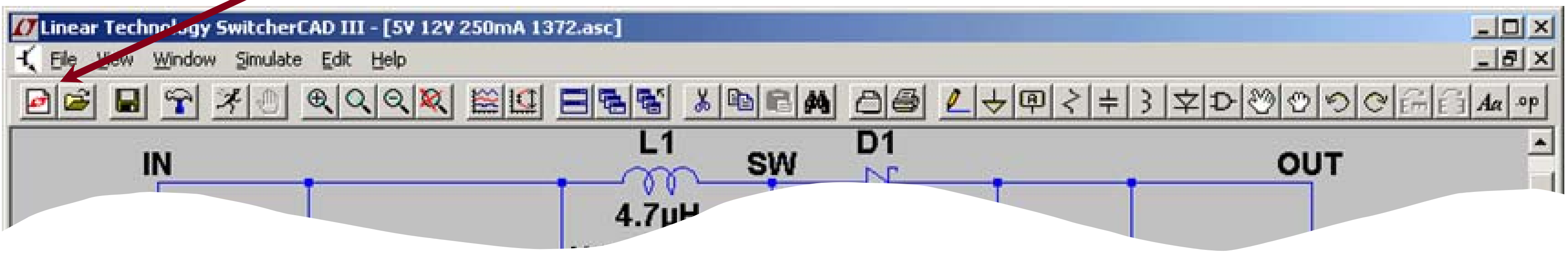
Getting Started

LINEAR TECHNOLOGY

Draft a Design Using the Schematic Editor

Start with a New Schematic

New Schematic

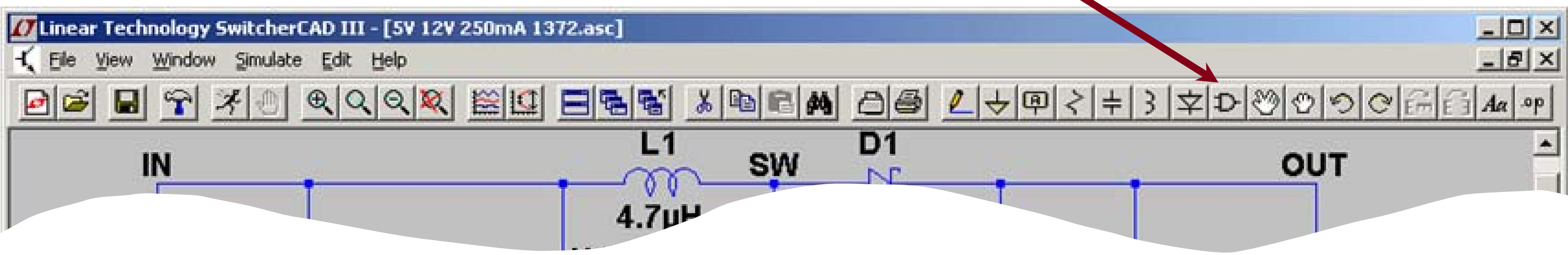


- Left click on the **New Schematic** symbol in the Schematic Editor Toolbar

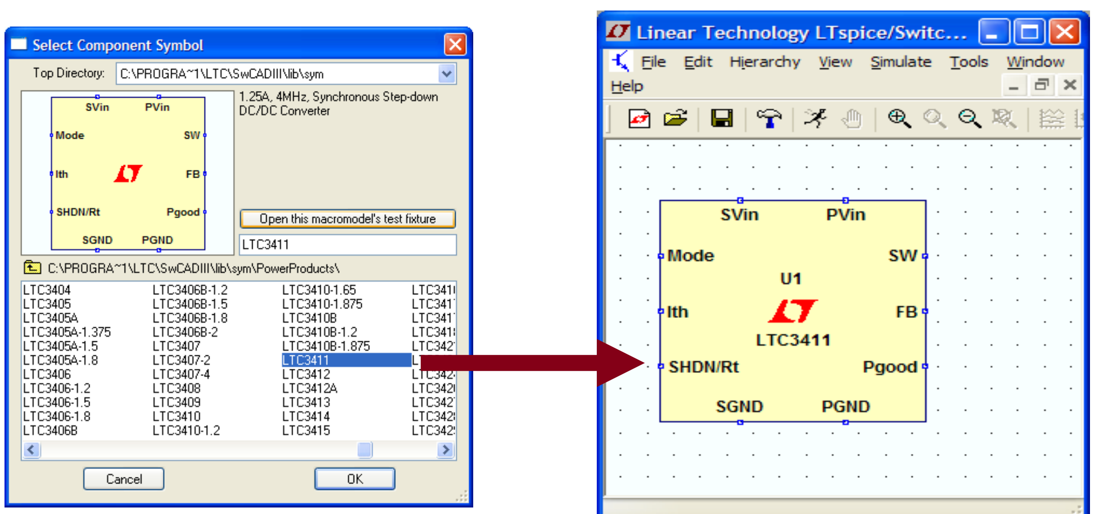
LTspice is also a great schematic capture

