LTspice IV Basic Lab Class & Getting Started Guide

Presented by:

Mats Hellberg

Linear Technology FAE

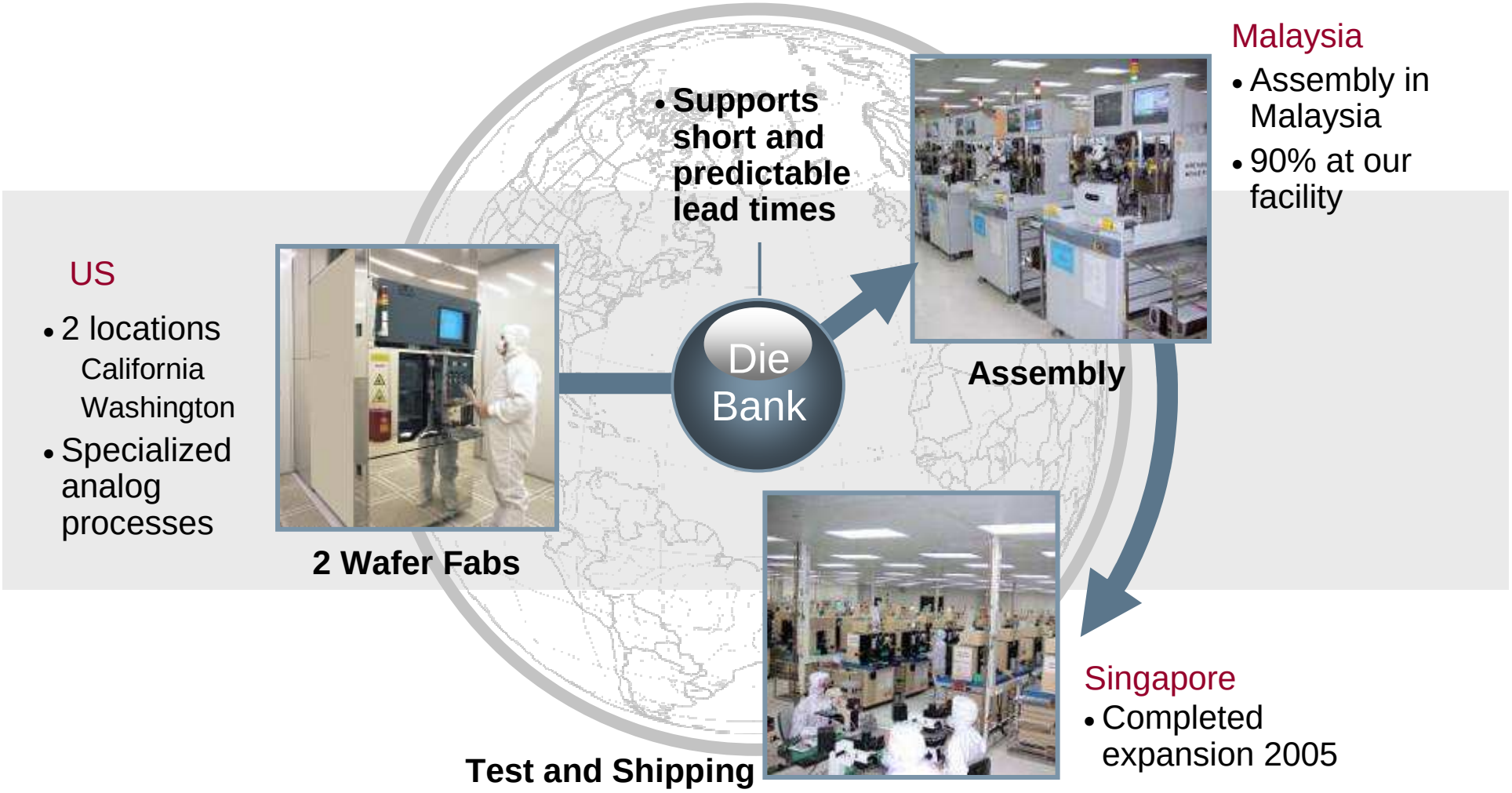
Copyright © 2010 Linear Technology. All rights reserved.

The Value We Deliver

- Innovative, High Performance Products
- Outstanding Price/Performance Solutions
- Expert Worldwide Technical Support
- Best Quality and Reliability
- Dependable Delivery

LINEAR TECHNOLOGY

Dependable Delivery



US

- 2 locations
  - California
  - Washington
- Specialized analog processes

Malaysia

- Assembly in Malaysia
- 90% at our facility

Singapore

- Completed expansion 2005

© 2007 Linear Technology

Why Use LTspice?

- Stable SPICE circuit simulation with
  - Unlimited number of nodes
  - Schematic/symbol editor
  - Waveform viewer
  - Library of passive devices
- Fast simulation of switch mode power supplies
  - Steady state detection
  - Turn on transient
  - Step response
  - Efficiency / power computations
- Advanced analysis and simulation options
  - Not covered in this lab class (sort of)
- Outperforms or as powerful as pay-for tools
  - In other words LTspice is free!
- Automatically builds syntax for common tasks

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

SPICE = Simulation Program with Integrated Circuit Emphasis

LTspice is also a great schematic capture / BOM tool

How Do I Get LTspice and Documentation?

- Go to http://www.linear.com/software
- Left-Click on Download LTspice IV
- Follow the instructions to install

LTspice IV LTspice IV 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 LTspice IV (Updated May 5, 2009) ▪ LTspice Users Guide ▪ LTspice Getting Started Guide ▪ LTspice Demo Circuit Collection

How Do I Get Started using LTspice?

How Do I Get Started Using LTspice?

- Use one of the 100's demo circuit available on linear.com
  - Designed and Reviewed by Factory Apps Group
  - Go to http://www.linear.com/software
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point, but is not production-ready
  - Used to prove out part models, and are not complete designs.
  - Components are typically "ideal" components and will need to be modified based on your operating conditions
- Use the schematic editor to create your own design
  - LTspice contains models for most LTC power devices and many more
- Use simulation circuits posted on the LTspice Yahoo! User's Group. tech.groups.yahoo.com/group/LTspice
  - Also contains many very helpful discussion threads

You can also check out LTspice capabilities using the education examples available on C:\Program Files\LTC\SwCADIII\examples\Educational

How Do I Get Started Using LTspice?

- Use one of the 100's demo circuit available on linear.com
  - Designed and Reviewed by Factory Apps Group
  - Go to http://www.linear.com/software
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point, but is not production-ready
  - Used to prove out part models, and are not complete designs.
  - Components are typically "ideal" components and will need to be modified based on your operating conditions
- Use the schematic editor to create your own design
  - LTspice contains models for most LTC power devices and many more
- Use simulation circuits posted on the LTspice Yahoo! User's Group. tech.groups.yahoo.com/group/LTspice
  - Also contains many very helpful discussion threads

Demo Circuits on linear.com

Go to http://www.linear.com/software

LTspice IV

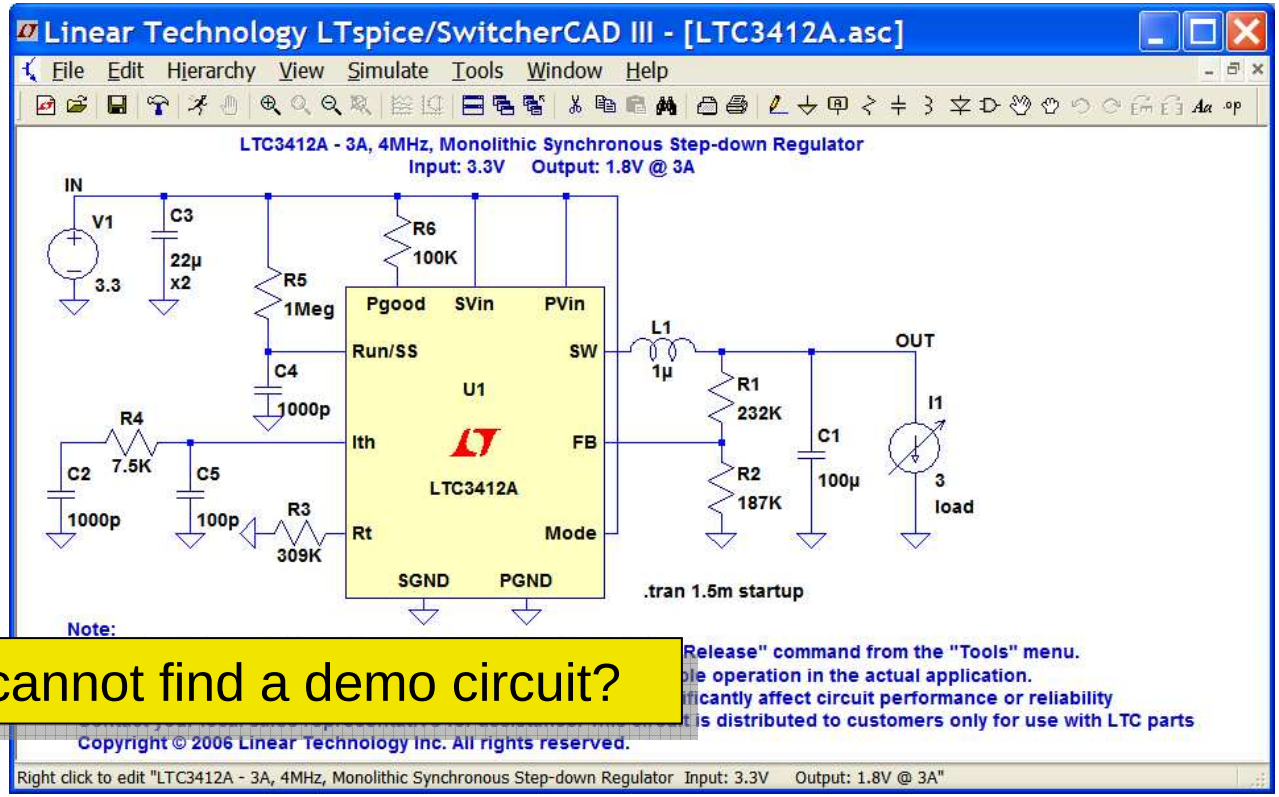
LTspice IV is a high performance Spice III simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulators. Our enhancements to Spice have made simulating switching regulators extremely fast compared to normal Spice simulators, allowing 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 LTspice IV (Updated May 5, 2009)
- LTspice Users Guide
- LTspice Getting Started Guide
- LTspice Demo Circuit Collection

Part Number	Updated	Download
LT1071HV - 5A and 2.5A High Efficiency Switching Regulators	May 5th, 2006	LT1071HV.asc
LT1072HV - 1.25A High Efficiency Switching Regulator	May 5th, 2006	LT1072HV.asc
LT1076HV - Step-Down Switching Regulator	May 5th, 2006	LT1076HV.asc
LT1111 - Micropower DC/DC Converter Adjustable and Fixed 5V, 12V	May 26th, 2006	LT1111.asc
LT1172HV - 100kHz, 5A, 2.5A and 1.25A High Efficiency Switching Regulators	May 5th, 2006	LT1172HV.asc
LT1173 - Micropower DC/DC Converter Adjustable and Fixed 5V, 12V	Jun 12th, 2006	LT1173.asc
LT1308B - Single Cell High CurrentMicropower 600kHz Boost DC/DC Converter	May 26th, 2006	LT1308B.asc
LT1370HV - 500kHz High Efficiency 6A Switching Regulator	May 26th, 2006	LT1370HV.asc

Demo Circuits

- ✓ Designed and reviewed by factory apps group
  - It remains the customer's responsibility to verify proper and reliable operation in the actual application
  - Component substitution and printed circuit board layout may significantly affect circuit performance or reliability



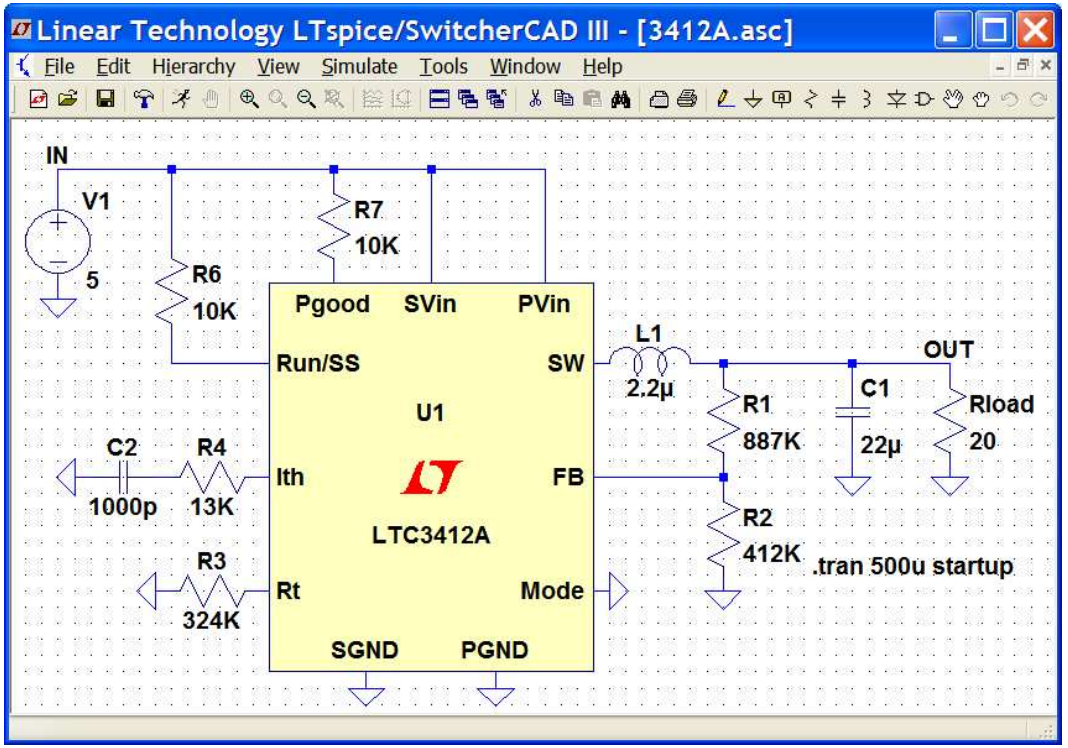
What if I cannot find a demo circuit?

How Do I Get Started Using LTspice?

- Use one of the 100's demo circuit available on linear.com
  - Designed and Reviewed by Factory Apps Group
  - Go to http://www.linear.com/software
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point, but is not production-ready
  - Used to prove out part models, and are not complete designs.
  - Components are typically "ideal" components and will need to be modified based on your operating conditions
- Use the schematic editor to create your own design
  - LTspice contains models for most LTC power devices and many more
- Use simulation circuits posted on the LTspice Yahoo! User's Group tech.groups.yahoo.com/group/LTspice
  - Also contains many very helpful discussion threads

Pre-drafted Test Fixture

- Provides a good starting point
  - These simulations / designs are not production-ready
  - Used to prove out part models, and are not complete designs.



