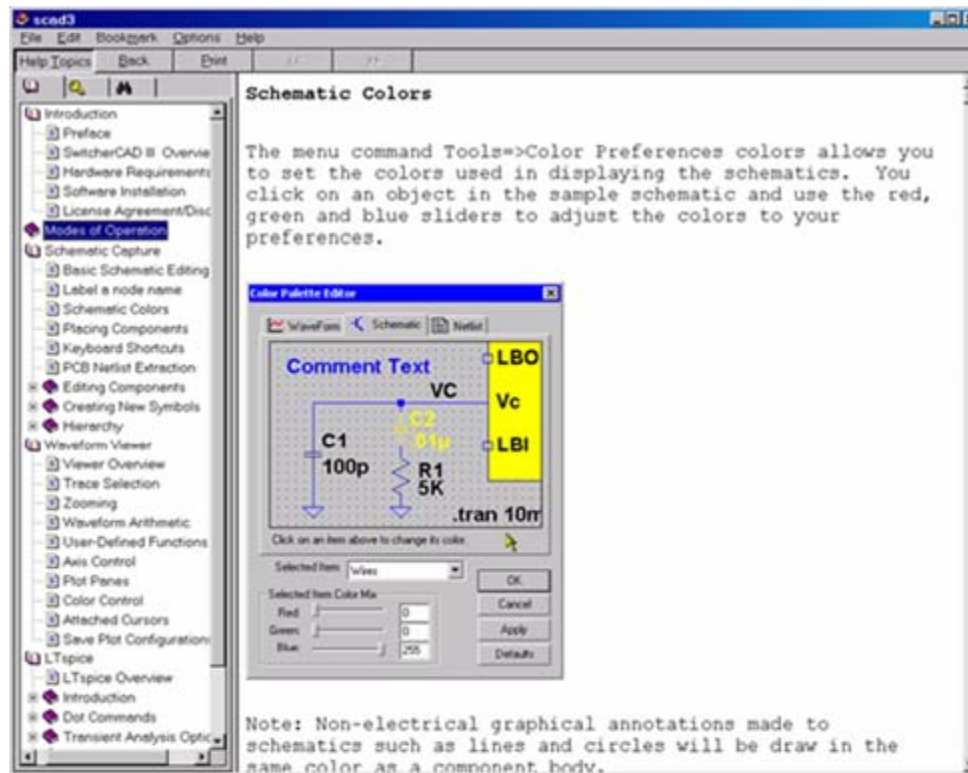> Sign up for *Linear Insider* via MyLinear (www.linear.com) for email news and updates



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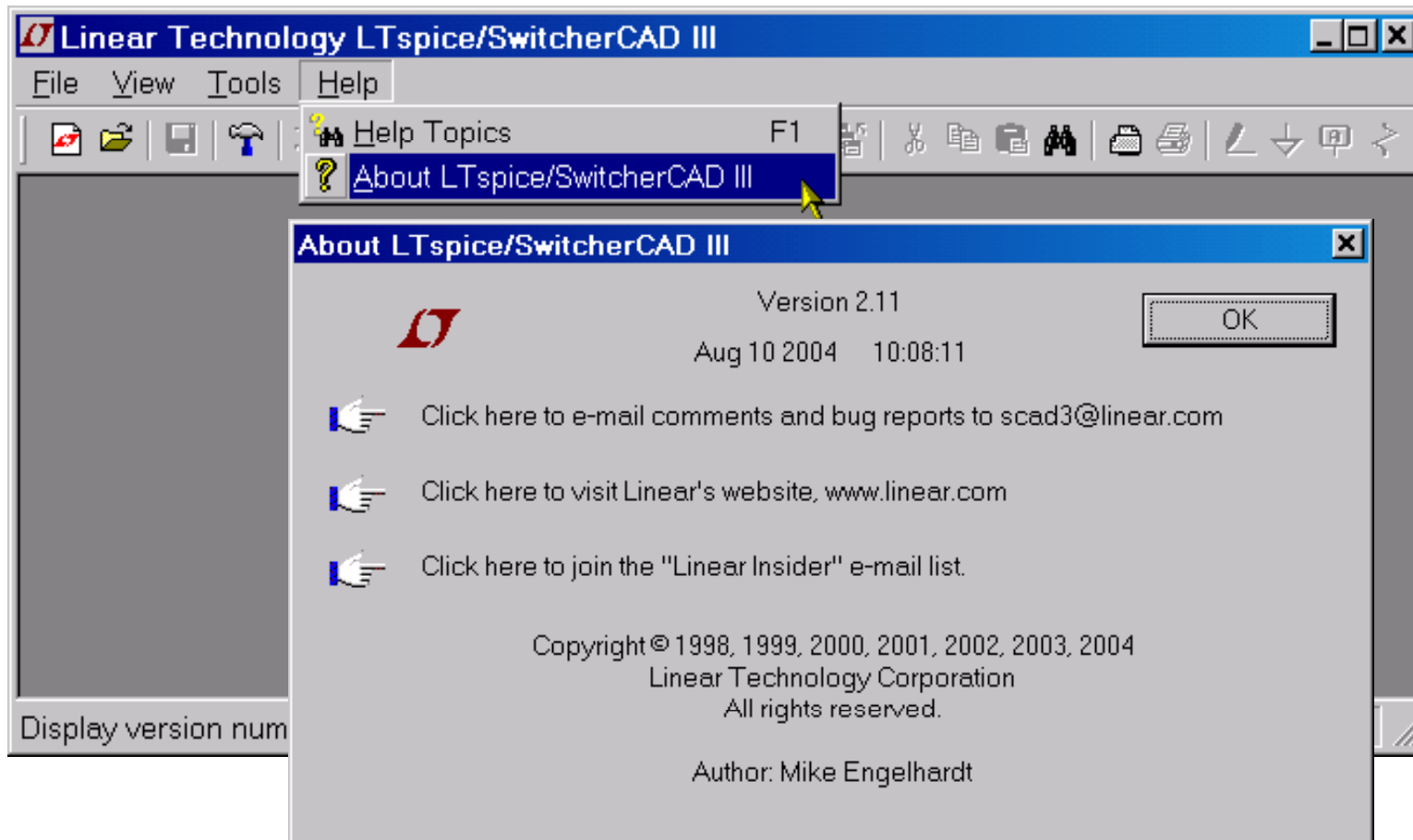
# Built-in Help System

◆ 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