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**

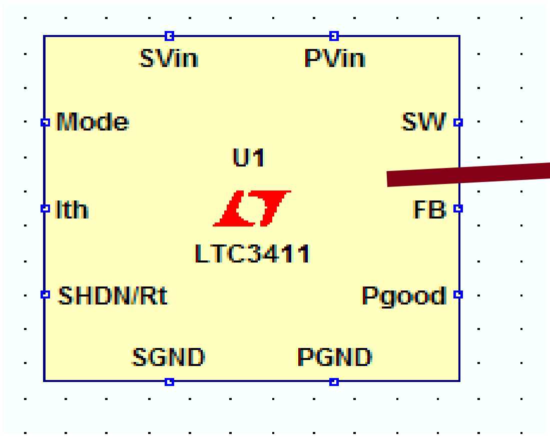


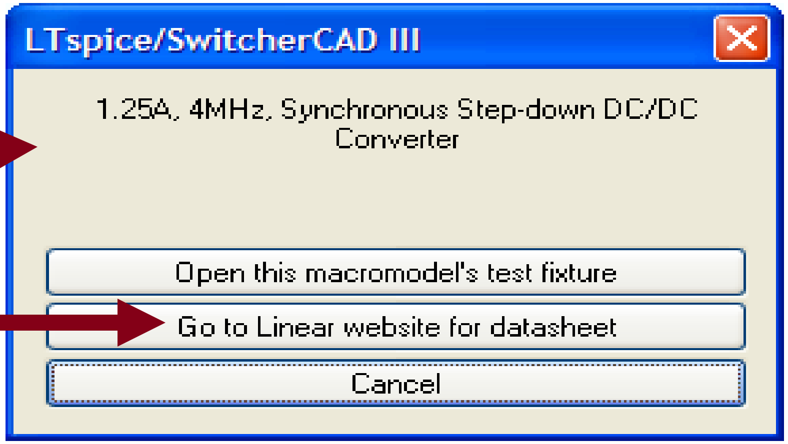
LINEAR TECHNOLOGY

Getting the Latest Datasheet

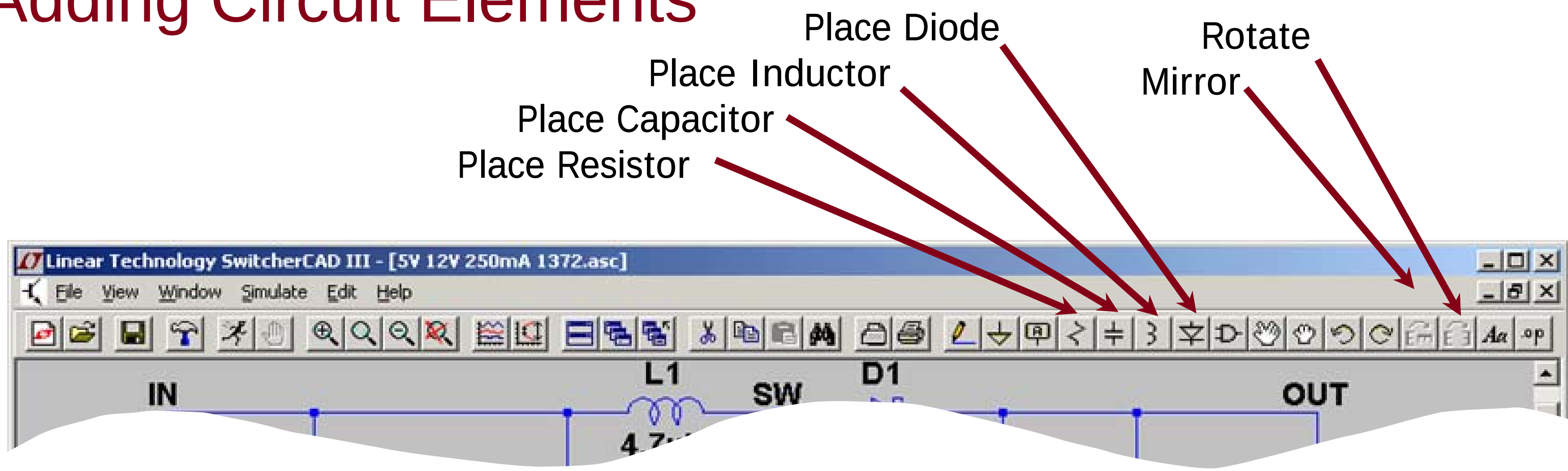
- Use the macromodel's shortcuts to download the *Datasheet* as a reference for your design
  - Hold Ctrl key and right click (*Ctrl – right click*) over the LT macromodel's symbol
  - Left click on Go to Linear website for datasheet on the dialog box that appears

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





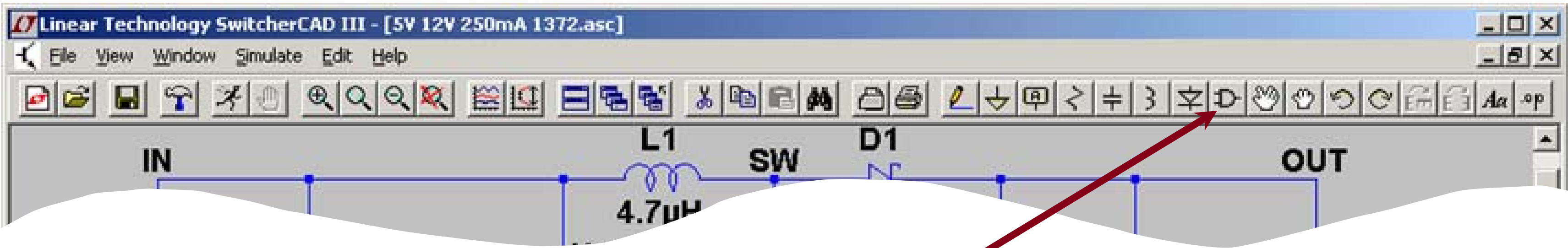
Adding Circuit Elements



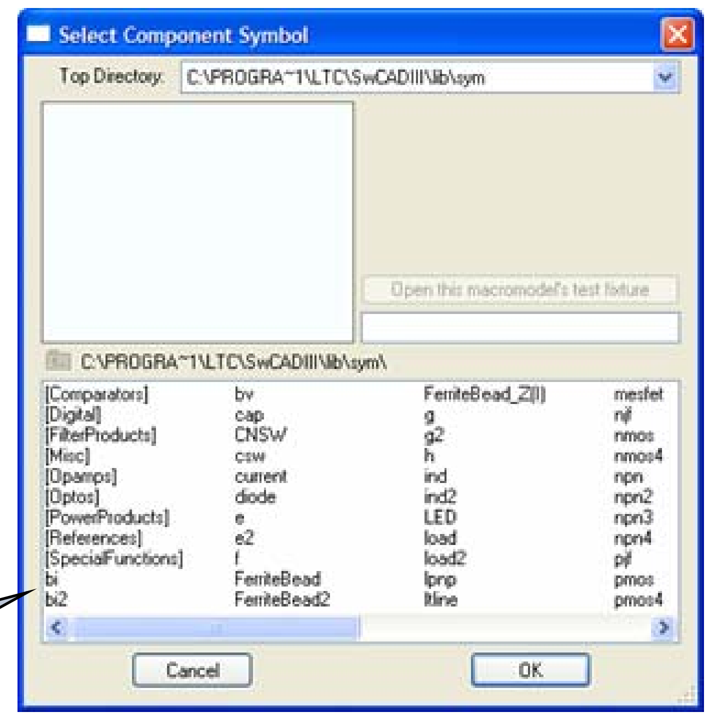
- Left click on the desired *component* in the Schematic Editor Toolbar
- Left click on **Rotate** or **Mirror** to adjust orientation
  - Alternate you can also use Ctrl – R and Ctrl – M key shortcuts
- Move the mouse to the position you want to place it
- Left click to place it