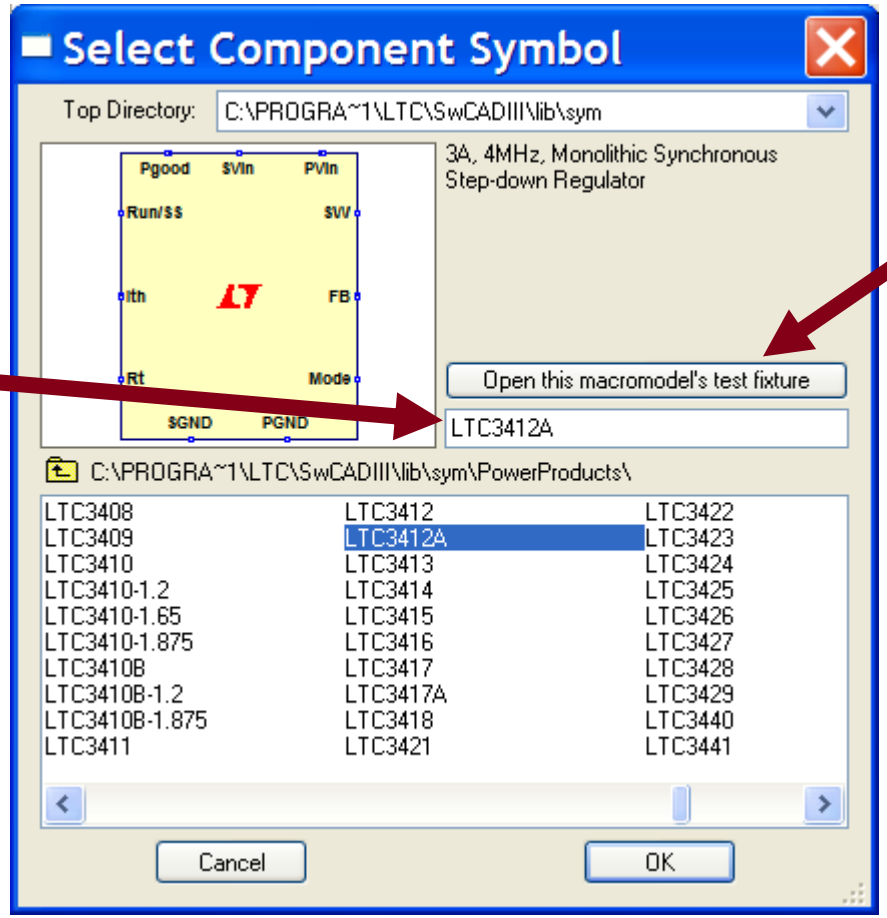
- It remains the customer's responsibility to verify proper and reliable operation in the actual application
- Printed circuit board layout may significantly affect circuit performance and reliability

Selecting a Model & Opening Test Fixture

- Use the “root” part to search for the model
  - i.e. 3412A
- Select “Open this macromodel’s test fixture”

1. Enter 3412A

2. Select

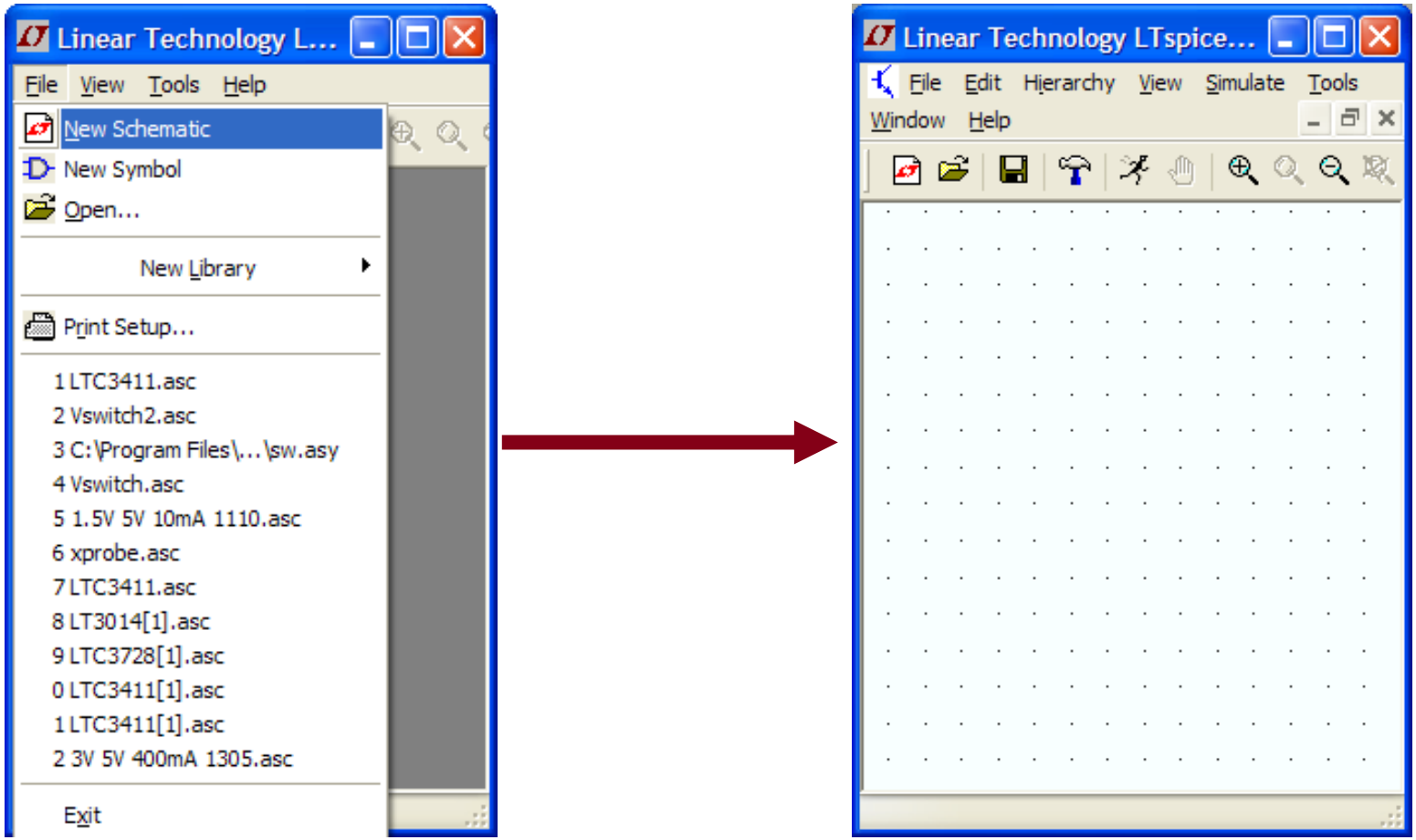


How Do I Get Started Using LTspice?

- Use one of the 100's demo circuit available on linear.com
  - Designed and Reviewed by Factory Apps Group
  - Go to http://www.linear.com/software
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point, but is not production-ready
  - Used to prove out part models, and are not complete designs.
  - Components are typically "ideal" components and will need to be modified based on your operating conditions
- Use the schematic editor to create your own design
  - LTspice contains models for most LTC power devices and many more
- Use simulation circuits posted on the LTspice Yahoo! User's Group tech.groups.yahoo.com/group/LTspice
  - Also contains many very helpful discussion threads

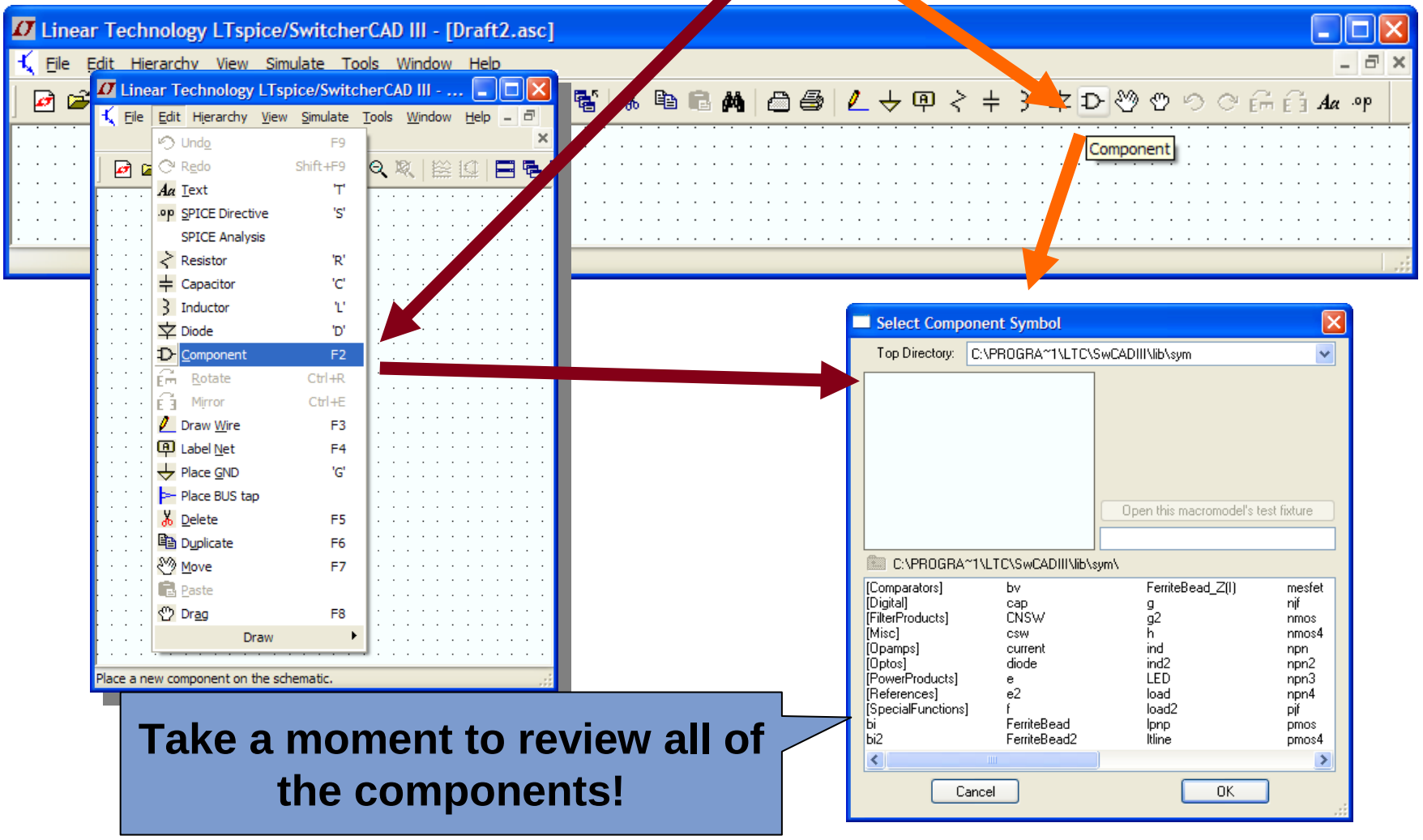
Start With a New Schematic

- Select File and New Schematic
  - Will open up a blank schematic screen



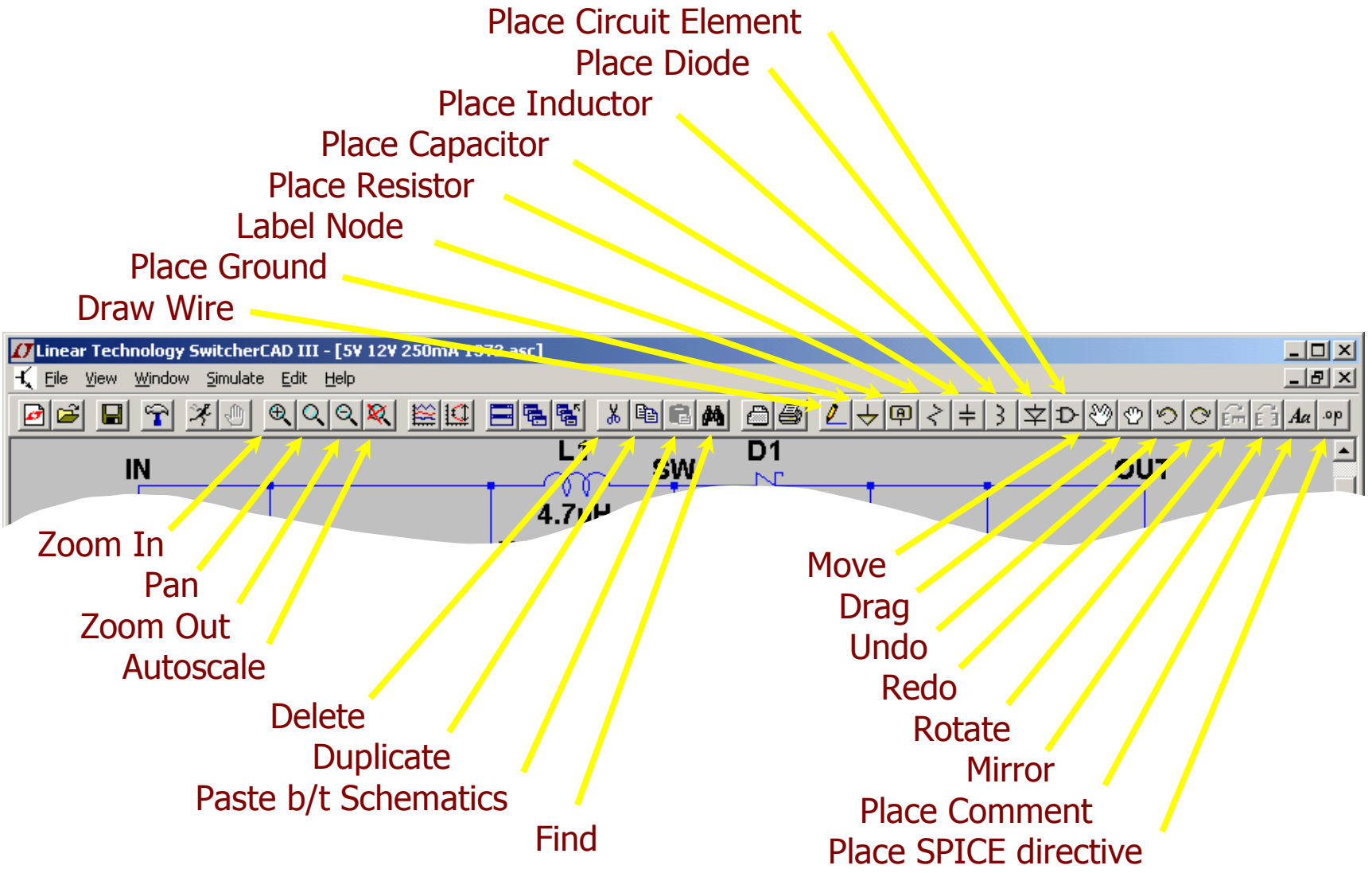
Add a Component

- Use Add a Component or F2



Take a moment to review all of the components!

Schematic Editing



Using Labels to Specify Units for Component Attributes

- K = k = kilo = 10^3
- MEG = meg = 10^6
- G = g = giga = 10^9
- T = t = tera = 10^{12}

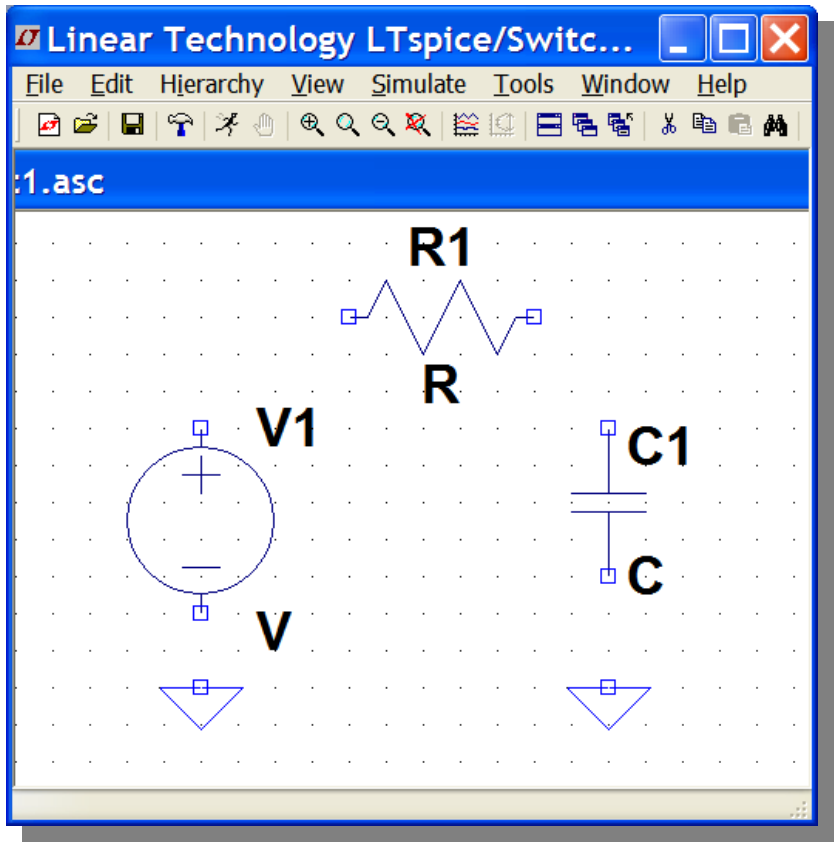
- M = m = milli = 10-3
- U = u = micro = 10-6
- N = n = nano = 10-9
- P = p = pico = 10-12
- F = f = femto = 10-15

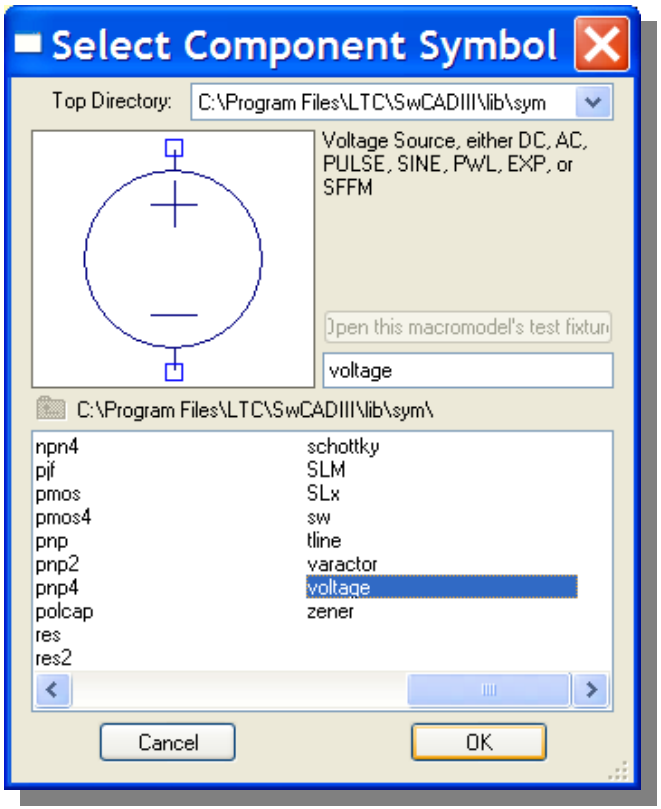
Hints

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

Wiring up a Simple RC Circuit

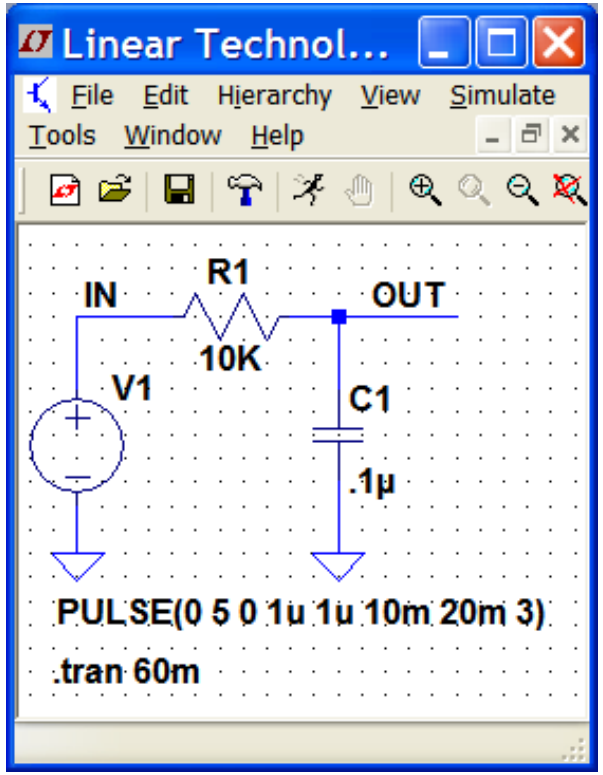
- Using the toolbar, select New Schematic
- Using the toolbar, select a Resistor, Capacitor and Ground. Place these on the schematic as shown below. Use **Ctrl R** to rotate before placement
- Using the toolbar, select Component. From the component window, type “voltage” in the dialog box, and click “OK” to place a voltage source

