Introduction to LTspice IV Workshop

Presented by:

Steve Knudtsen
FAE Linear Technology
sknudtsen@linear.com

Why Use LTspice?

- Stable SPICE circuit simulation with
  - Unlimited number of nodes
  - Schematic/symbol editor
  - Waveform viewer
  - Library of passive devices

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

- Fast simulation of switch mode power supplies
  - Steady state detection
  - Turn on transient
  - Step response
  - Efficiency / power computations

SPICE = Simulation Program with Integrated Circuit Emphasis

- Advanced analysis and simulation options
  - Not covered in this lab class
- Outperforms or as powerful as pay-for tools

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 on your computer
- Standalone application that runs on your computer
- Users Guide and Demo Circuit collection also available

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?

1. Use one of the 100's of demo circuit available on linear.com
   - Designed and Reviewed by Factory Apps Group
   - Go to http://www.linear.com/software
2. 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
3. Use simulation circuits posted on the LTspice Yahoo! User's Group. URL = http://tech.groups.yahoo.com/group/LTspice
   - Also contains many very helpful discussion threads and tutorials
4. Use the schematic editor to create your own design
   - LTspice contains models for most LTC power devices and many more

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?

1. 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
2. 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
3. Use simulation circuits posted on the LTspice Yahoo! User's Group. URL = http://tech.groups.yahoo.com/group/LTspice
   - Also contains many very helpful discussion threads and tutorials
4. Use the schematic editor to create your own design
   - LTspice contains models for most LTC power devices and many more

Demo Circuits on linear.com

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

LTSPICE IV

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.

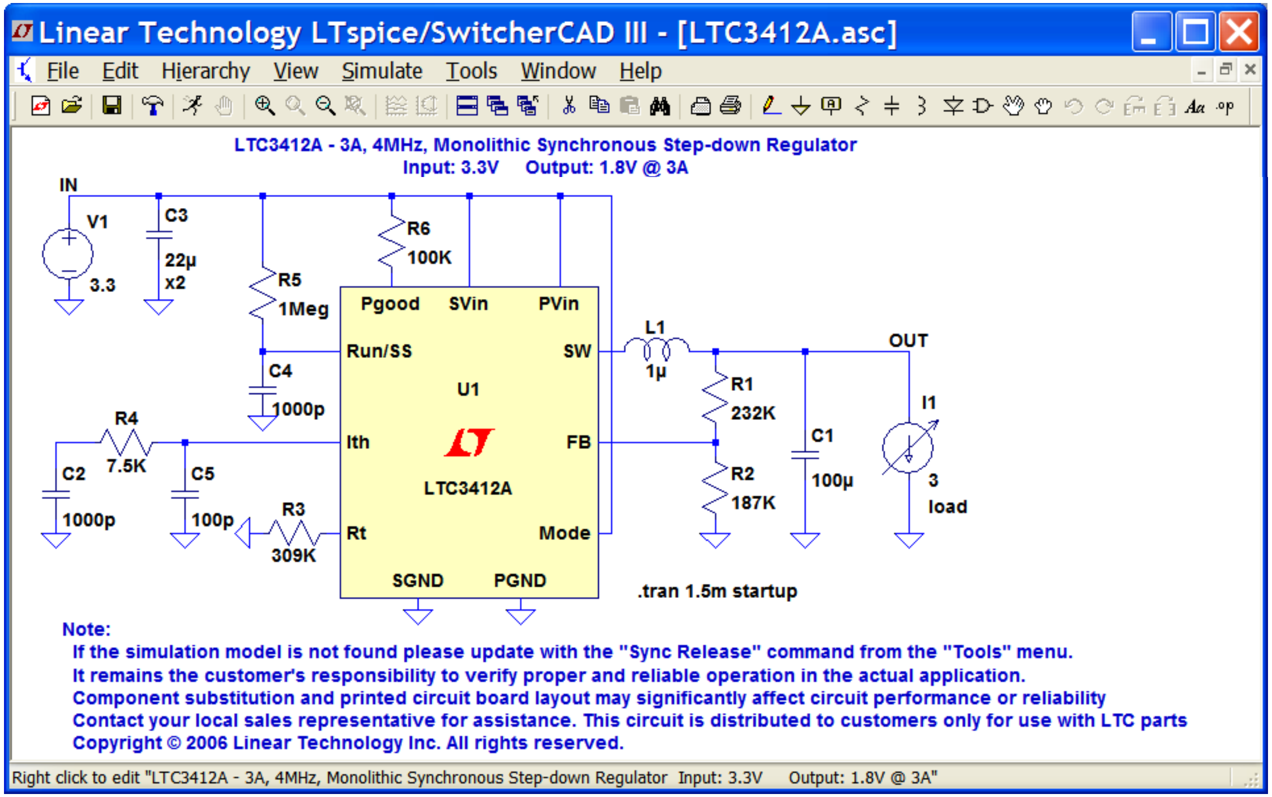
- Download LTspice IV (Updated January 23, 2009)
- LTspice Users Guide
- LTspice Getting Started Guide
- LTspice Demo Circuit Collection