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

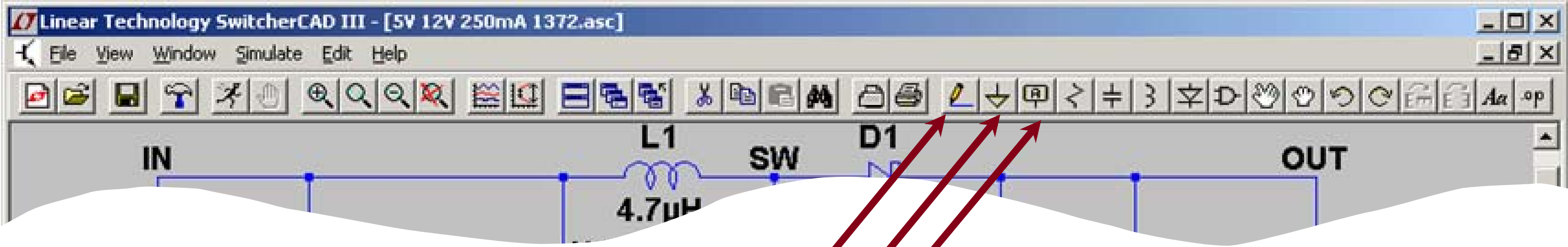


Additional Circuit Elements Like Sources and Loads

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

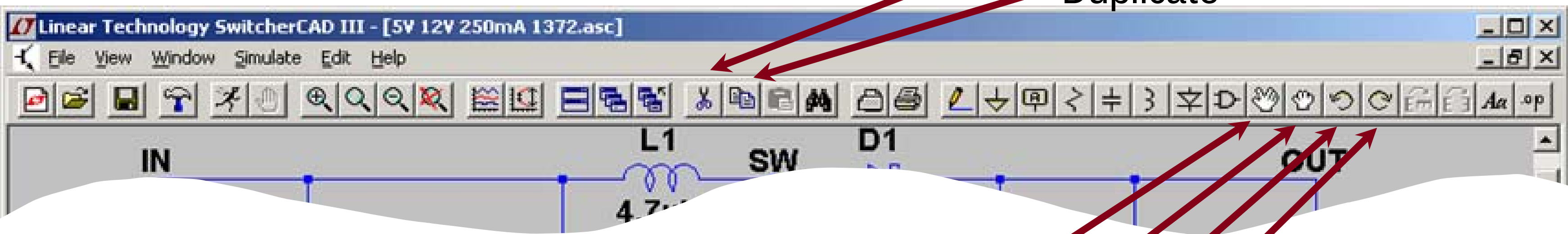


Draw Wire
Place Ground
Label Node

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

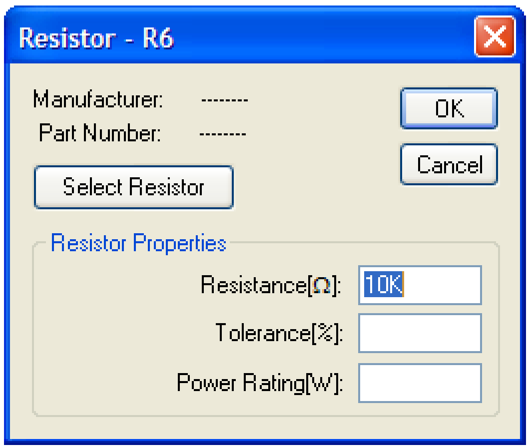


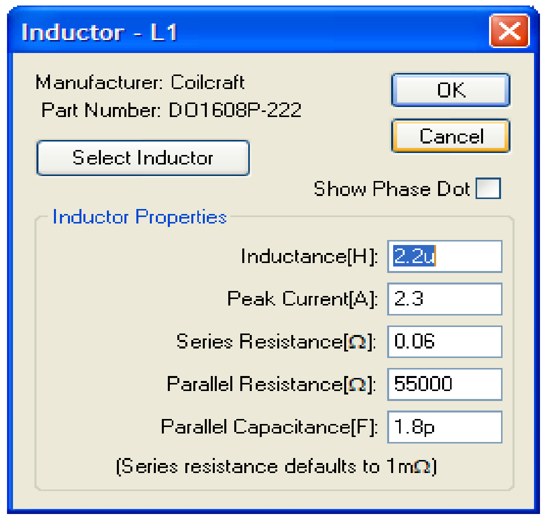
- Left click on the desired editing option
- Left click on the circuit element

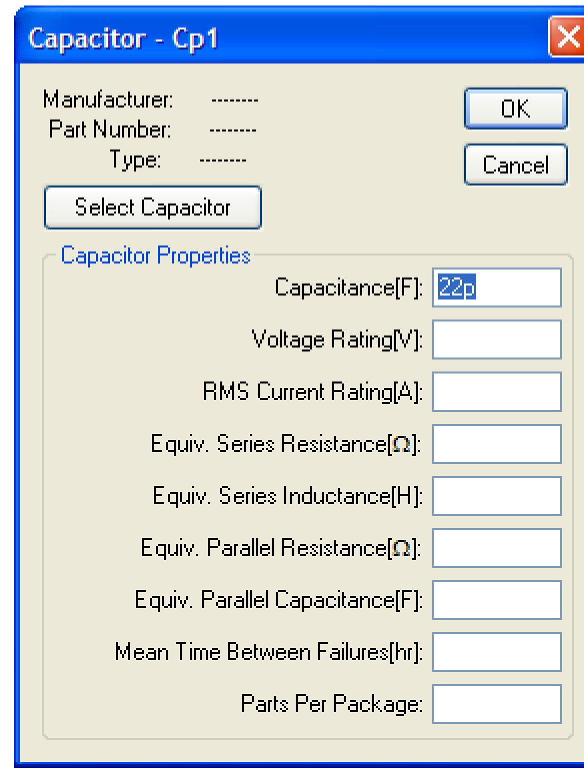
To organize your layout, use the ***Drag*** option to move circuit elements around and to adjust lines between terminals

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

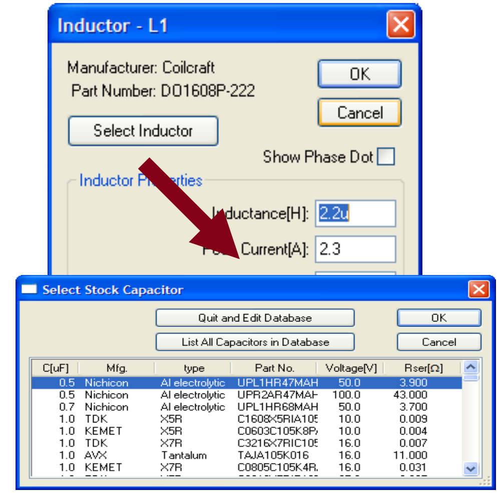
- K = k = kilo = 10^3
- MEG = meg = 10^6
- G = g = giga = 10^9
- T = t = terra = 10^{12}
- M = m = milli = 10-3
- U = u = micro = 10-6
- N = n = nano = 10-9
- P = p = pico = 10-12
- F = f = femto = 10-15

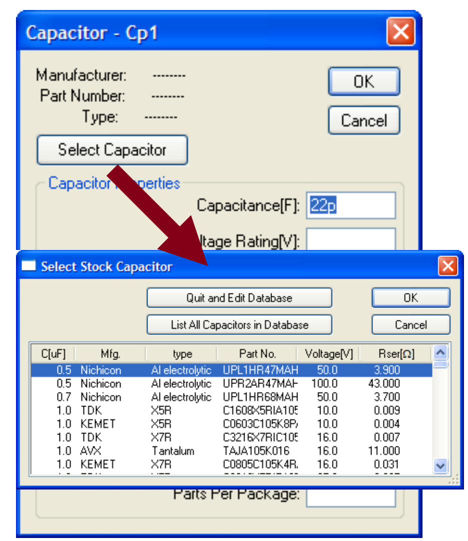
Hints

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

Circuit Elements Database

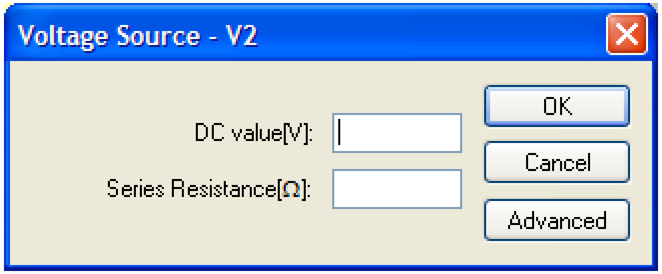
- Some components have an available database of manufacturers' attributes
  - Resistors, capacitors, inductors, diodes,
  - Bipolar transistors, MOSFET transistors, JFET transistors
  - Independent voltage and current sources
- To configure a component to a manufacture's attributes
  - Right click on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