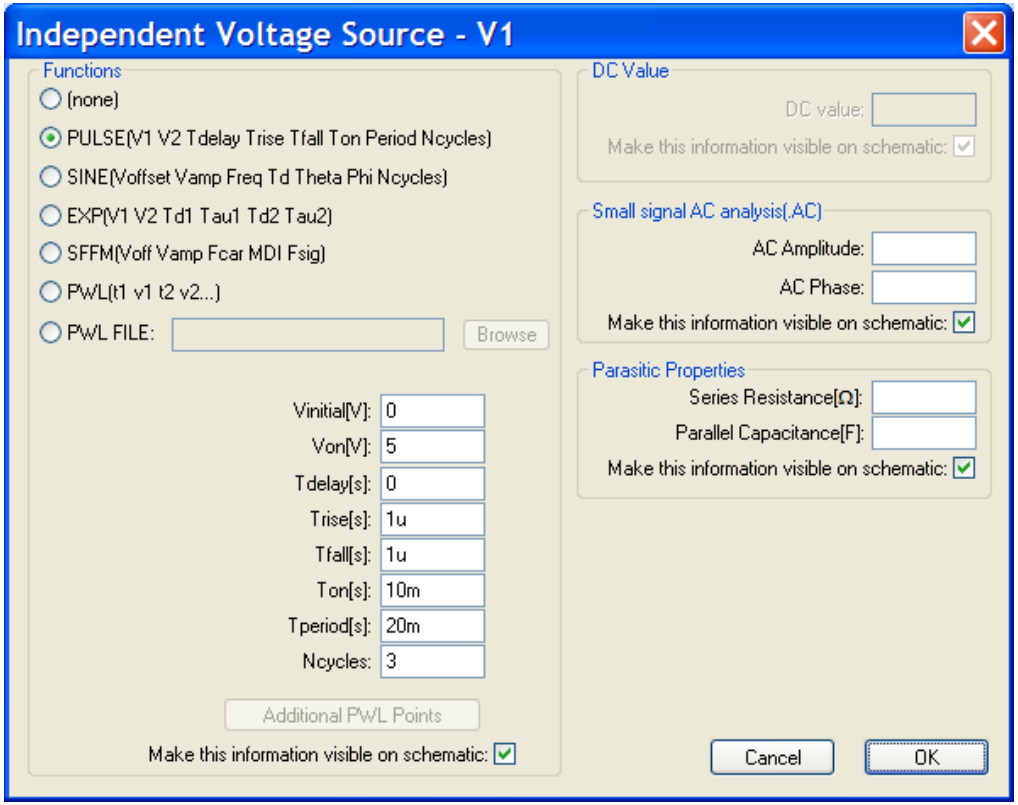


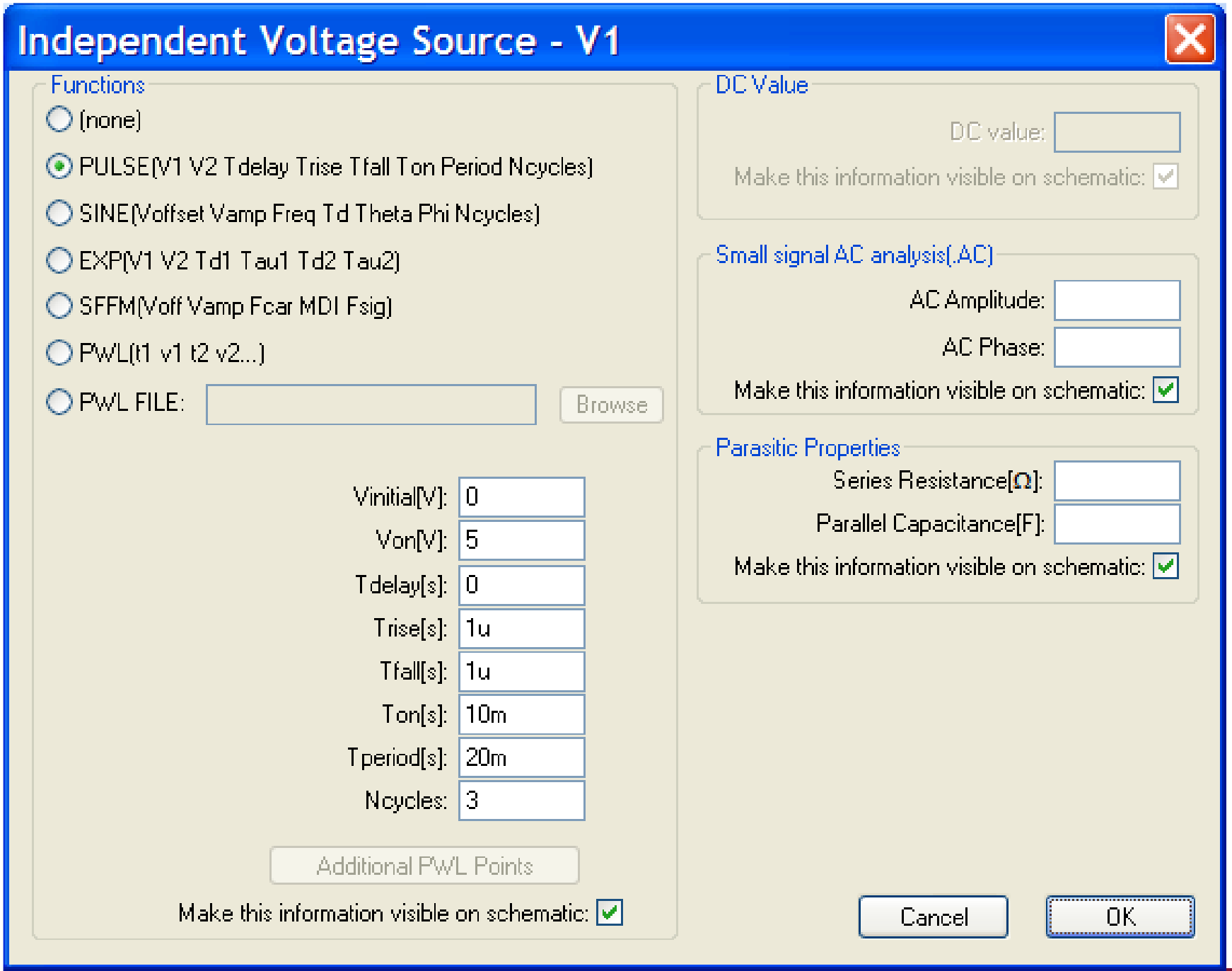


Wiring up a Simple RC Circuit

- Using the toolbar, select Wire. Wire up the RC circuit as shown below.
- Using the toolbar, select Label Net. Label the input/output nodes as shown below
- Right-Click on each component to change its value as shown below
- Right-Click on the voltage source and enter the parameters shown below under the “Advanced” tab.

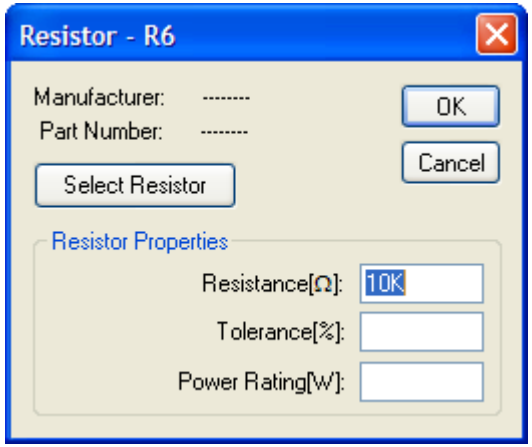


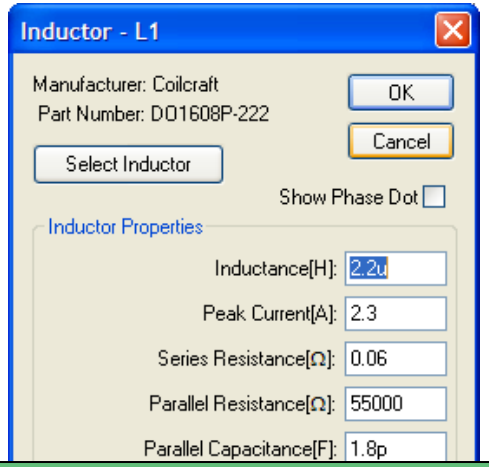


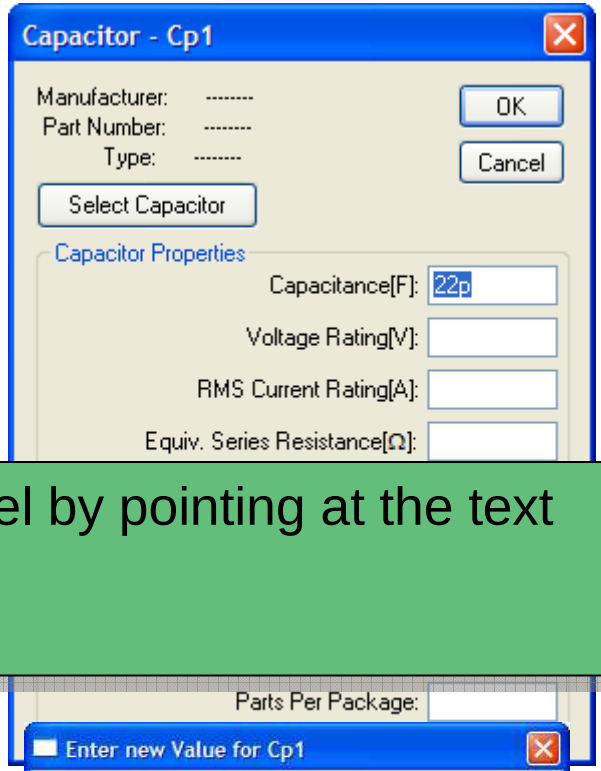

Independent Voltage Source - V1
Functions
(none)
PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
EXP(V1 V2 Td1 Tau1 Td2 Tau2)
SFFM(Voff Vamp Fcar MDI Fsig)
PWL(t1 v1 t2 v2...)
PWL FILE:
Browse
Vinitial[V]: 0
Von[V]: 5
Tdelay[s]: 0
Trise[s]: 1u
Tfall[s]: 1u
Ton[s]: 10m
Tperiod[s]: 20m
Ncycles: 3
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
Series Resistance[Ω]:
Parallel Capacitance[F]:
Make this information visible on schematic:
Cancel
OK

Editing Components

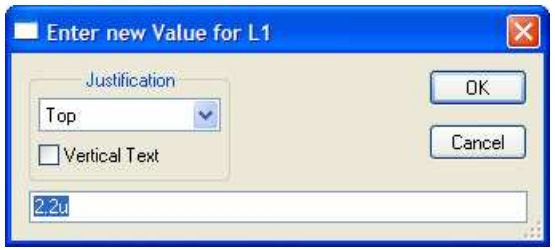
- Component attributes can be edited by pointing at the component with the mouse and Right-Clicking

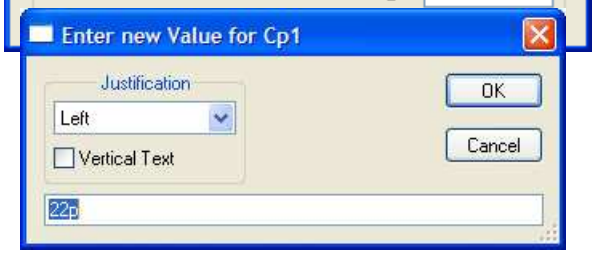






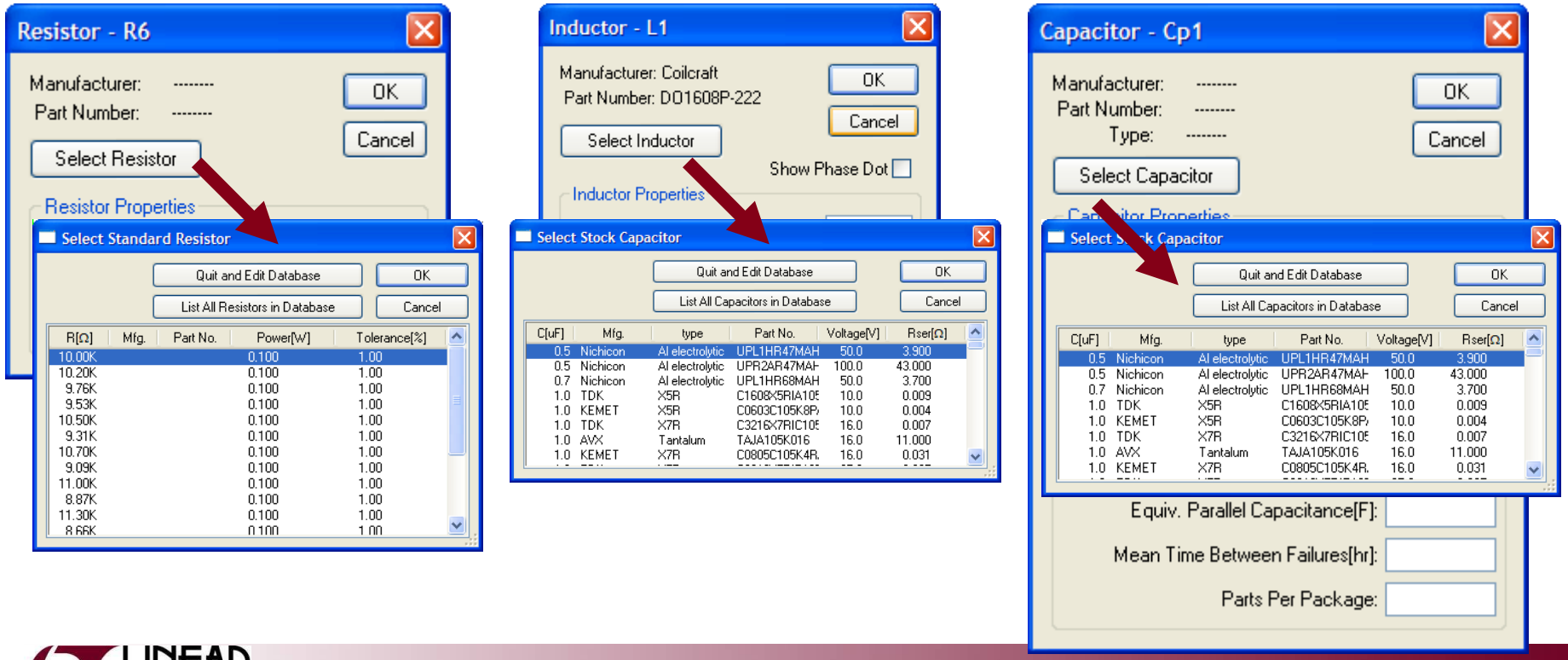
- You can also edit the visible attribute and label by pointing at the text with the mouse and then right-clicking
  - Mouse cursor will turn into a text caret





Component Database

- Components such as
  - Resistors, capacitors, inductors, diodes,
  - Bipolar transistors, MOSFET transistors, JFET transistors
  - Independent voltage and current sources
  - You can access a database of known devices



How Do I Get Started using LTspice?