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 on linear.com

✓ 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

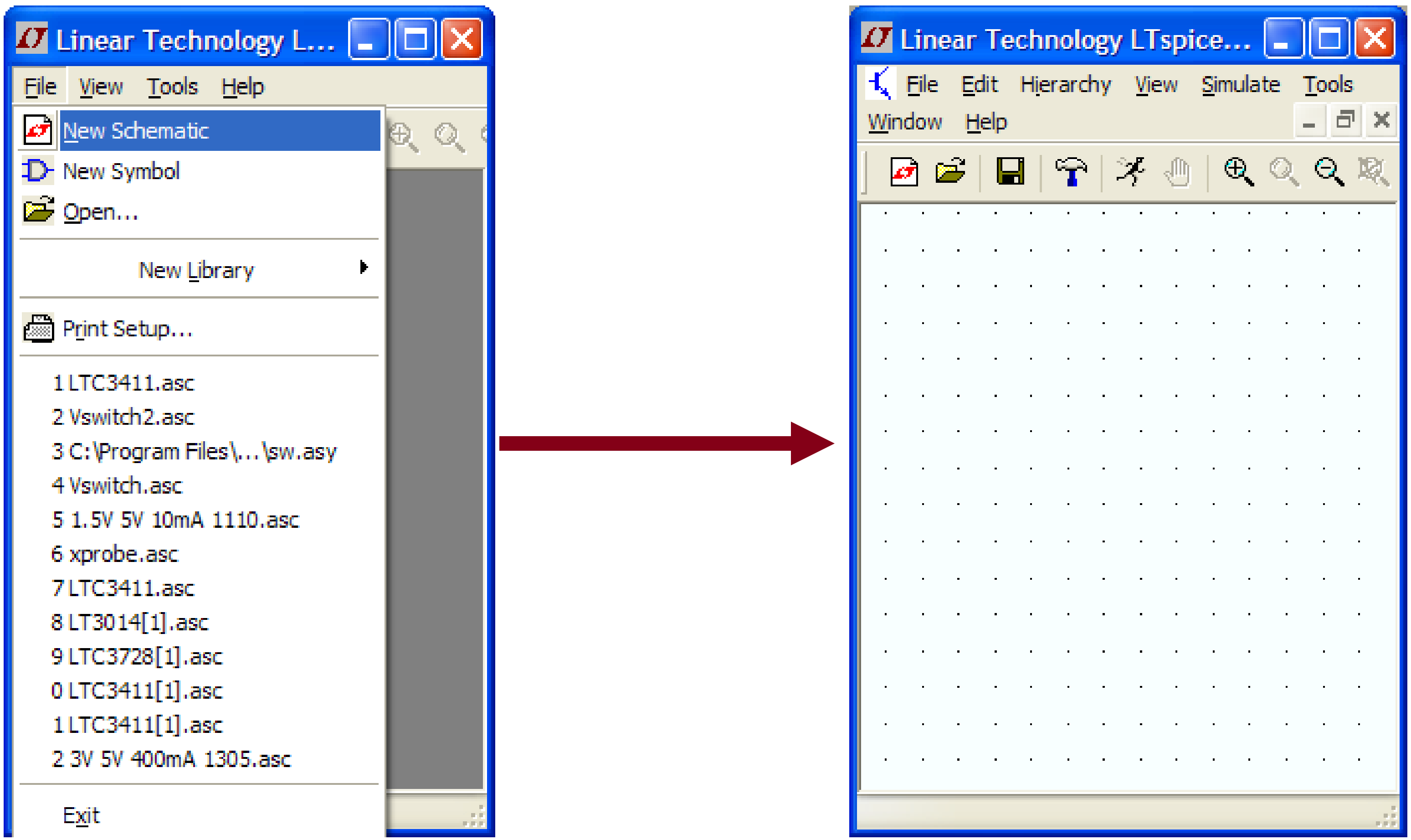


How Do I Get Started using LTspice?