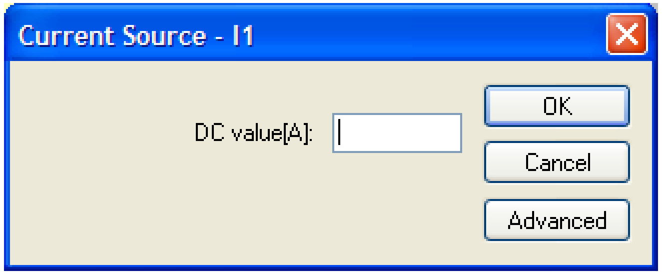




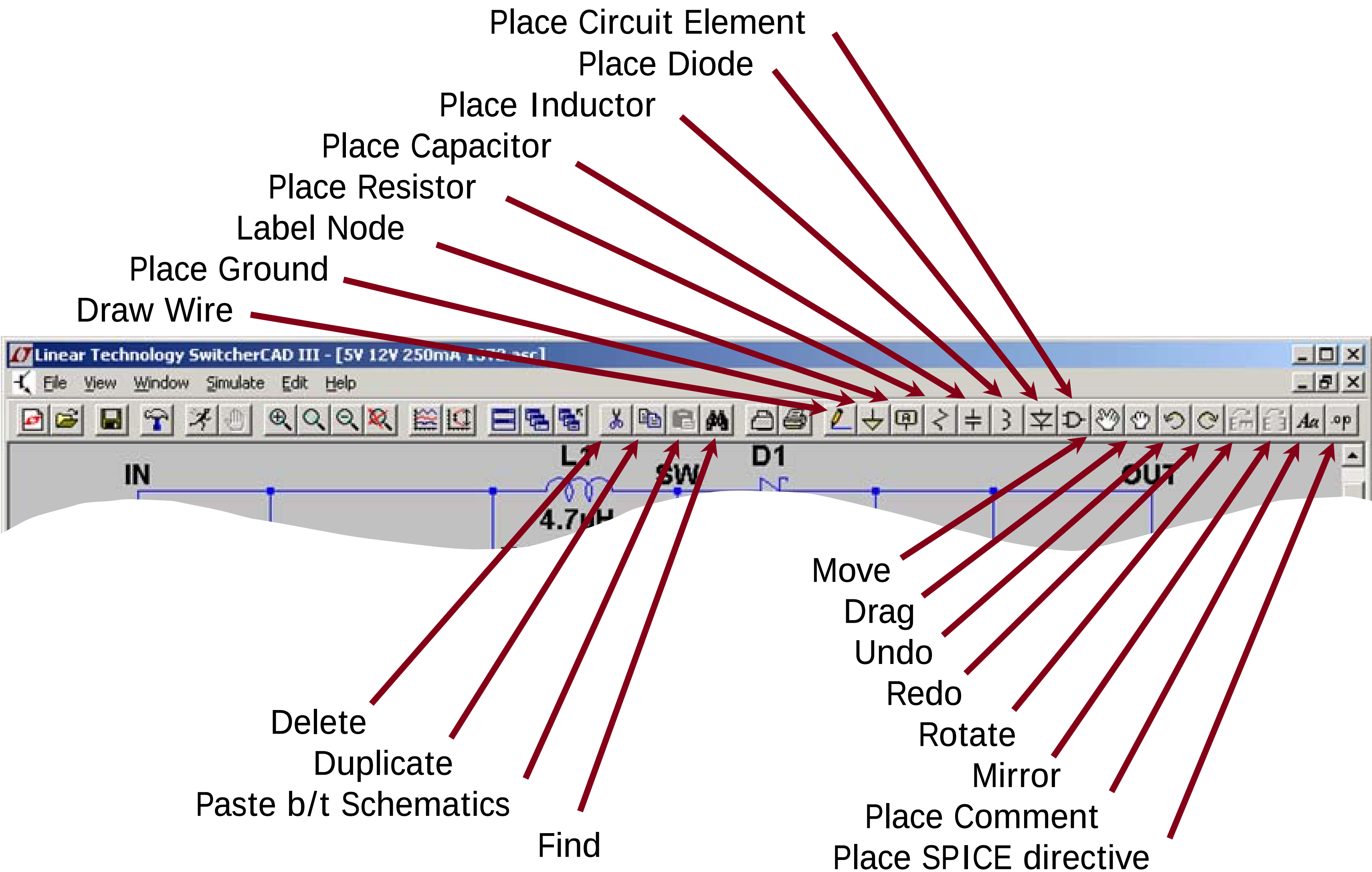
Editing Voltage Sources and Loads

- 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

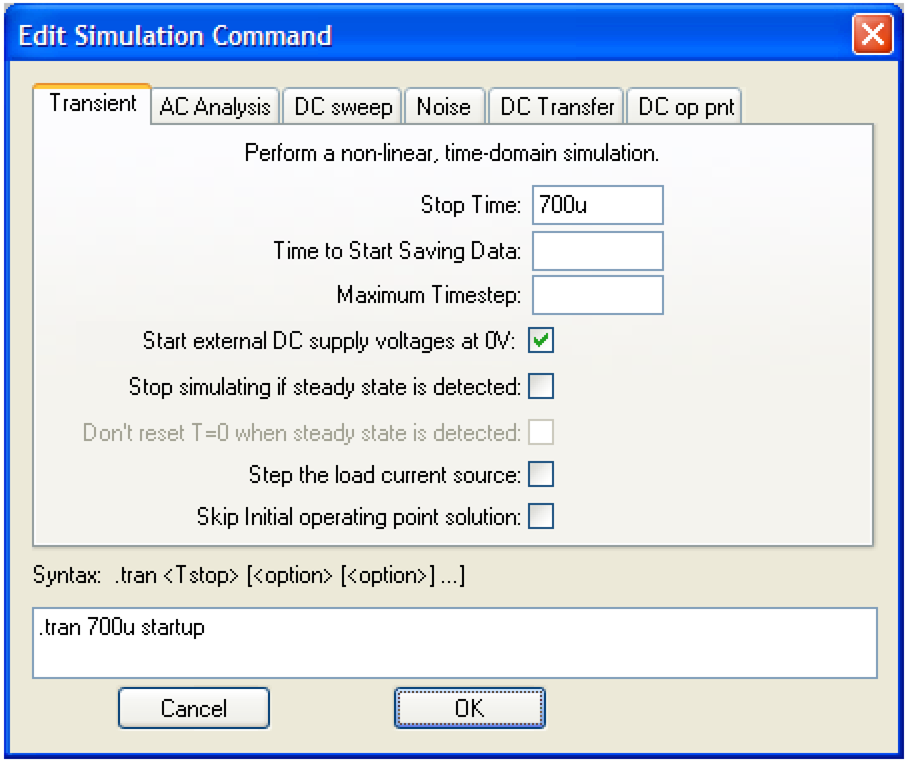
- To run a simulation, specify the type of analysis to be performed
- There are six different types of analyses:
  - Transient analysis
  - Small signal AC
  - DC sweep
  - Noise
  - DC transfer function
  - DC operating point
- Simulation commands are placed on the schematic as text
  - Called dot commands