- Use one of the 100's demo circuit available on linear.com
  - Designed and Reviewed by Factory Apps Group
  - Go to http://www.linear.com/software
- Use a pre-drafted test fixture (JIG)
  - Provides a good starting point, but is not production-ready
  - Used to prove out part models, and are not complete designs.
  - Components are typically "ideal" components and will need to be modified based on your operating conditions
- Use the schematic editor to create your own design
  - LTspice contains models for most LTC power devices and many more
- Use simulation circuits posted on the LTspice Yahoo! User's Group tech.groups.yahoo.com/group/LTspice
  - Also contains many very helpful discussion threads

LTspice Yahoo! User's Group Web Page

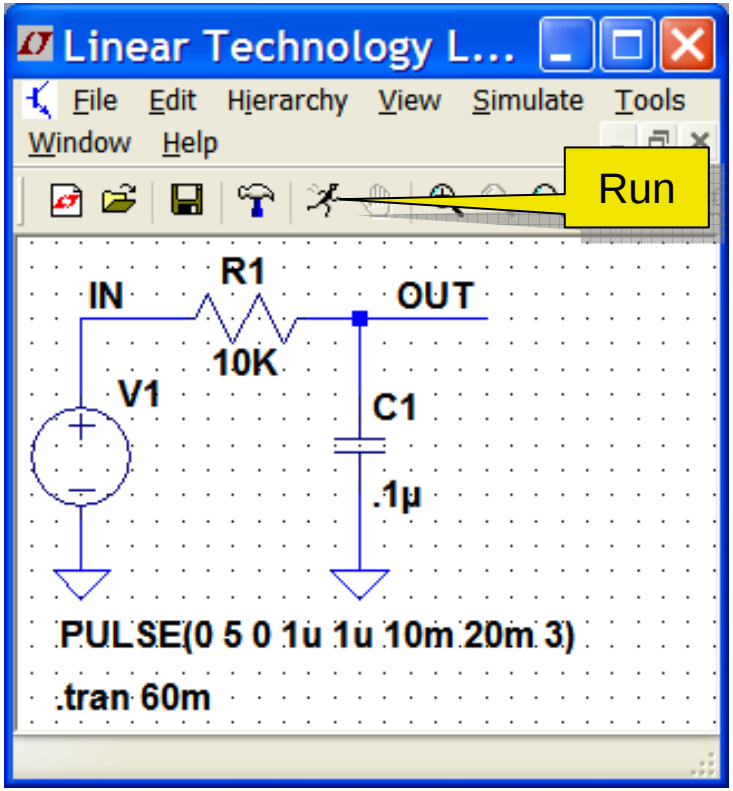
URL

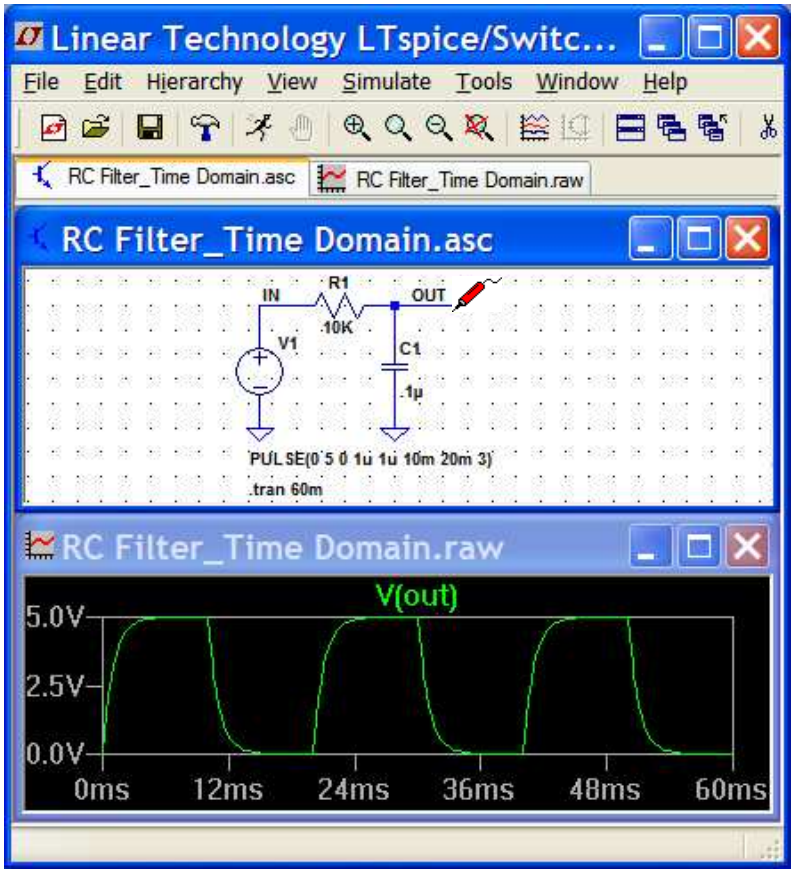
Join the group here. As of Jan 2010, there are over 21,600 members!

How Do You Run and Probe a Circuit in LTspice?

Running the RC Circuit Simulation

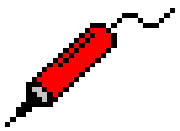
- With the RC circuit in the active window, click on the “Running Person” button on the tool bar
- The Edit Simulation Command window will appear. Set the Stop Time for 60msec, and click “OK”
- Using the mouse, click on the “OUT” node to display the output voltage waveform





Waveform Viewer

- LTspice has an integrated waveform viewer
  - Plot the voltage on any wire by simply point and click

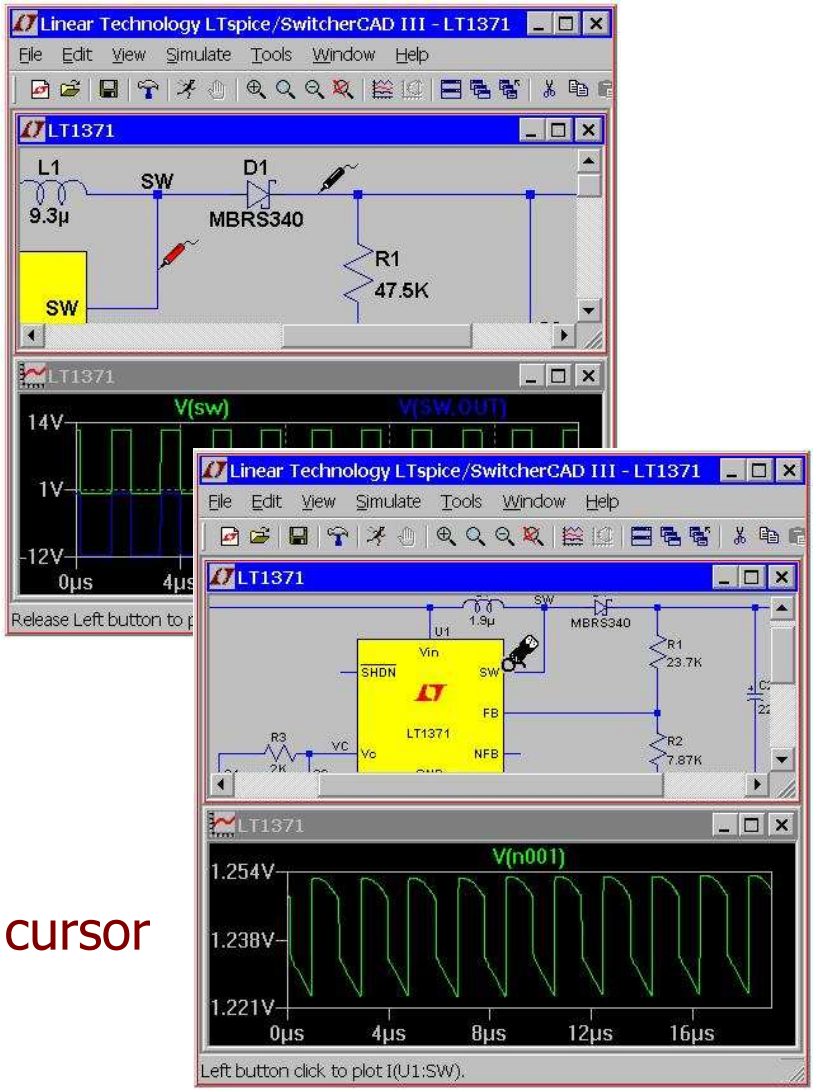


Voltage probe cursor

  - Plot the current through any component with two connections by clicking on the body of the component

    - R, C, L
    - Convention of positive current is in the direction into the pin

Current probe cursor



Running a Demo Circuit

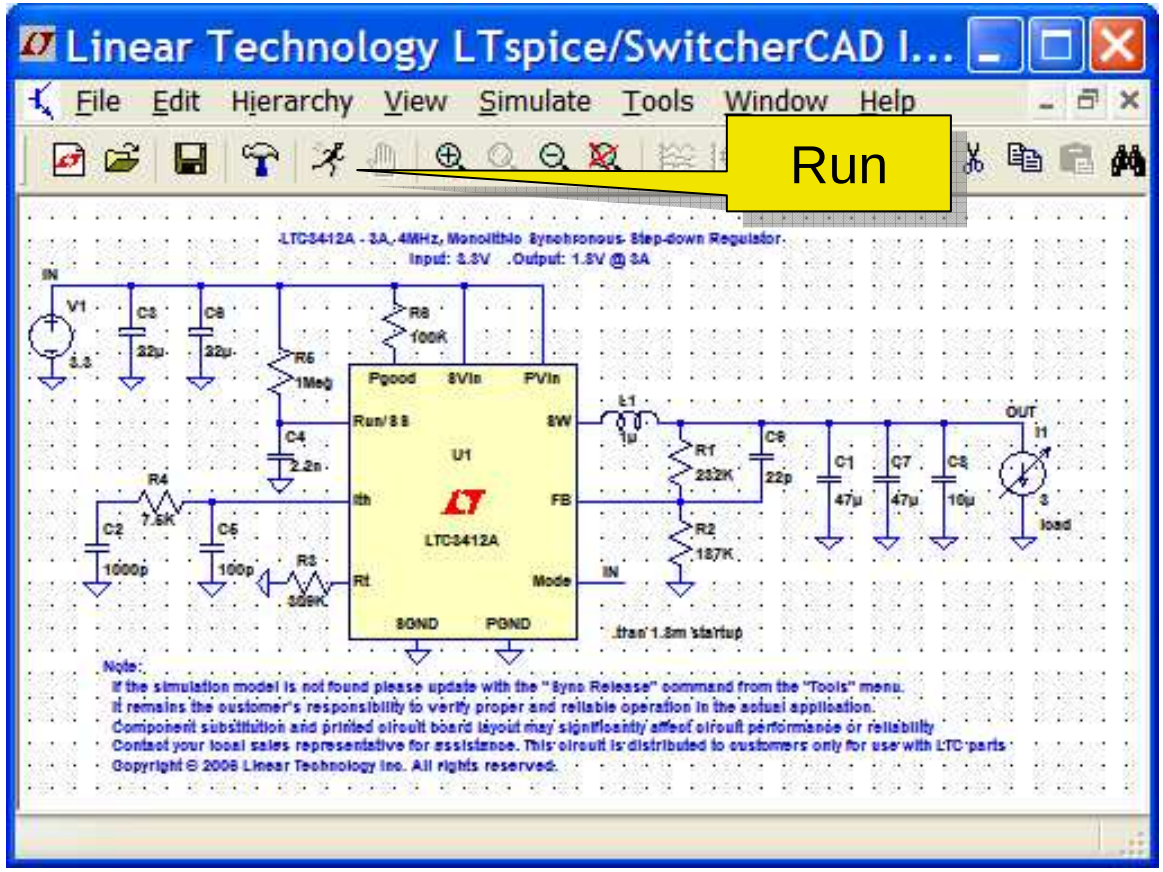
- Access the LTC3412A demo circuit from the LTC website. It is located in the “LTspice Basic Lab Class” folder on your desktop
  - Click File ---> Open, and navigate to the LTspice Basic Lab Class folder on your desktop. Look for the file titled “LTC3412ADCLoad.asc”
  - Hotlink Nomenclature:

Class exercise

Solution to exercise

Circuits to explorer at your leisure

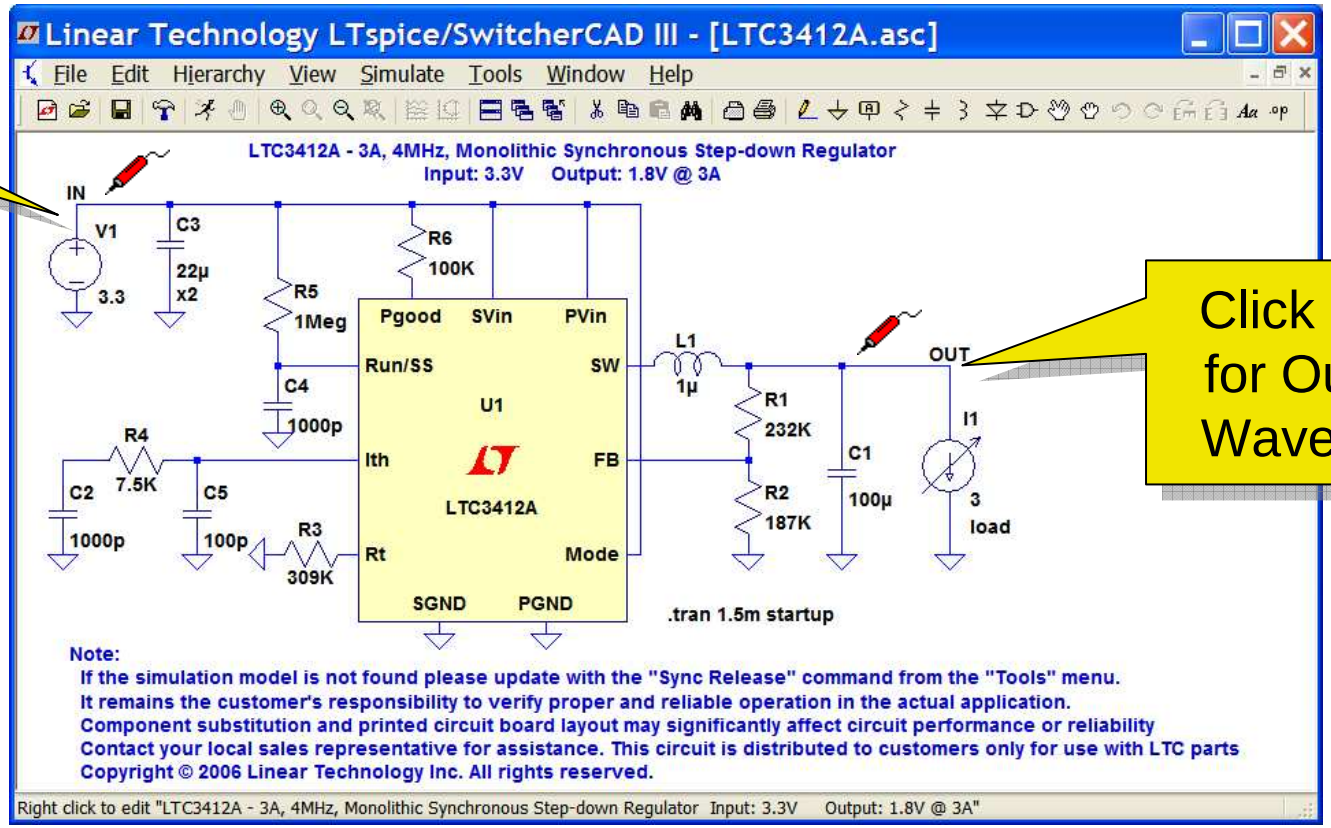
Running a Demo Circuit



- Select the “Running Man” button on the toolbar
  - The Simulation will start and waveform window will open up
  - To view waveforms, please continue to the next page….

Probing a Demo Circuit

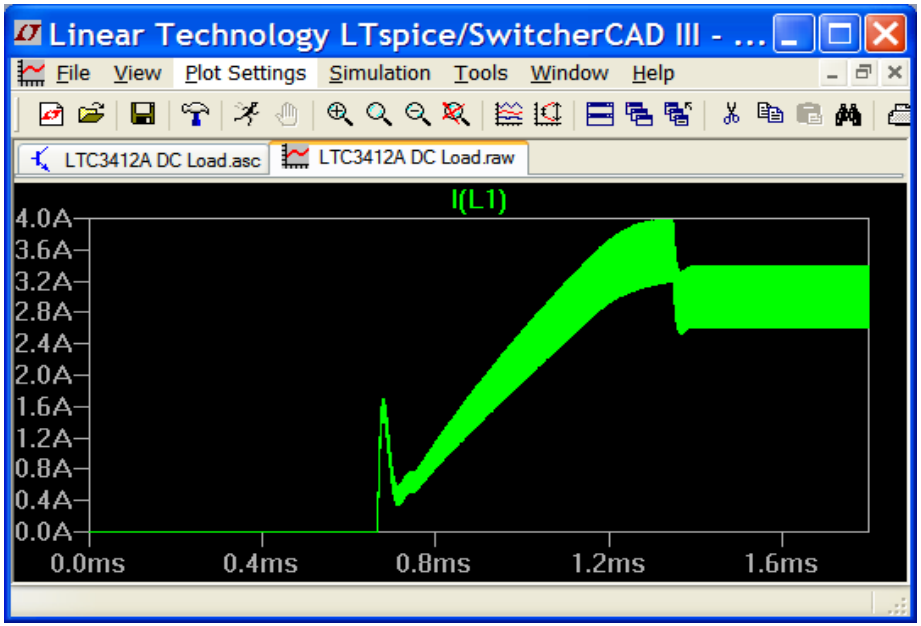
Click Here for Input Waveform



Click Here for Output Waveform

- All Demo Circuits have INs and OUTs clearly labeled to help you quickly select them
- Select the waveform of a node by clicking on IN and OUT

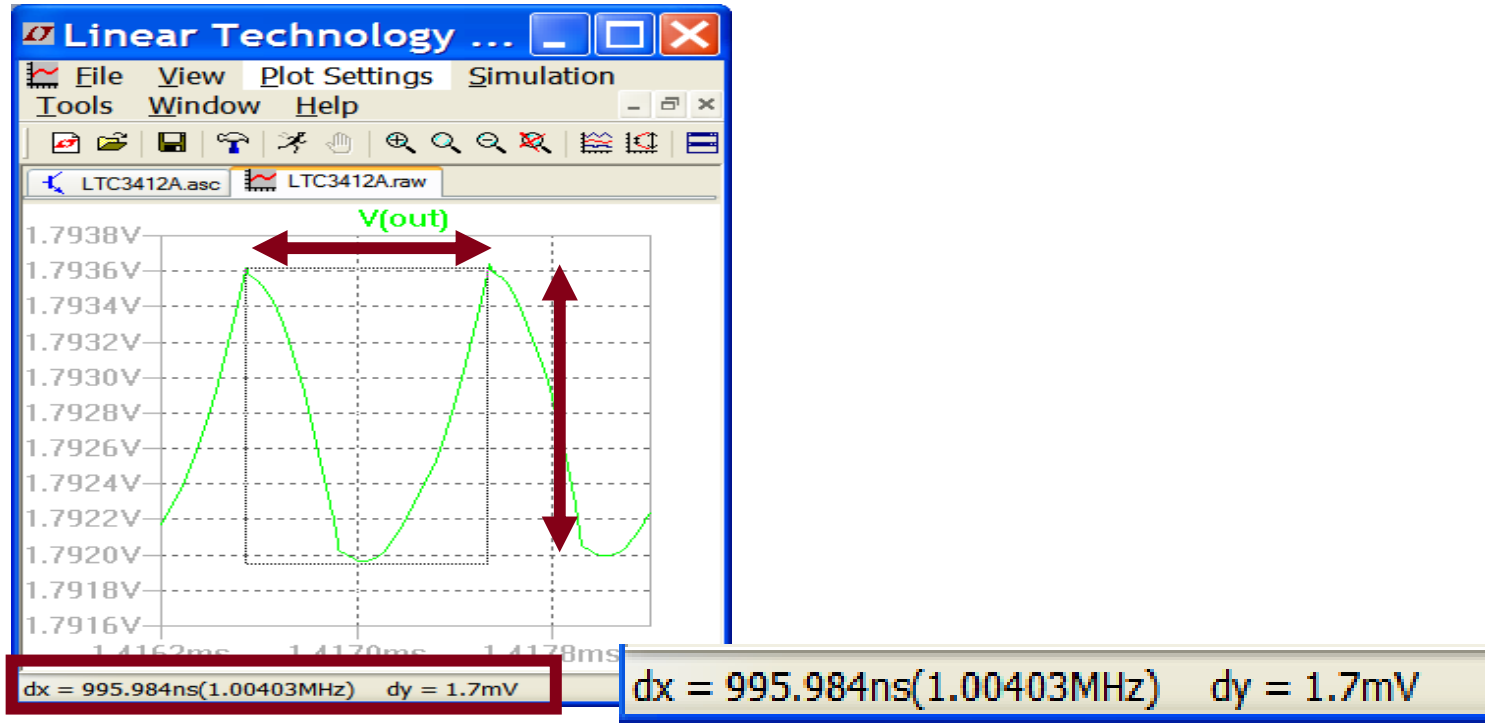
Zooming In and Out on a Waveform



- Using the mouse, click on inductor L1 to display the inductor current waveform
- In the waveform window, use the mouse to zoom in and out
  - Click and drag a box about the region you wish to see drawn larger
- Using the toolbar, click on “Zoom full extents”, to zoom back out

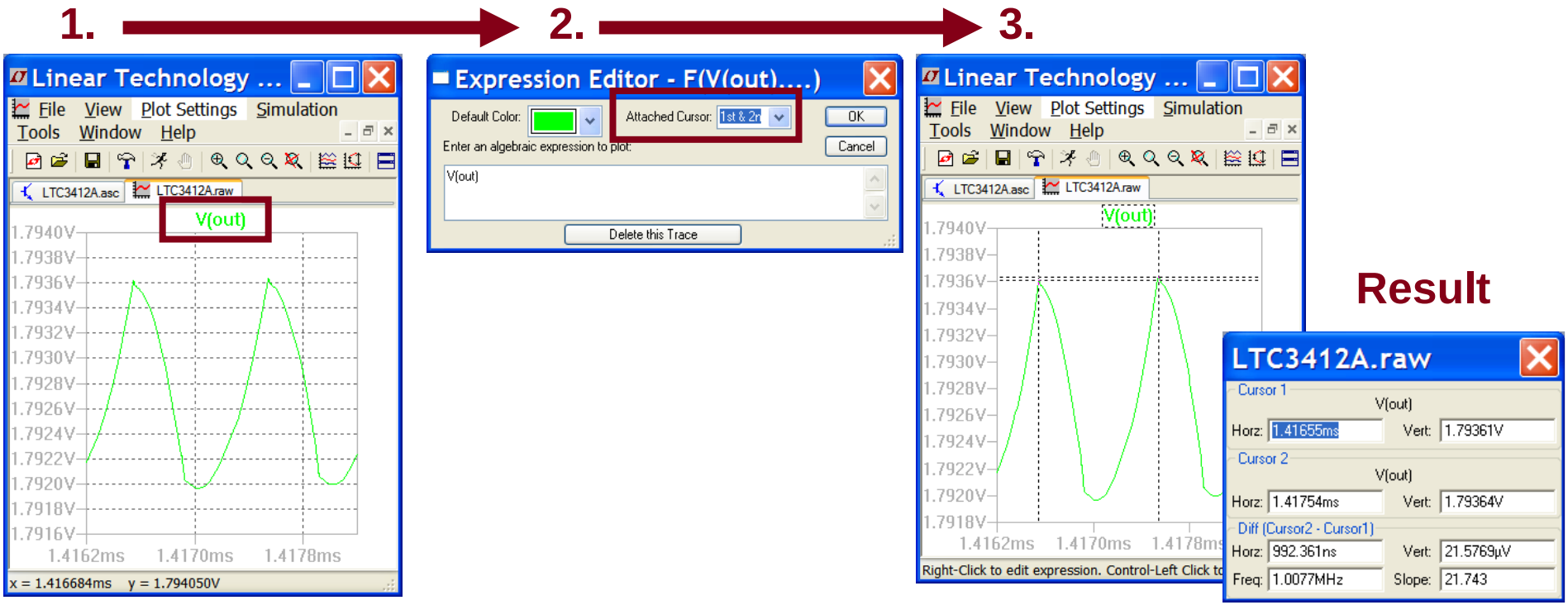
Measuring V, I and Time in the Waveform (Measurement Using Zoom)

1. Drag a box about the region you wish to measure
   - Left-Click, drag, and hold
2. View the lower left corner of the window for the status bar. The dx and dy measurement data is displayed here.
3. Use Undo from the File menu or press “F9”



Measuring V, I and Time in the Waveform (Measurement Using Cursors)

1. Right-Click on the waveform name in the waveform window
2. For “Attached Cursor”, select “1st & 2nd”
3. Position cursors to make desired measurements.

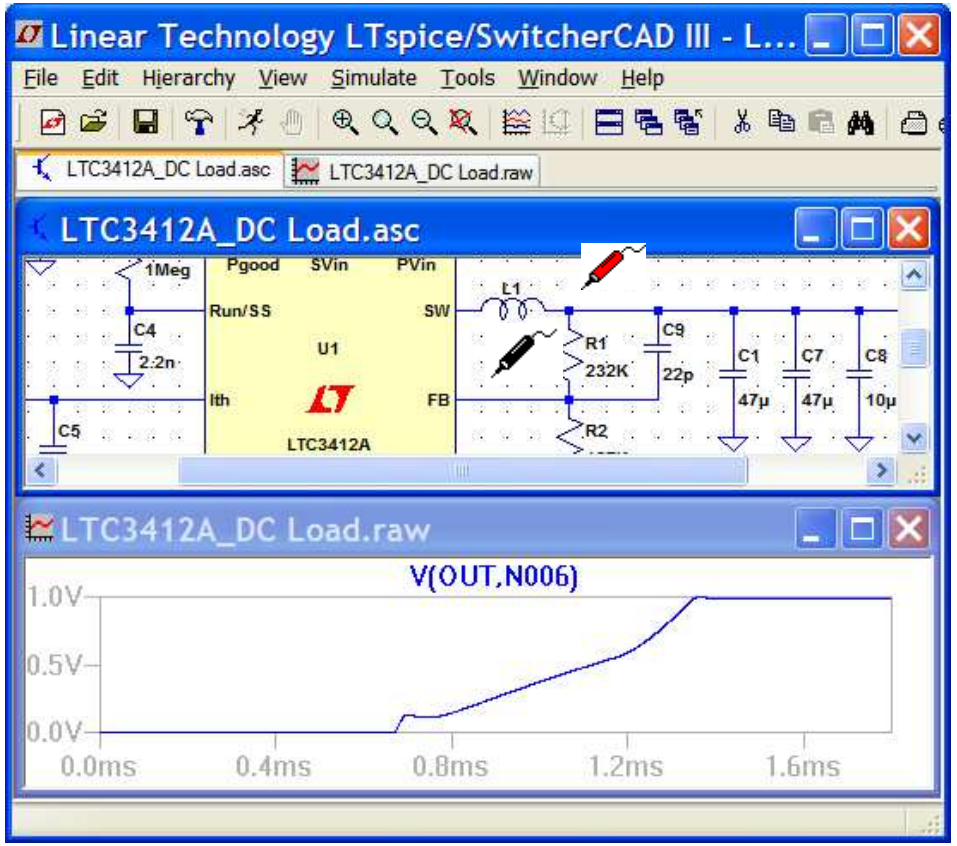


Differential Voltage Measurement

- Click on one node and drag the mouse to another node
  - Red voltage probe at the first node
  - Black probe on the second
- Will produce a differential voltage measurement

Example:

Measure across LTC3412A top resistor in feedback divider

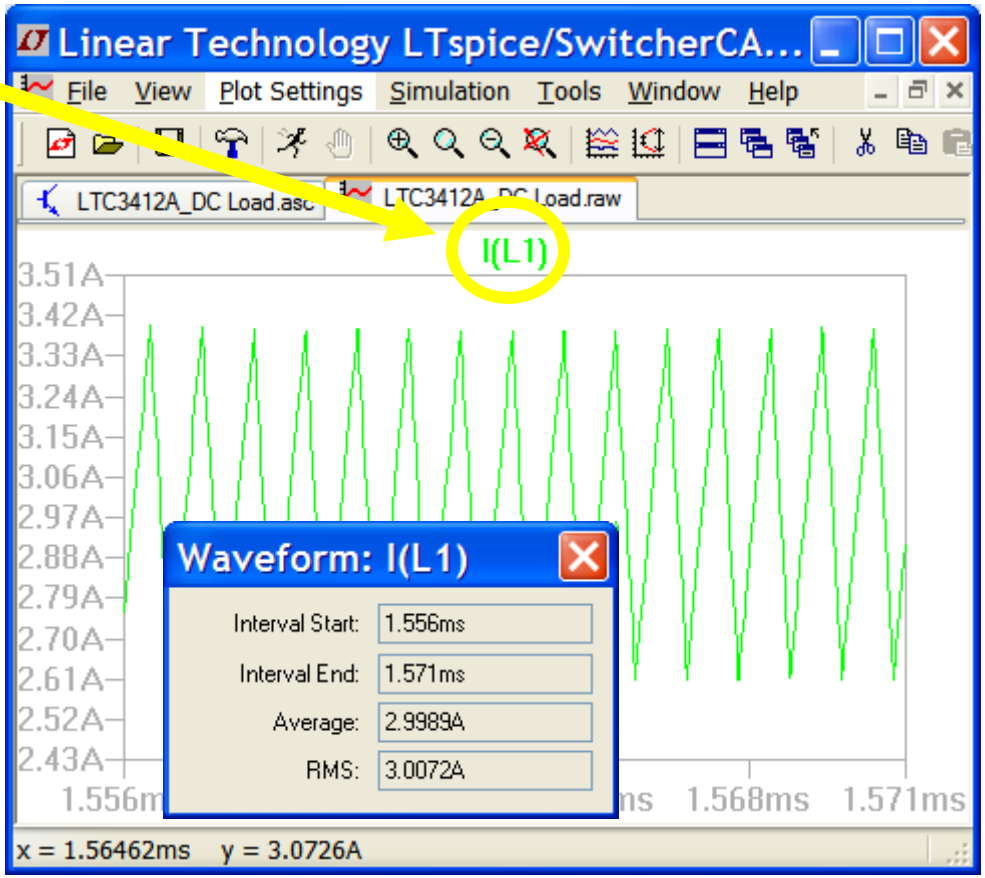


Average & RMS Calculations

- Average & RMS Current, Voltage, or Power Dissipation
- Click on inductor L1 to display the inductor current waveform
  - **Ctrl-Left-Click** the I(L1) trace label in the waveform view

Example:

Measure average and RMS current for inductor in LTC3412A circuit. Zoom in as shown for this waveform.

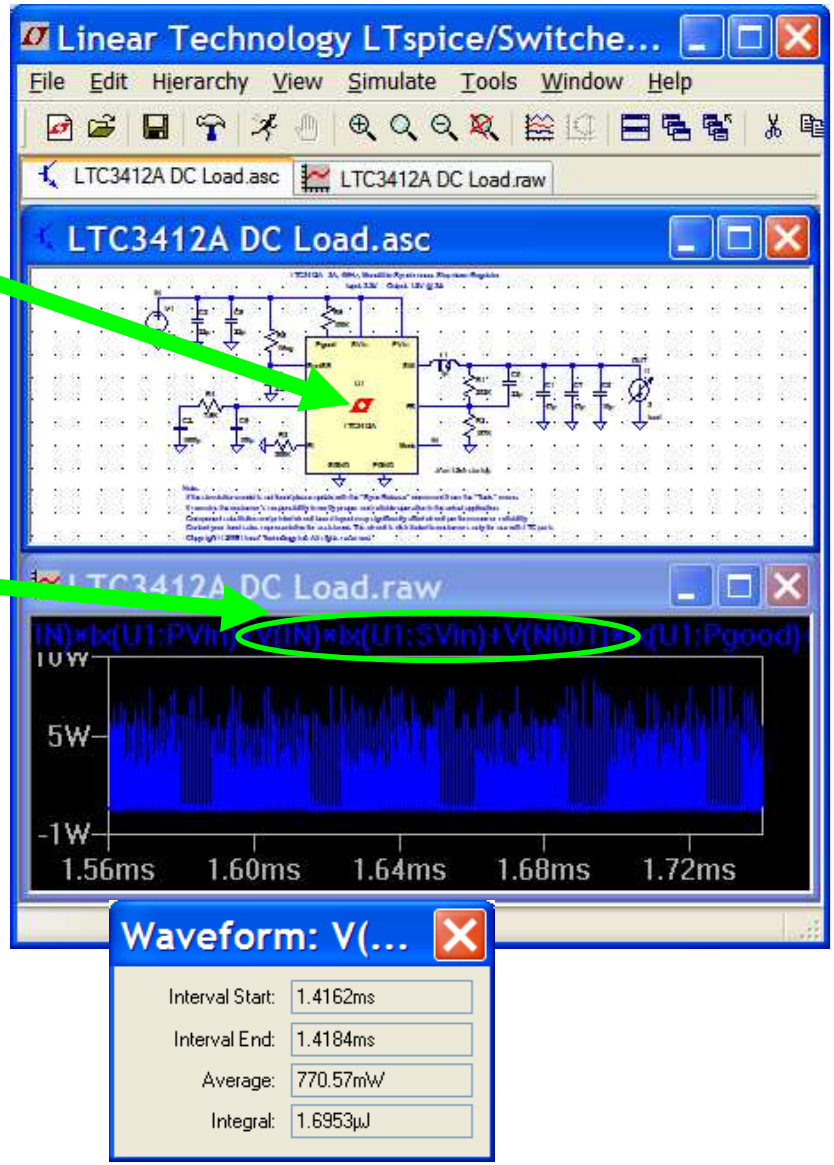


Instantaneous & Average Power Dissipation

- Instantaneous Power Dissipation
  - **Hold down the Alt key and Left-Click** on the symbol of the LTC3412A
  - Waveform is displayed in units of Watts
- Average Power Dissipation
  - **Click, hold, and drag** in the waveform window to display waveform at steady state
  - **Ctrl-Left-Click** on the Power Dissipation Trace Label in the waveform view
  - Waveform summary window will appear which shows power dissipation in the IC

Example:

Measure the power dissipation in the LTC3412A IC

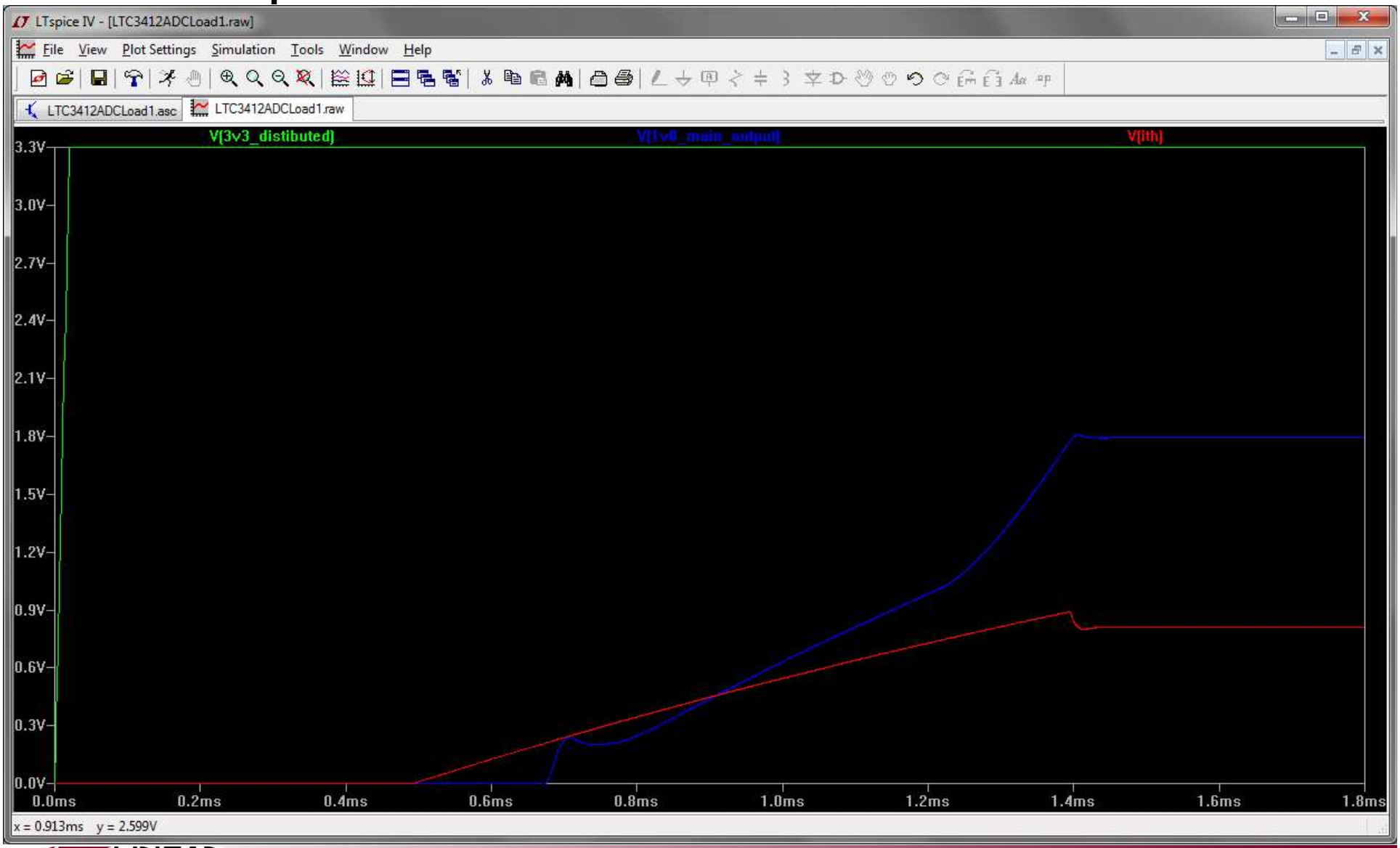


Advantages of Labeling

- Replaces arcane SPICE machine node names with easy to understand and remember human names
- Allows LTspice circuit nodes to match those on your production schematic, i.e. “TP15”

Advantages of Labeling

- **Compare this….**



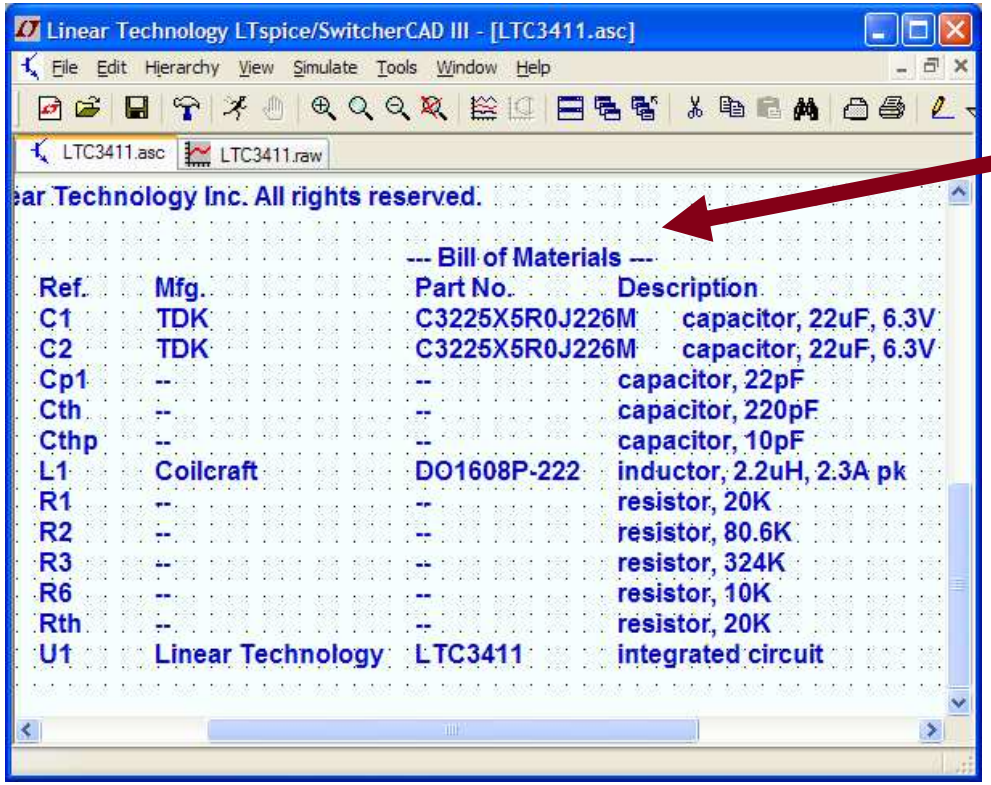
Advantages of Labeling

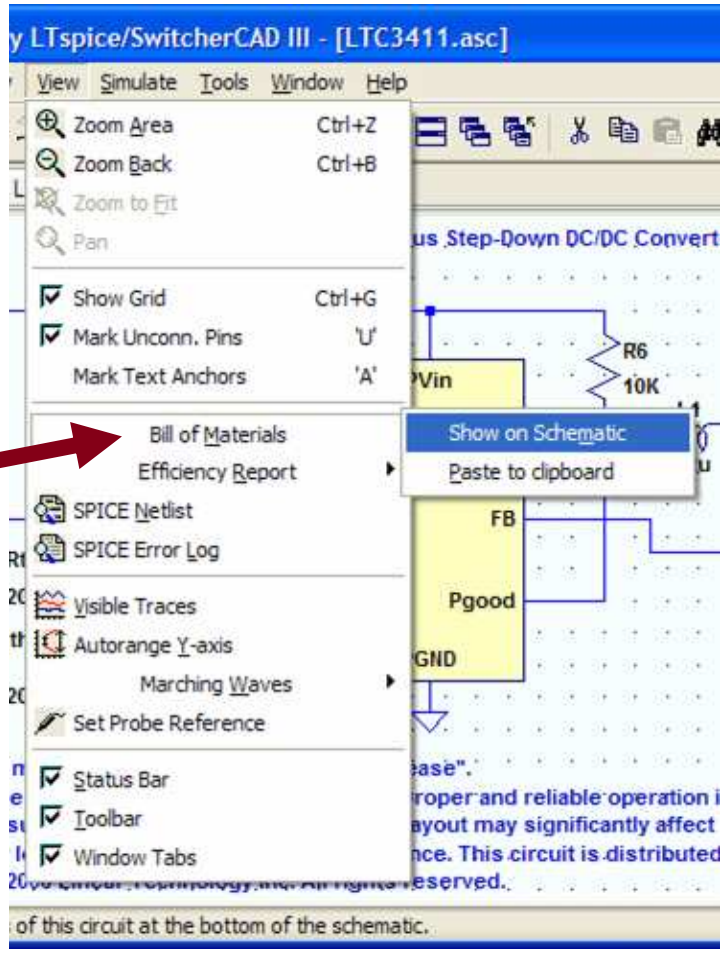
- To this….

Generating a BOM and Efficiency Report

BOM

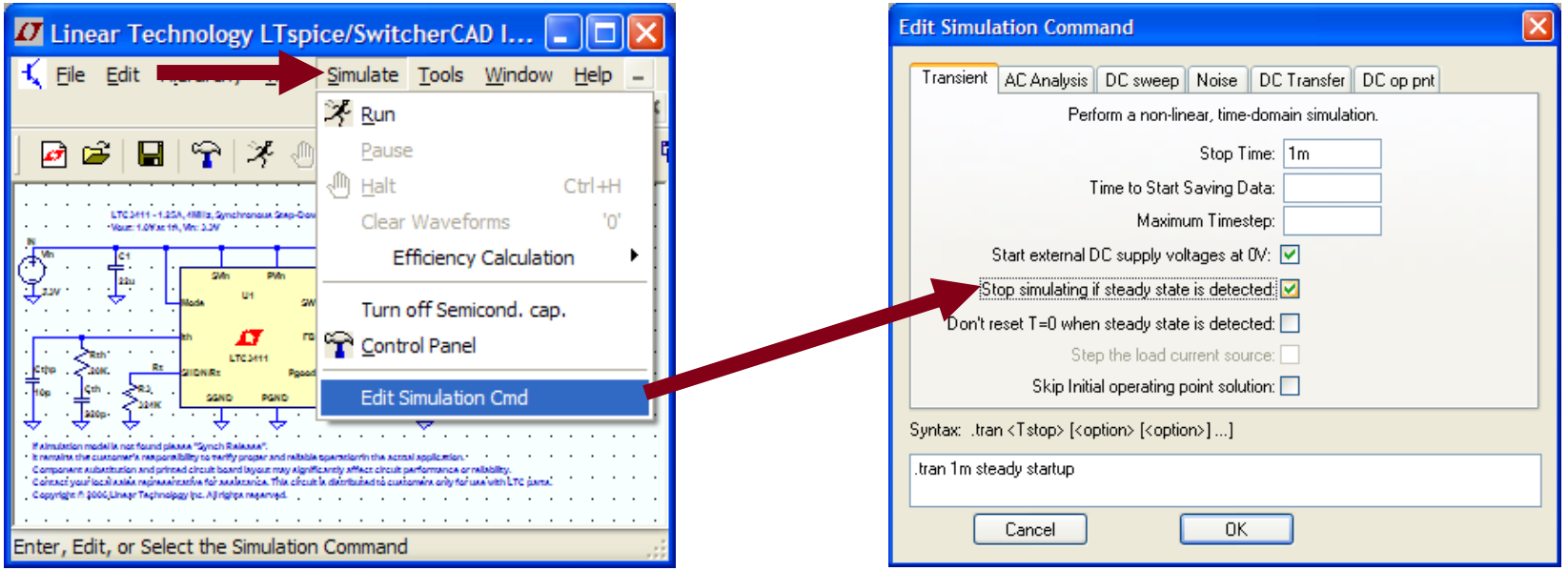
- Under View select Bill of Material
  - Displayed on Diagram
  - Paste to Clipboard





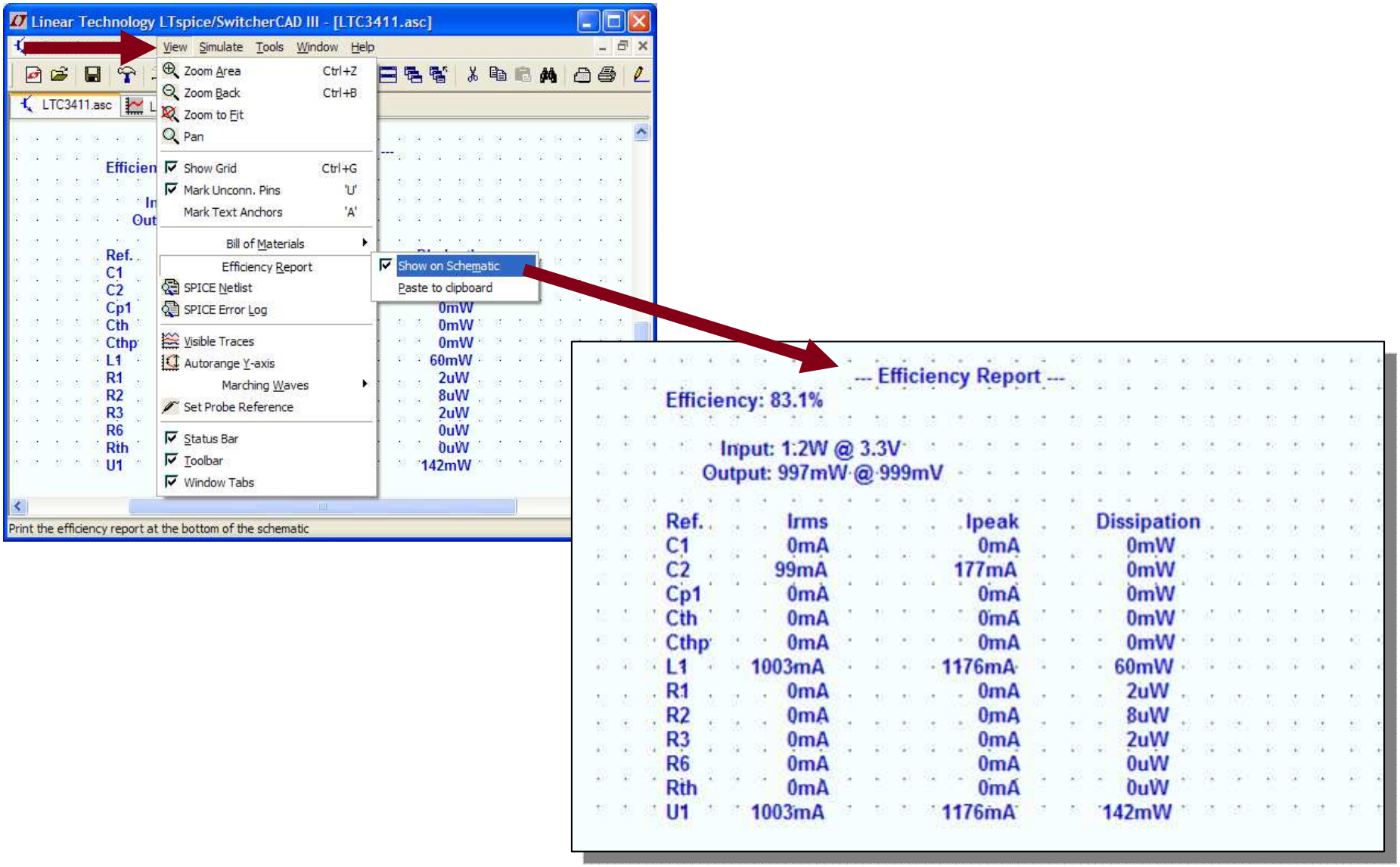
Computing Efficiency & Dissipation

- To compute efficiency of SMPS circuits:
  - Check the "Stop simulating if steady state is detected" on the Edit Simulation Command editor
  - Rerun simulation
  - Use the menu command View=>Efficiency Report