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

Editing Simulation Commands

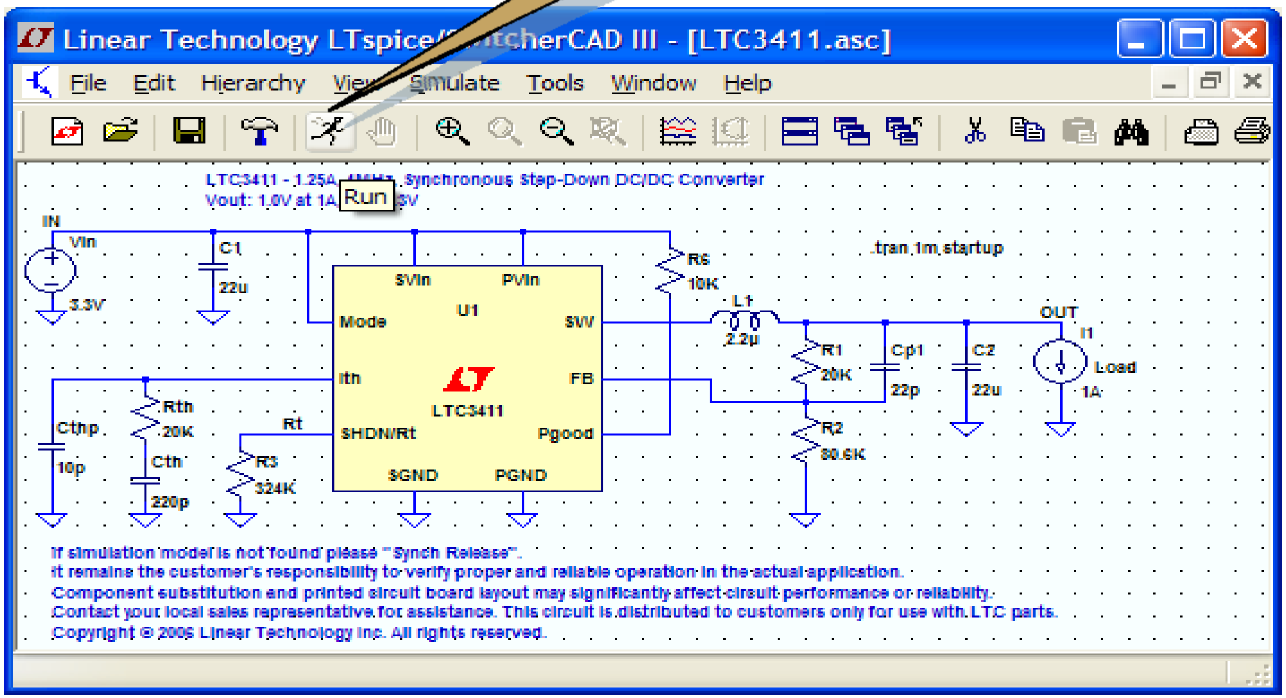
- Left click on **Simulation** menu
- Left click on **Edit Simulation Cmd**
- As a starting point in a simulation
  - Left click on **Transient** tab
  - Enter a ***Stop Time***
    - You may need to adjust this again later
- Select **OK**