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

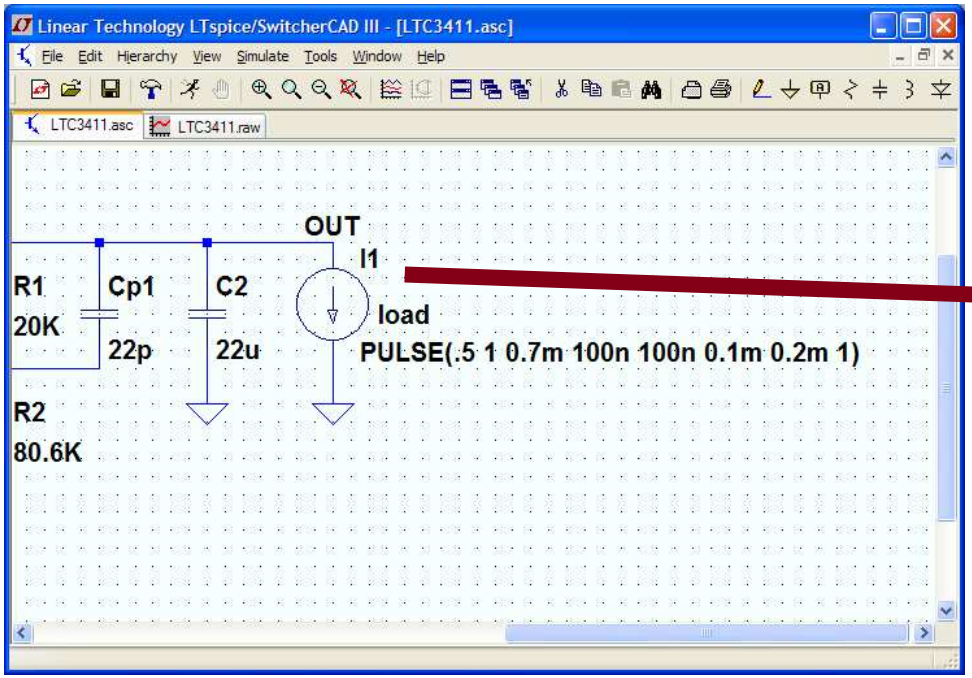
Viewing Efficiency Report

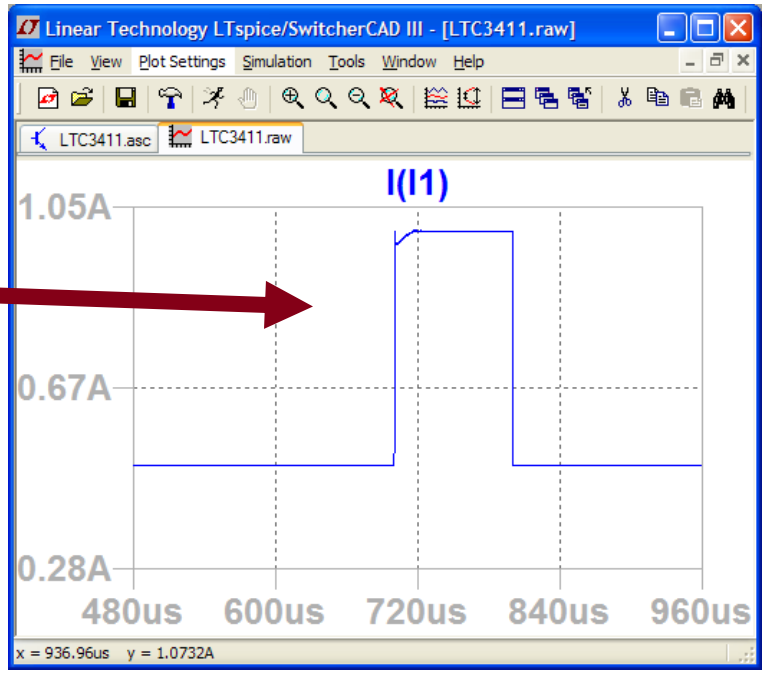


Simulating a Transient Response

Current Load and Pulse Function

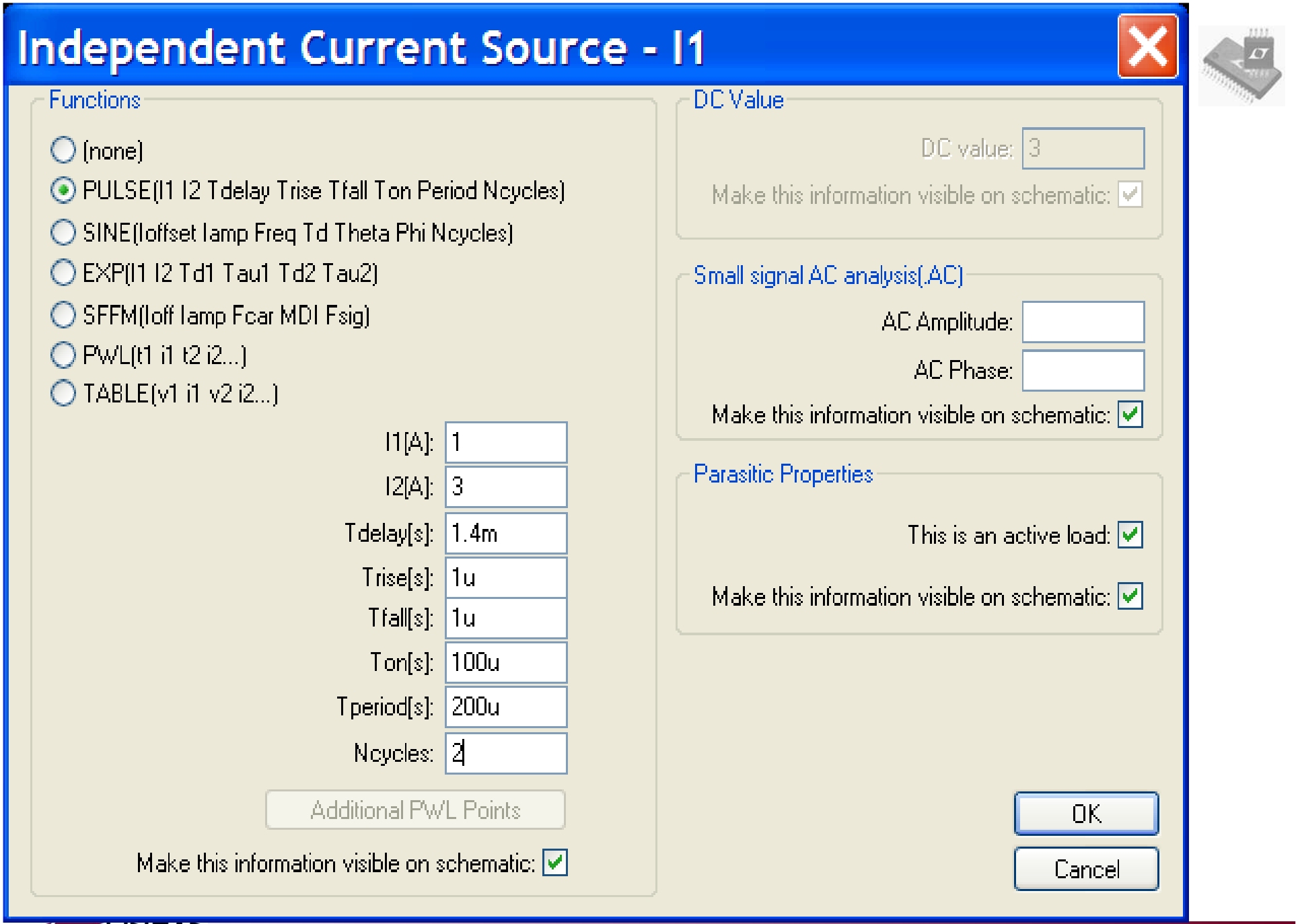
- You can simulate a load with a Resistor or Current load
- In particular the Pulse function in a current load is helpful in transient response analysis
  - Steps a current load from one value to another value





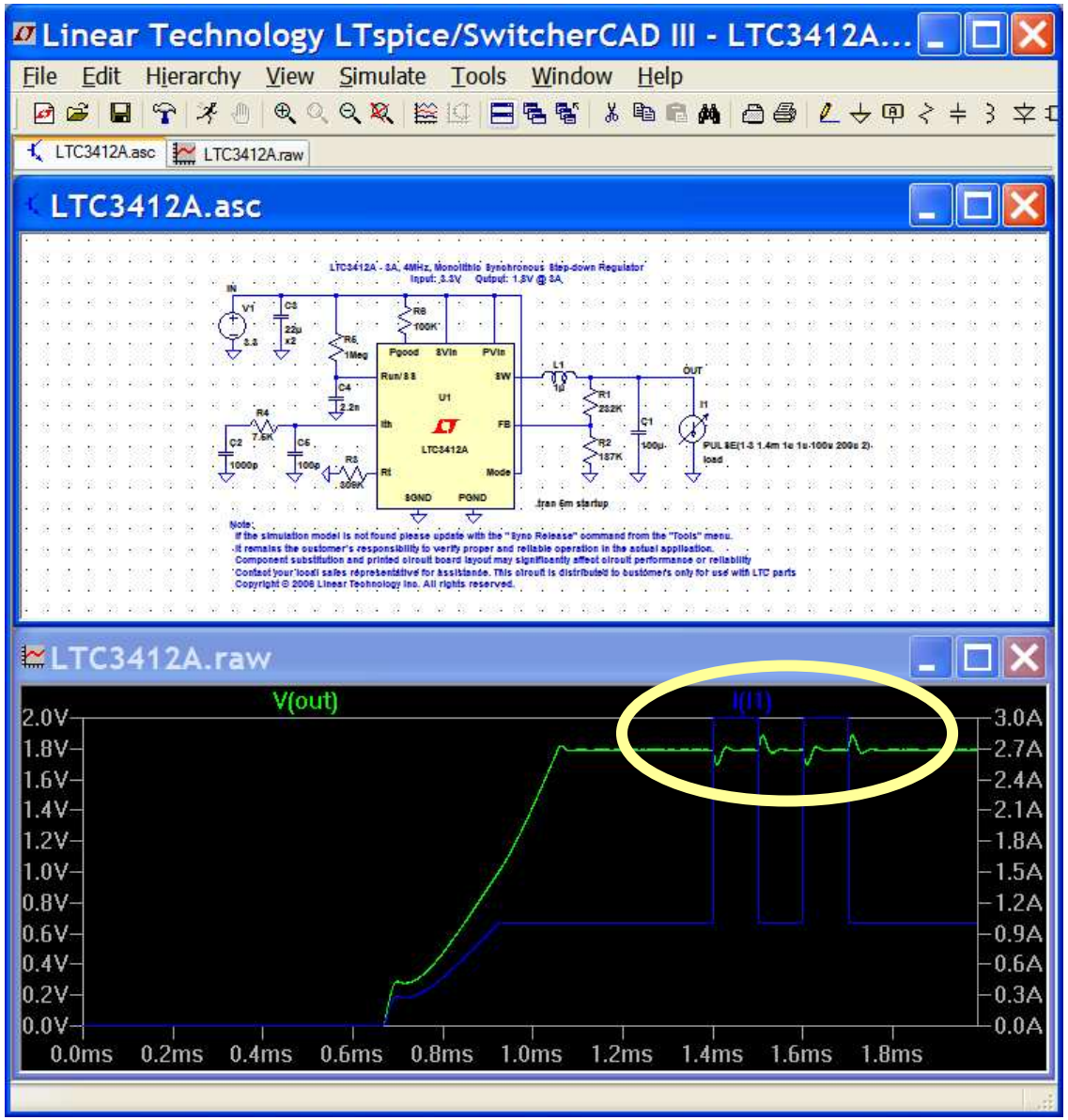
Edit the Current Load to a Pulse Function

- In the LTC3412A simulation, Right-Click on the current load
- Select “Pulse”
- Modify the Attributes (see next slide). Click “OK”.


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]: 1
I2[A]: 3
Tdelay[s]: 1.4m
Trise[s]: 1u
Tfall[s]: 1u
Ton[s]: 100u
Tperiod[s]: 200u
Ncycles: 2
Additional PWL Points
Make this information visible on schematic:
DC Value
DC value: 3
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

Run the Simulation for Transient Response

- Run the simulation
- Click on the OUT node to display Vout
- Click on the output current load to display Iout
- Notice the presence of the pulse load



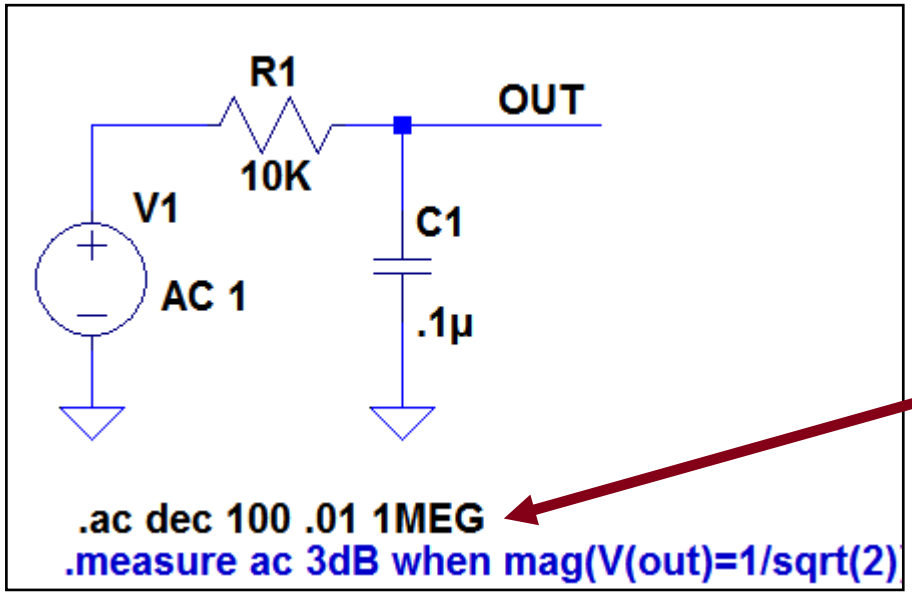
AC Analysis

AC Analysis Overview

- Performs small signal AC analysis linearized about the DC operating point
- Useful for analysis of filters, networks, stability analysis, and noise considerations

Simulating AC Analysis – RC Filter

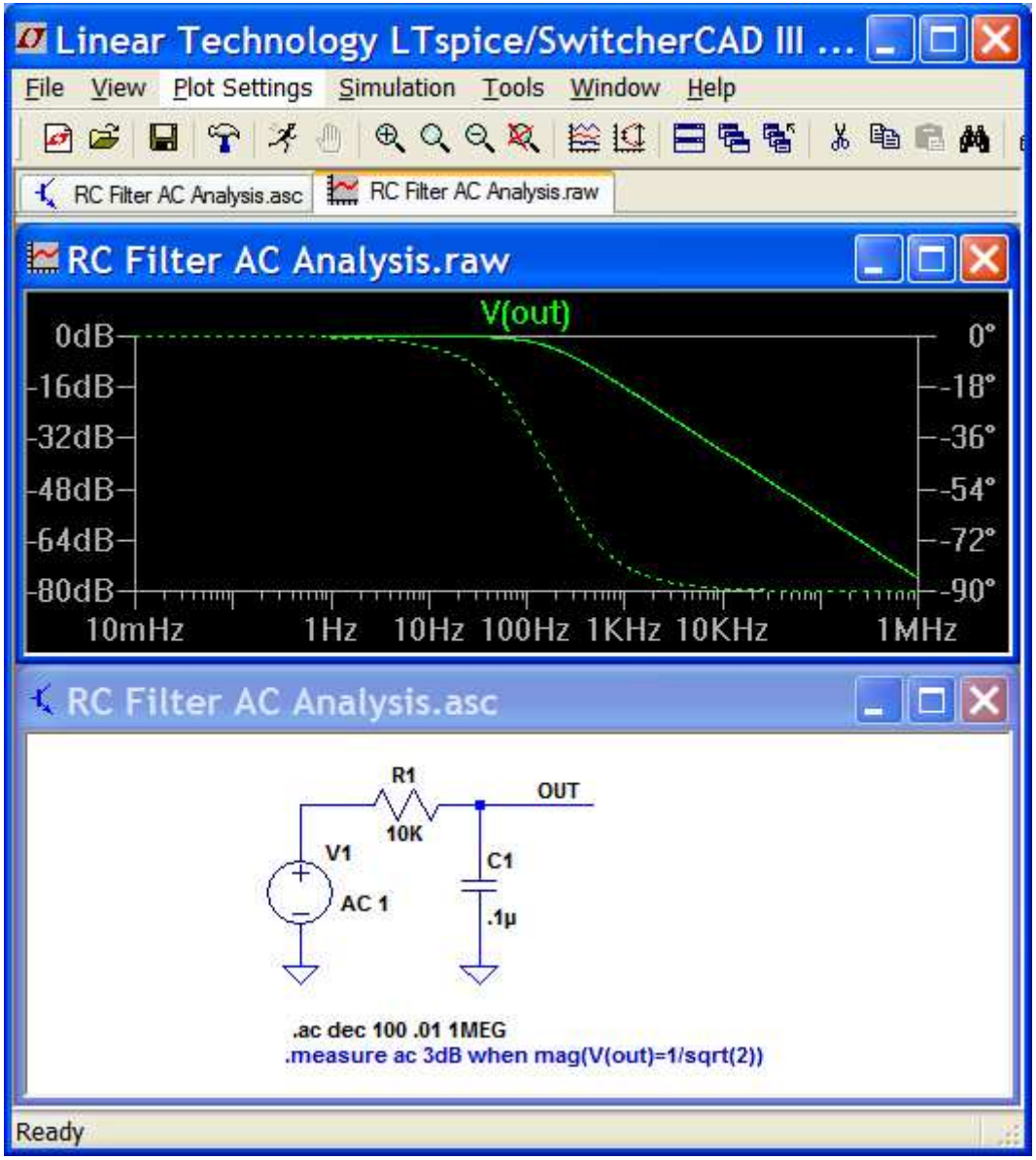
- Single pole filter using RC network
- Syntax: .ac <oct, dec, lin> <Nsteps> <StartFreq> <EndFreq>
- Example: RC network and .ac dec 100 .01 1MEG



-3dB point:
1/(2*pi*R*C) = 159Hz

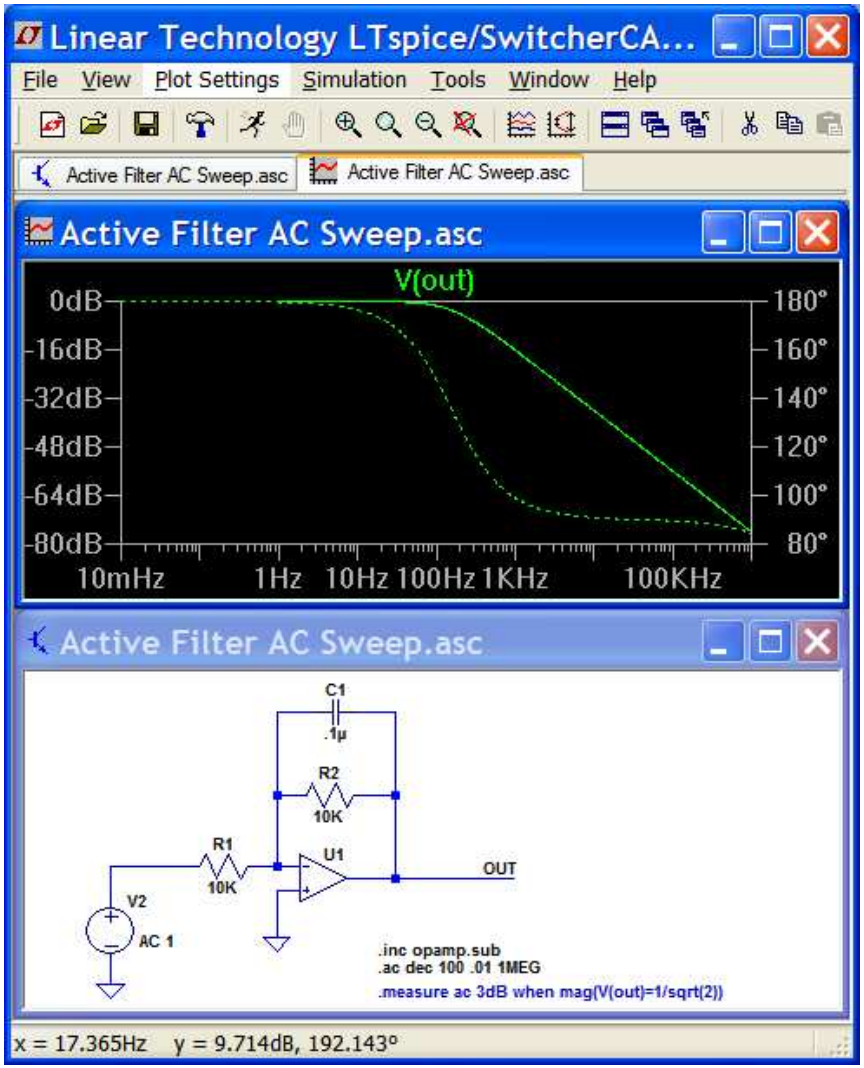
Right-click on .tran command and select "AC Analysis"

Simulating AC Analysis – RC Result



Simulating AC Analysis – Active Filter

- Single pole active filter using an opamp



Importing Third-Party SPICE Models

Importing Third-Party Spice Models

To import a third party spice model:

1.) Download the spice model file from the manufacturer's website

2.) Make sure that the spice model file is located in the same directory as the LTspice simulation file

3.) Add the following spice directive to the LTspice simulation file (Edit ---> SPICE Directive):

.include spice_model_file_name.abc

4.) Modify the component name in the LTspice schematic to match the component name contained in the spice model file (Right-Click on the device name, and modify accordingly)

Importing Third-Party Spice Models

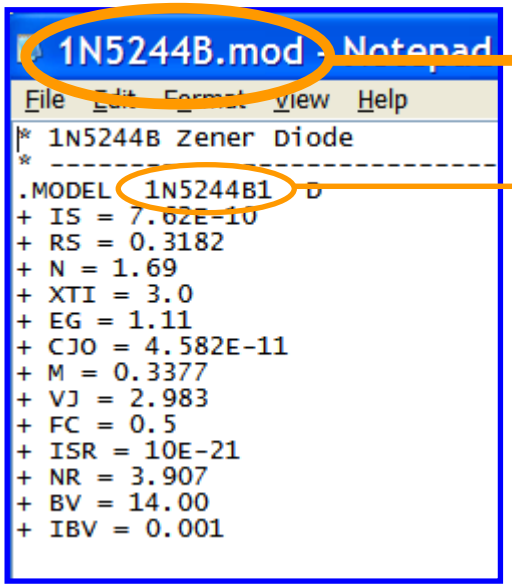
The following items are CRITICAL!