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



Running a Circuit

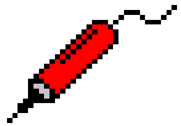
Run



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

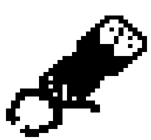
Probing a Circuit & Waveform Viewer

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

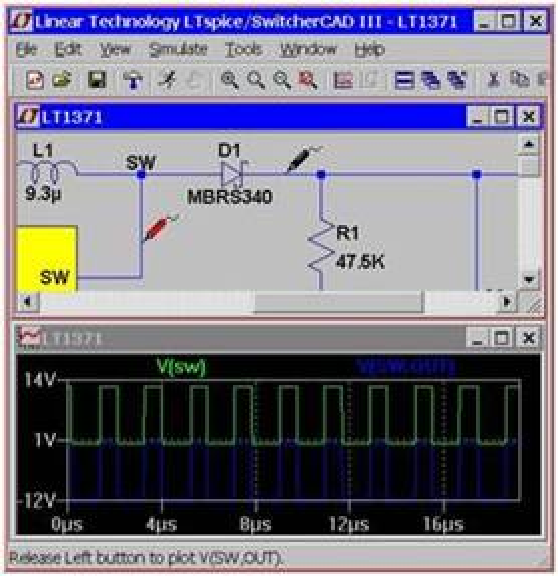


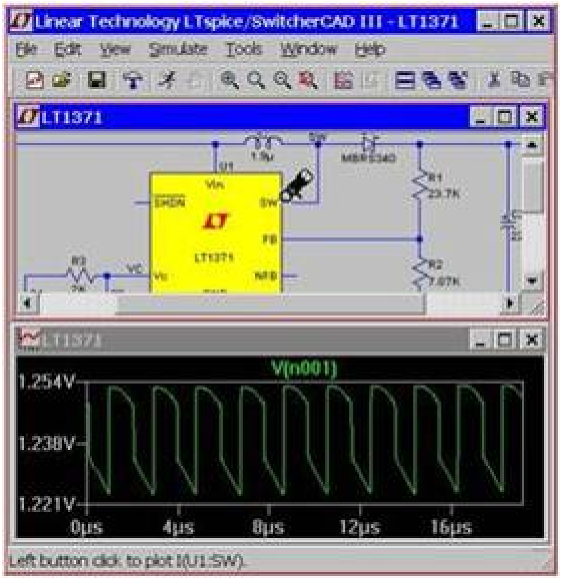
Voltage probe cursor

- 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



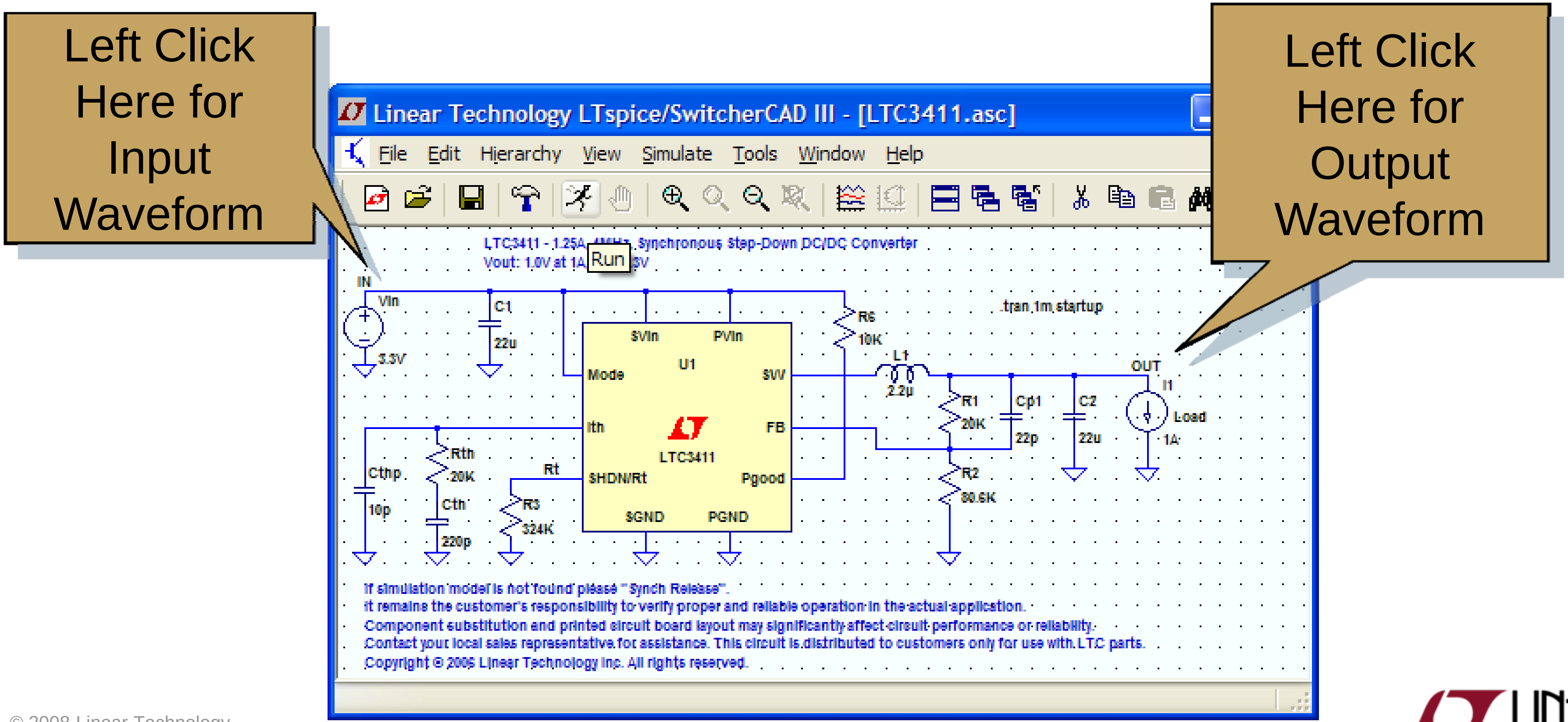
Current probe cursor





Probing a Demo Circuit and Test Fixture

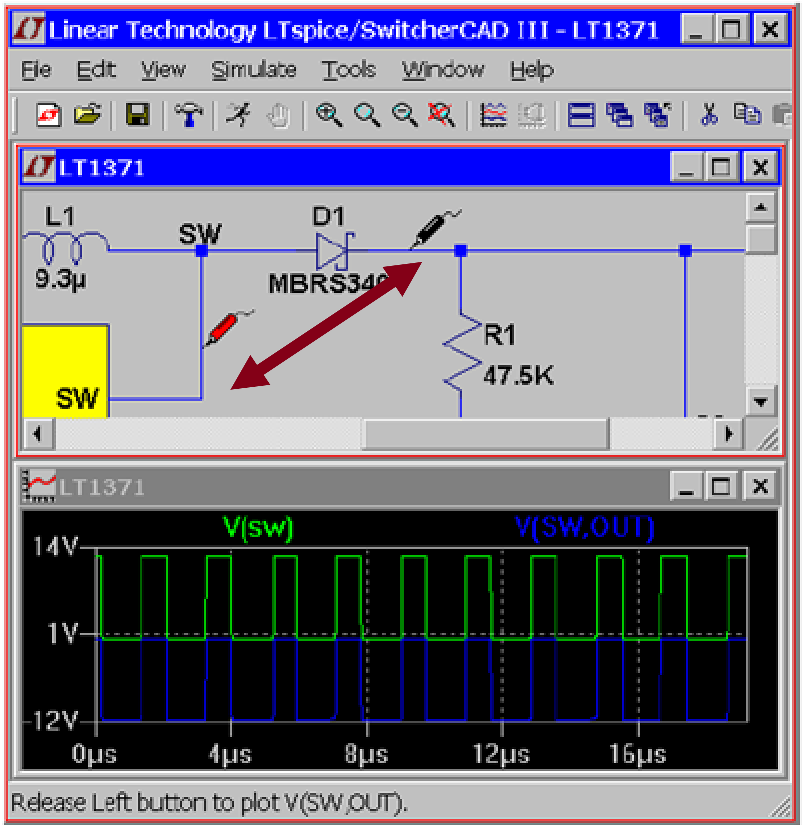
- Demo Circuits and Test Fixtures have INs and OUTs clearly labeled to help you quickly select them
- To view the waveform left click on IN and OUT



Voltage Differences Across Nodes

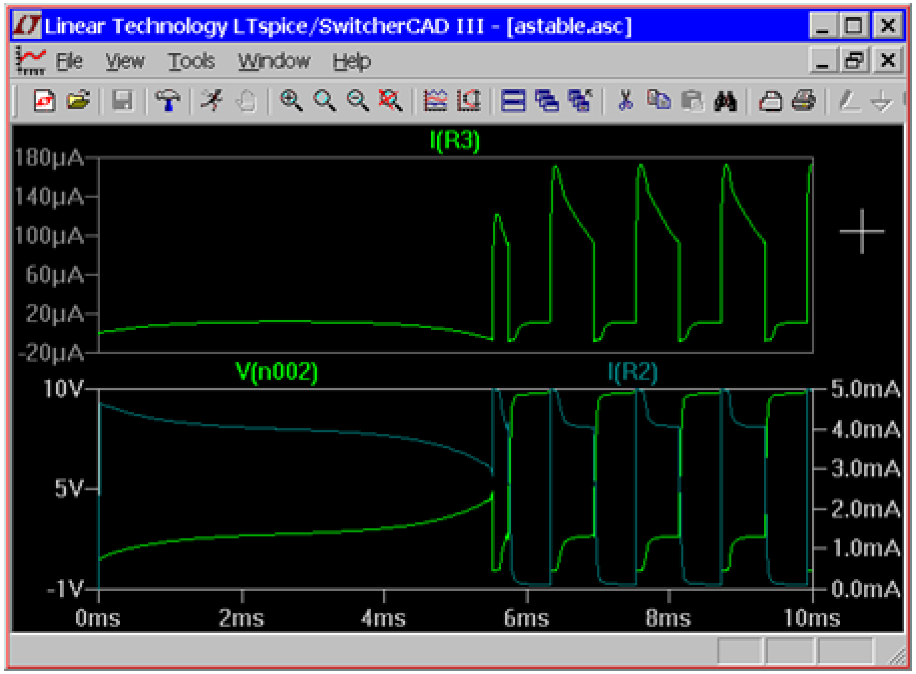
- Left click and hold on one node and drag the mouse to another node
  - Red voltage probe at the first node
  - Black probe on the second

Differential voltages are displayed in the waveform viewer



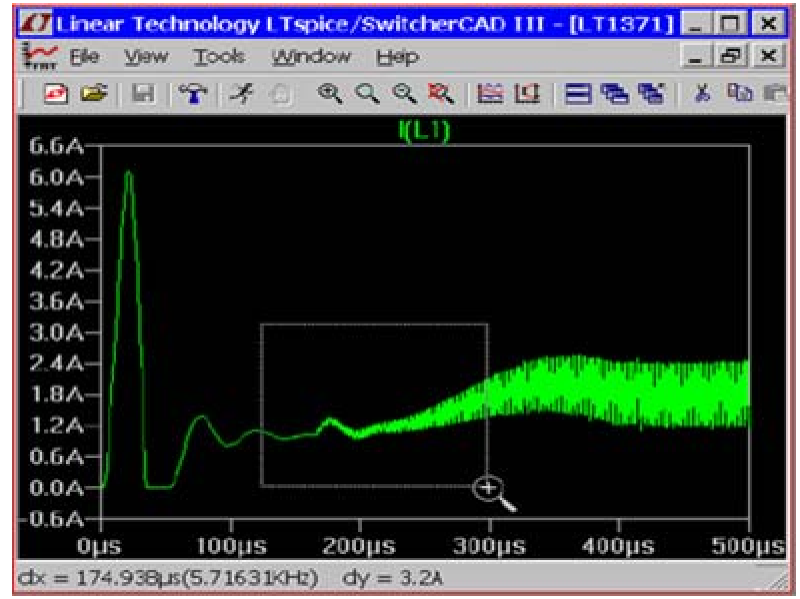
Plot Planes

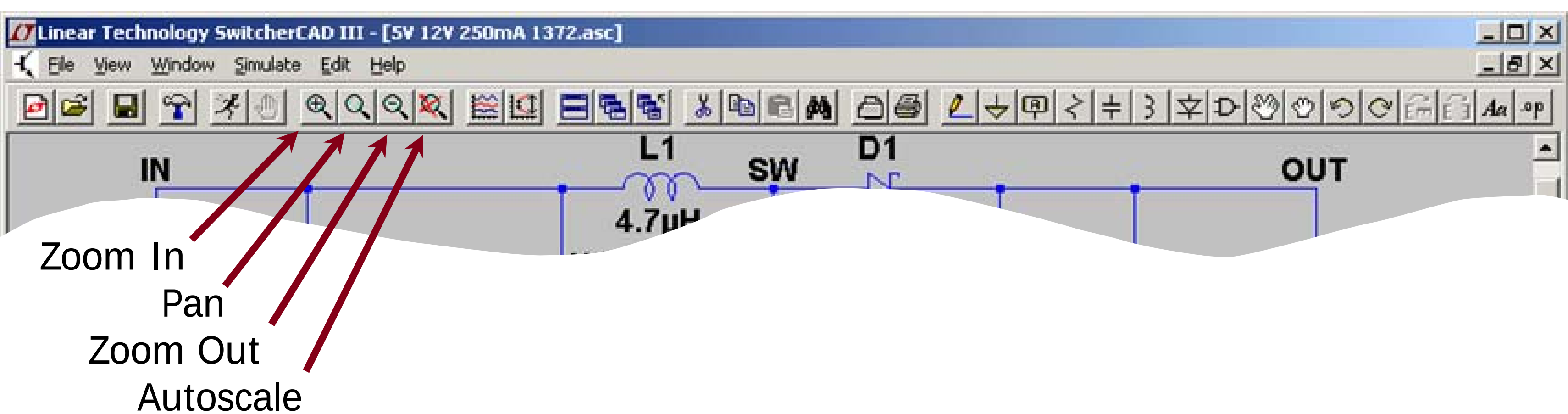
- Multiple plot panes can be displayed on one window to allow better separation between traces permitting different traces to be independently autoscaled
  - Right click in the waveform pane
  - Select Add Plot Pane
  - Left click and hold to drag a label to a new plot pane



Zooming In and Out in the Waveform Viewer

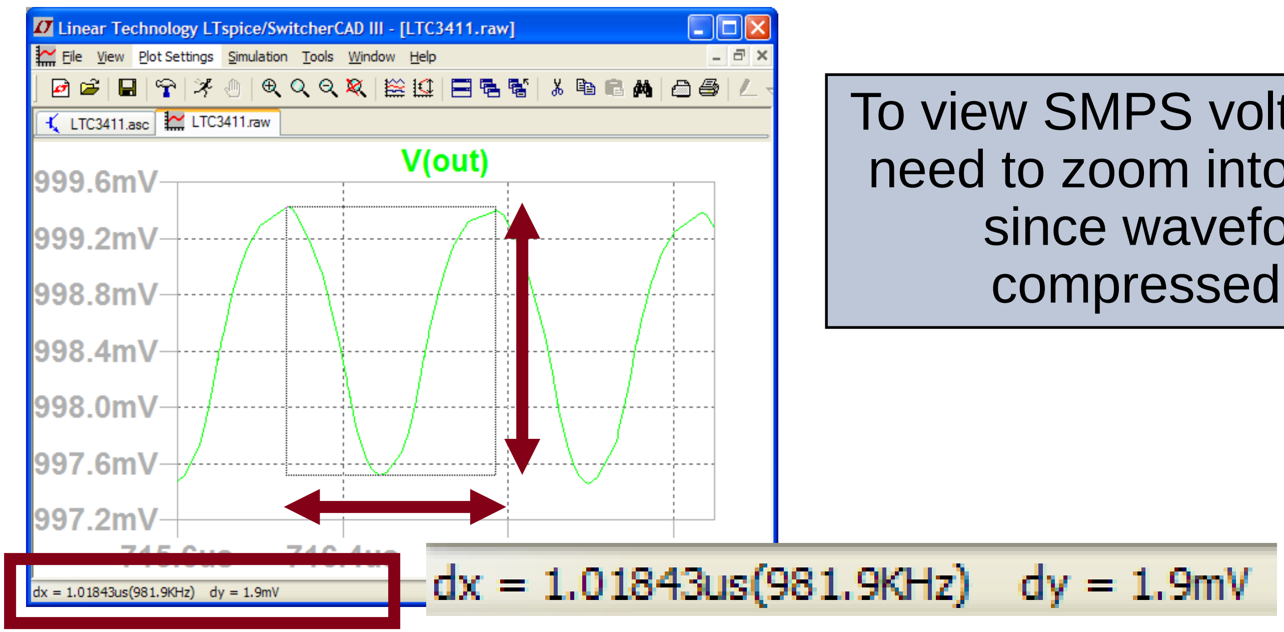
- To zoom in
  - Left click and hold as you drag a box about the region you wish to zoom in then release
- To zoom out
  - Right click and select **Zoom to Fit** or **Zoom Back**