1.) The file name in the .include statement must match the spice model file name identically (except for case)! The file name syntax can be anything, just make sure that all of the characters match.

2.) The model name in the spice model file must match the device name in the LTspice schematic identically! The model name syntax can be anything, just make sure that all of the characters match.

Importing Third-Party Spice Models

Spice Model Example #1:

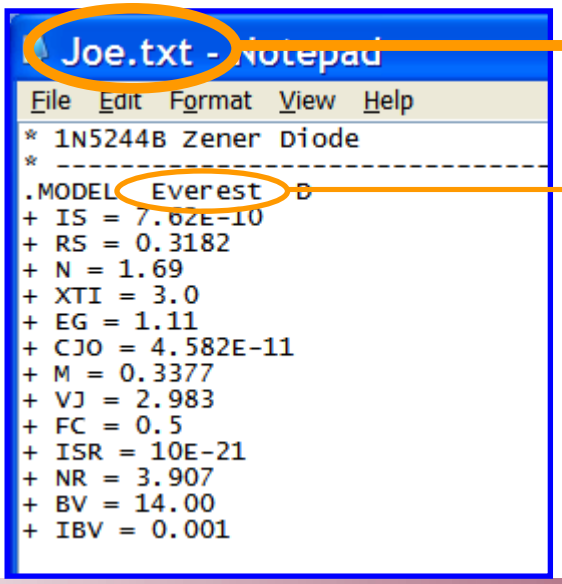


File name = 1N5244B.mod

Model name = 1N5244B1

Summary: The file and model names are irrelevant. Just make sure that the LTspice simulation device name and .include file name match those of the spice model file.

Spice Model Example #2:



File name = Joe.txt

Model name = Everest

Importing Third-Party Spice Models

Hands-on Exercise:

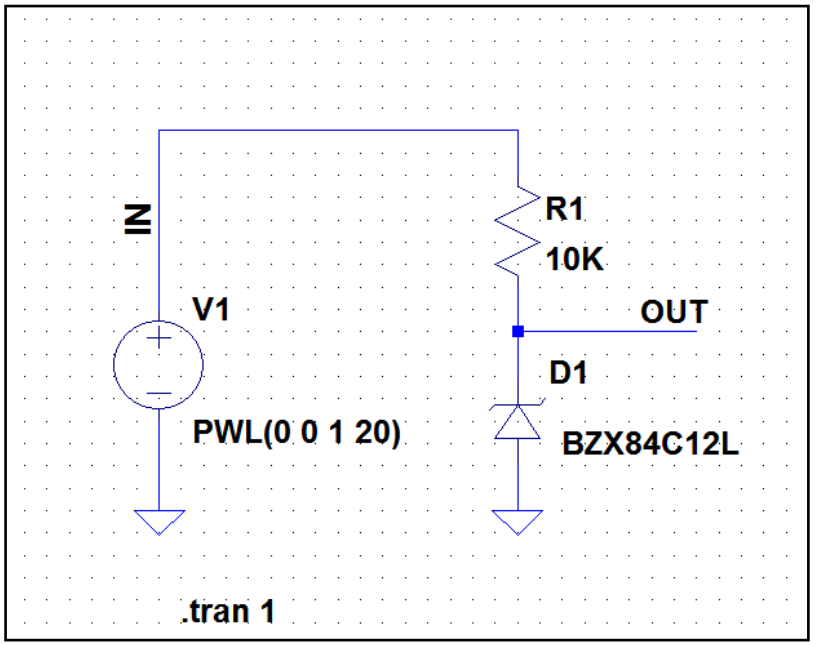
1.) Navigate to the LTspice Basic Lab Class folder

2.) Open up the simulation file titled “ZenerImportExample.asc”

3.) Open up the SPICE model file titled “1N5244B.txt” and note the device model name.

4.) Modify the simulation file so that it uses the 1N5244B third-party SPICE model based on the instructions provide on the previous slides

5.) Run the simulation and probe the IN and OUT nodes



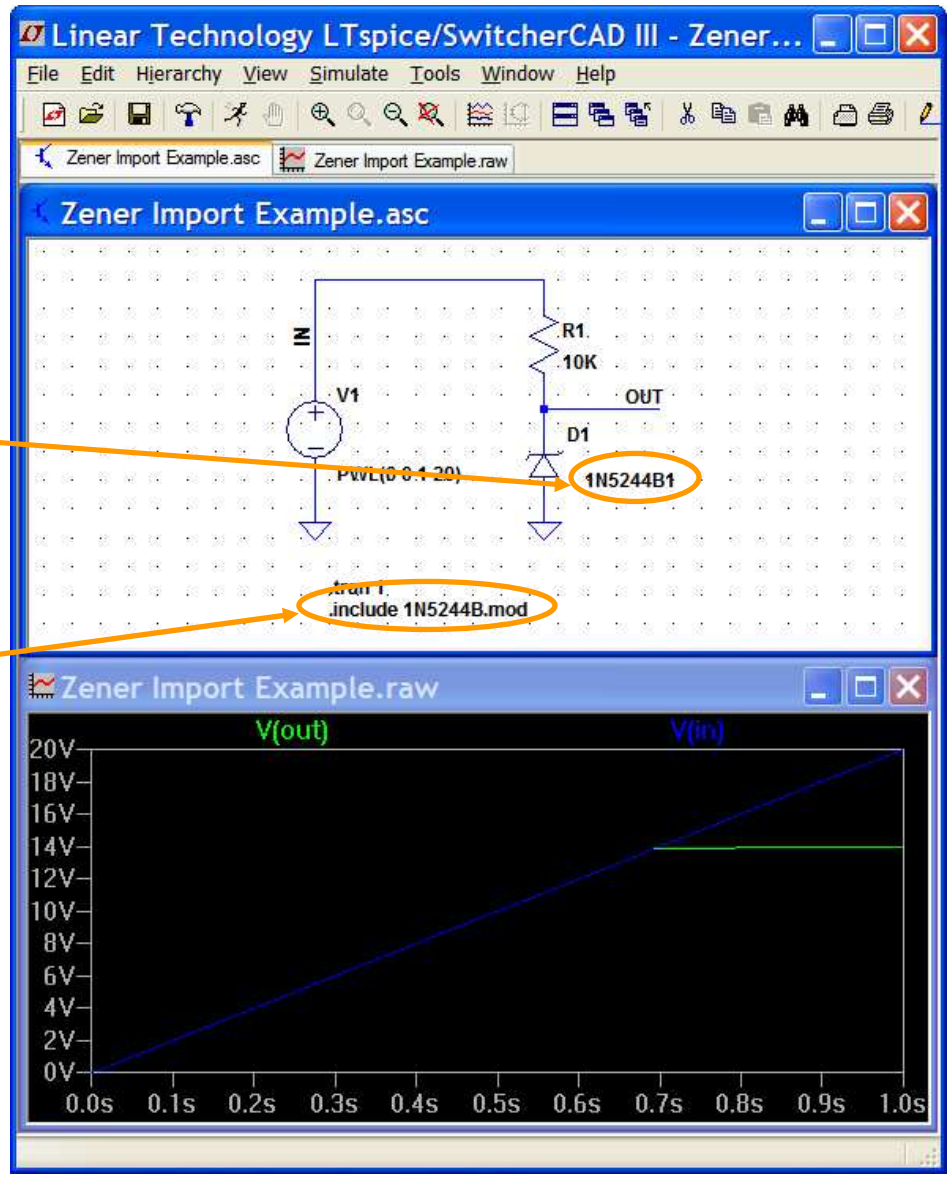
Importing Third-Party Spice Models

Solution:

1.) Zener name changed to 1N5244B1 to match model name in the SPICE model file. Right-Click on the diode name text to change.

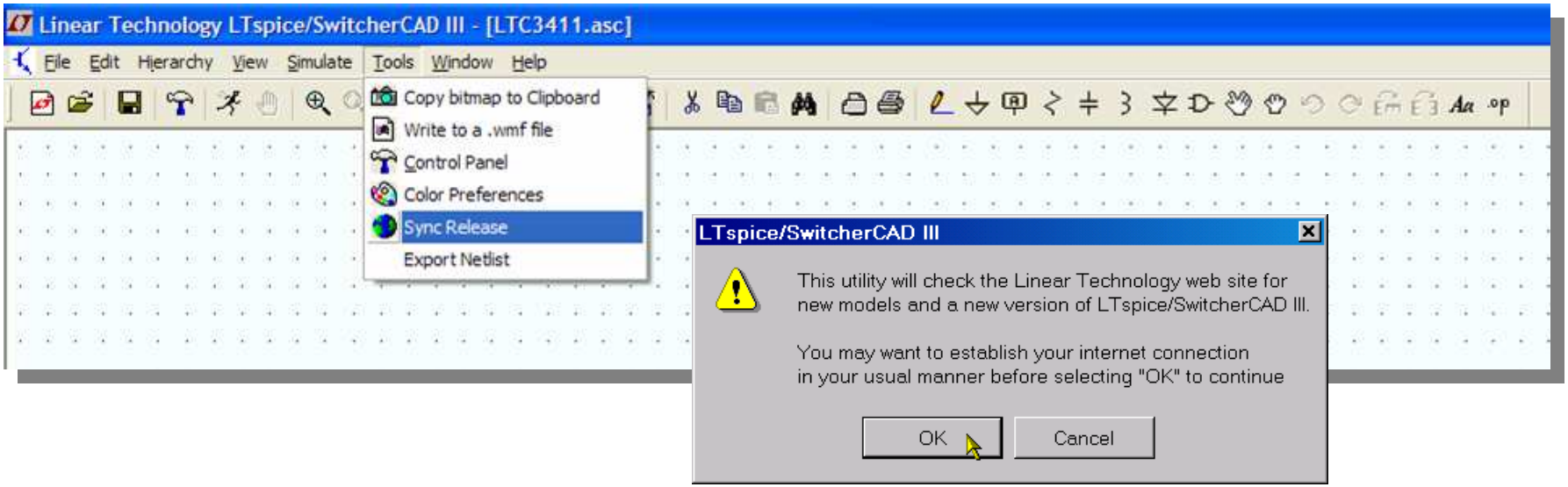
2.) .include SPICE directive added to link to the SPICE model file. Use the Edit pulldown menu ---> Spice Directive to add this SPICE directive to your simulation.

3.) Result after clicking on the Running Person symbol on the toolbar and probing the IN and OUT nodes.



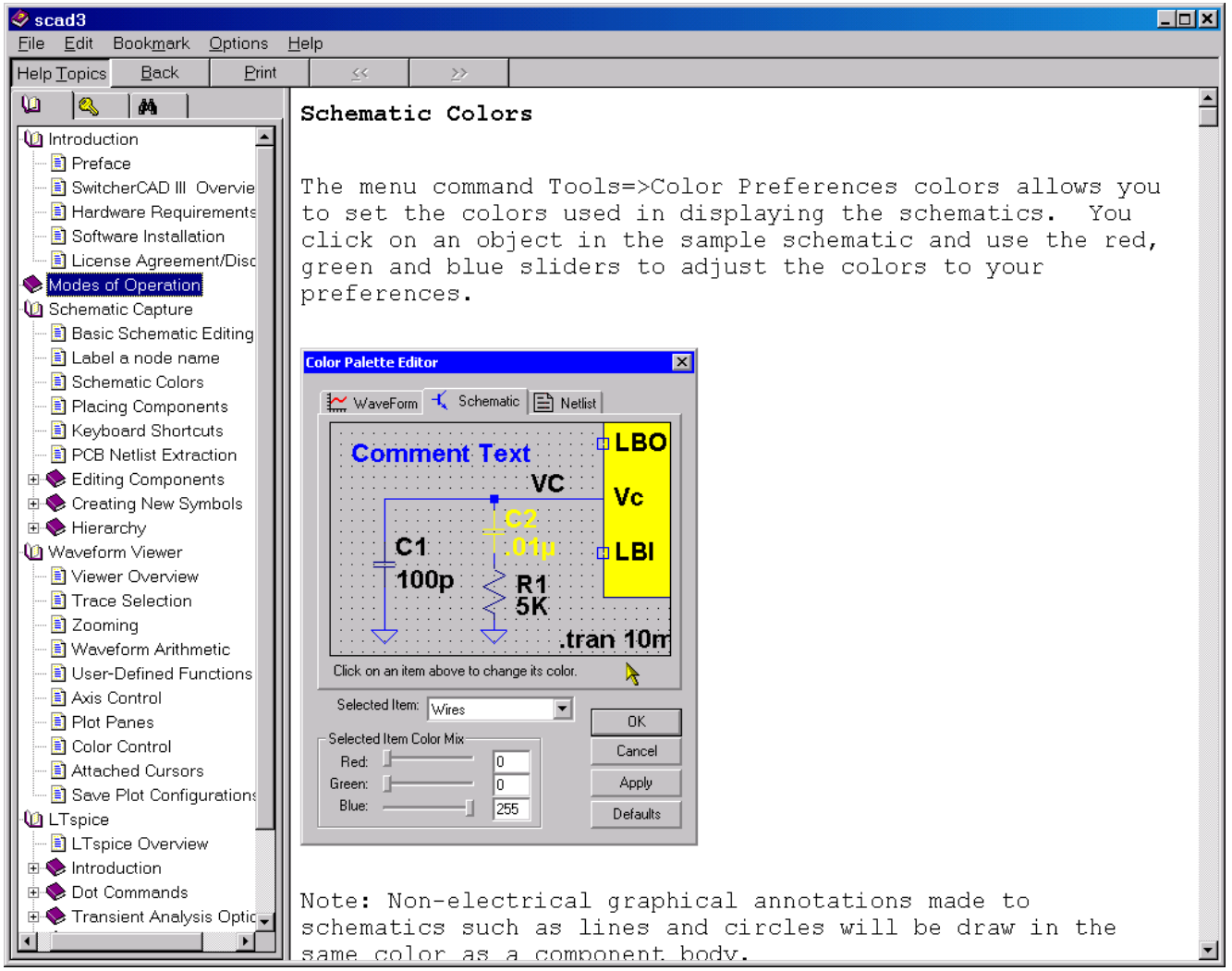
More Information and Support

Reminder to Periodically Synch Release



- It is important to sync your release of LTspice once a month to get the latest updates
  - Software update and bug fix
  - Models
  - Sample circuits and examples
- Vista users
  - You must “Run as administrator” scad.exe or its shortcut even if you are logged in as an administrator

Built-in Help System



PDF User's Guide

- Download the PDF User's Guide Manual:
  - http://LTspice.linear.com/software/scad3.pdf

Appendix A – Summary of Special Mouse and Keyboard Commands

Schematic-Based Special Commands:

1. Alt-Left-Click on a wire
   - This will display the waveform for the current flowing in the wire
2. Alt-Left-Click on a component
   - This will display the instantaneous power dissipation in the component
3. Ctrl-Right-Click on a component
   - Allows you to edit embedded component attributes

Waveform-Based Special Commands:

1. Ctrl-Left-Click on a waveform title
   - Displays the average and RMS values for the waveform
2. Left-Click on node and drag to another node
   - Displays differential voltage

Appendix B – Summary of Additional Features

1. To pause a simulation:
   - “Simulate” pull down menu ---> Pause
   - There is no toolbar button for this function
2. To zoom in/out using the schematic editor:
   - Just use the wheel on your mouse
3. To pan around a schematic
   - Left-Click the mouse and hold, then drag
   - Tilt wheel to move right and left

Thank you for attending, and happy simulating!

Homework: Once you return to the office, go back over the training materials within a week!