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

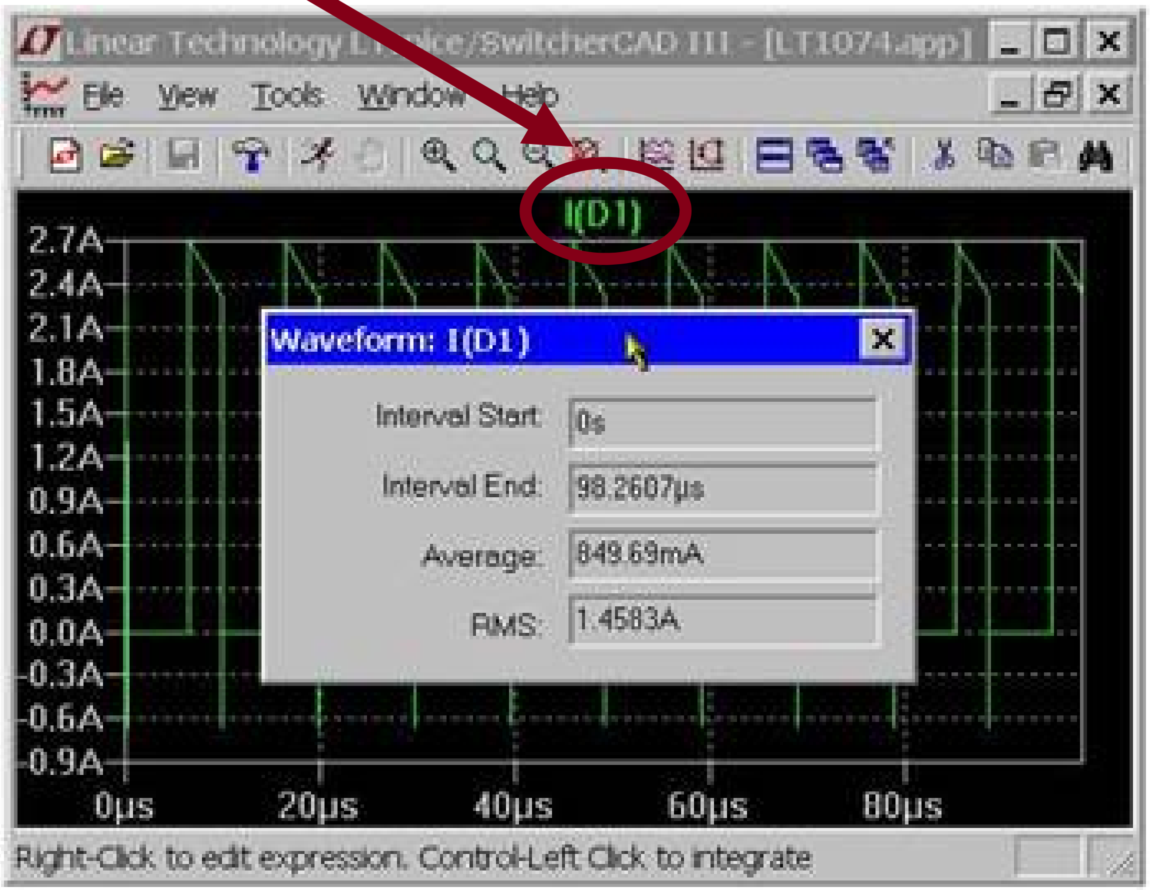
- Drag a box about the region you wish to measure (peak to peak over a period)
  - Left click and *hold* to drag a box over the portion of interest
- View the lower left hand side of the screen
  - To avoid resizing, shrink your box before you let go of the left mouse click or use the Undo command in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

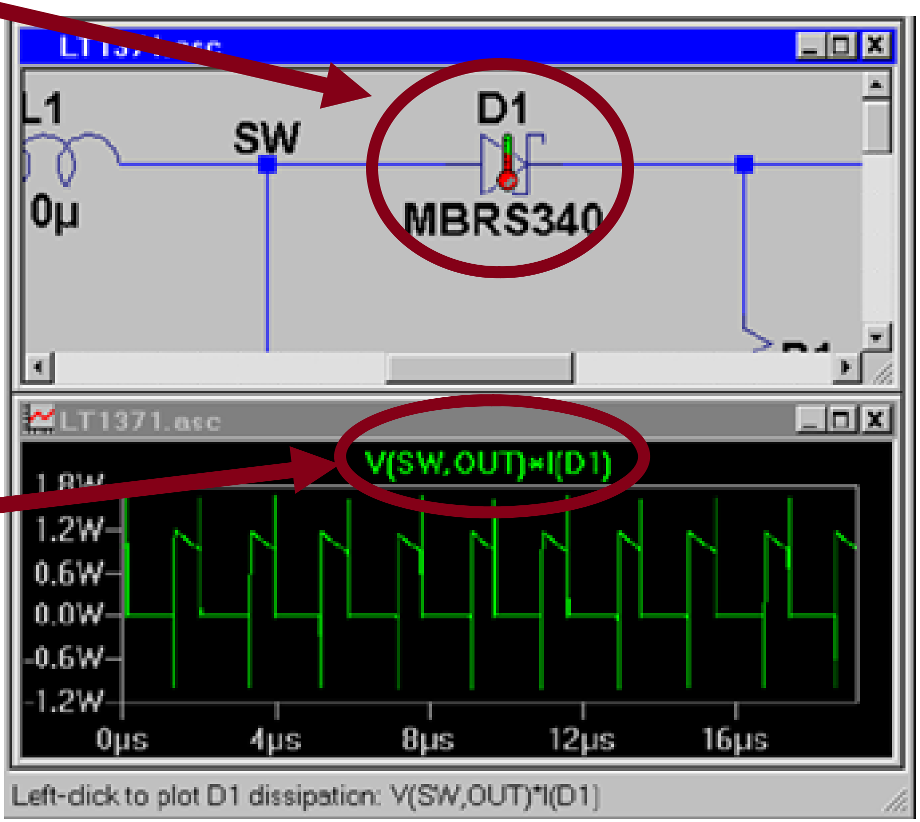
Average/RMS Current or Voltage Calculations

- 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



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