1. 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
2. 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
3. Use simulation circuits posted on the LTspice Yahoo! User's Group. URL = http://tech.groups.yahoo.com/group/LTspice
   - Also contains many very helpful discussion threads and tutorials
4. Use the schematic editor to create your own design
   - LTspice contains models for most LTC power devices and many more

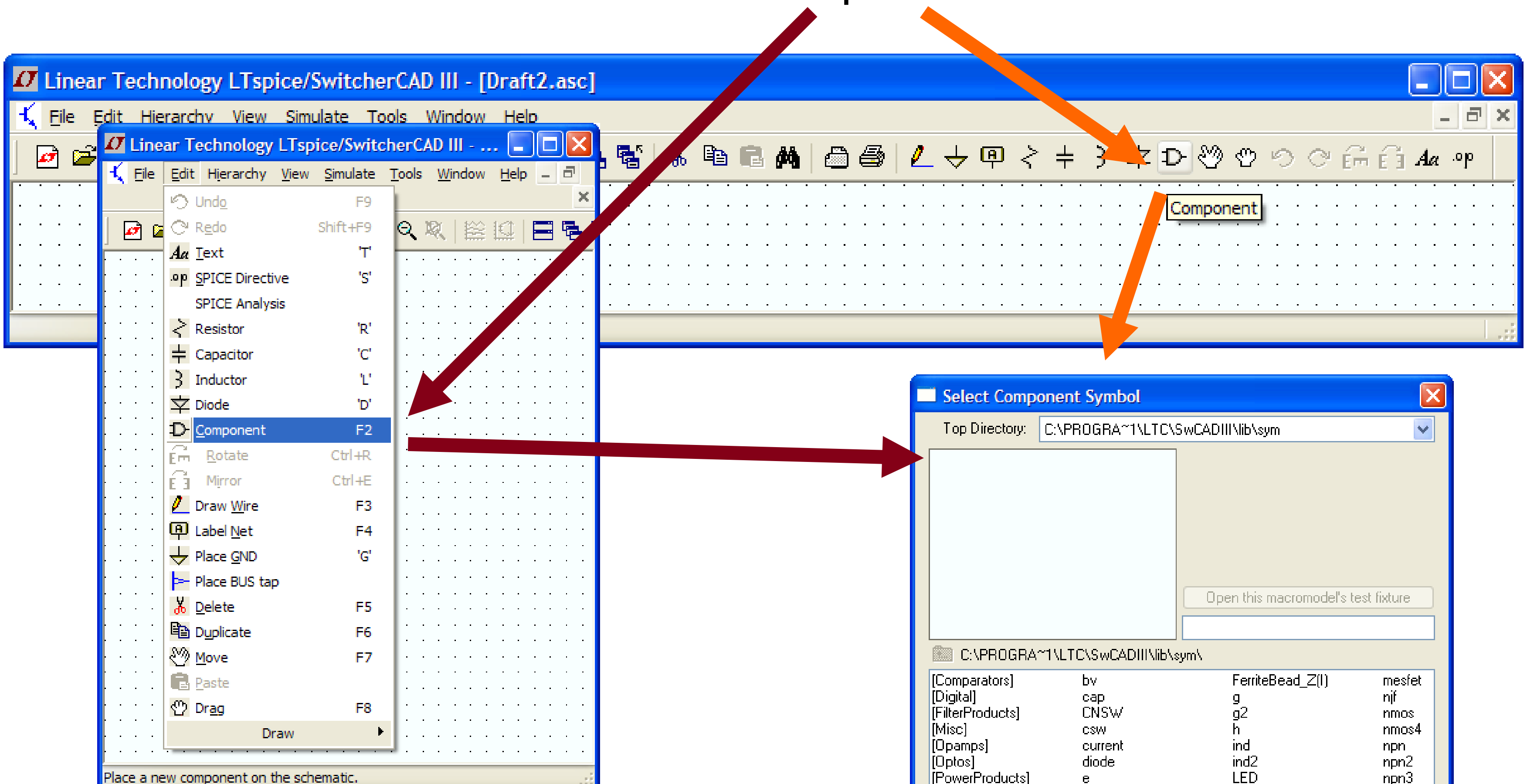
Start with a New Schematic

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



Add a Component

- Use Add a Component or F2



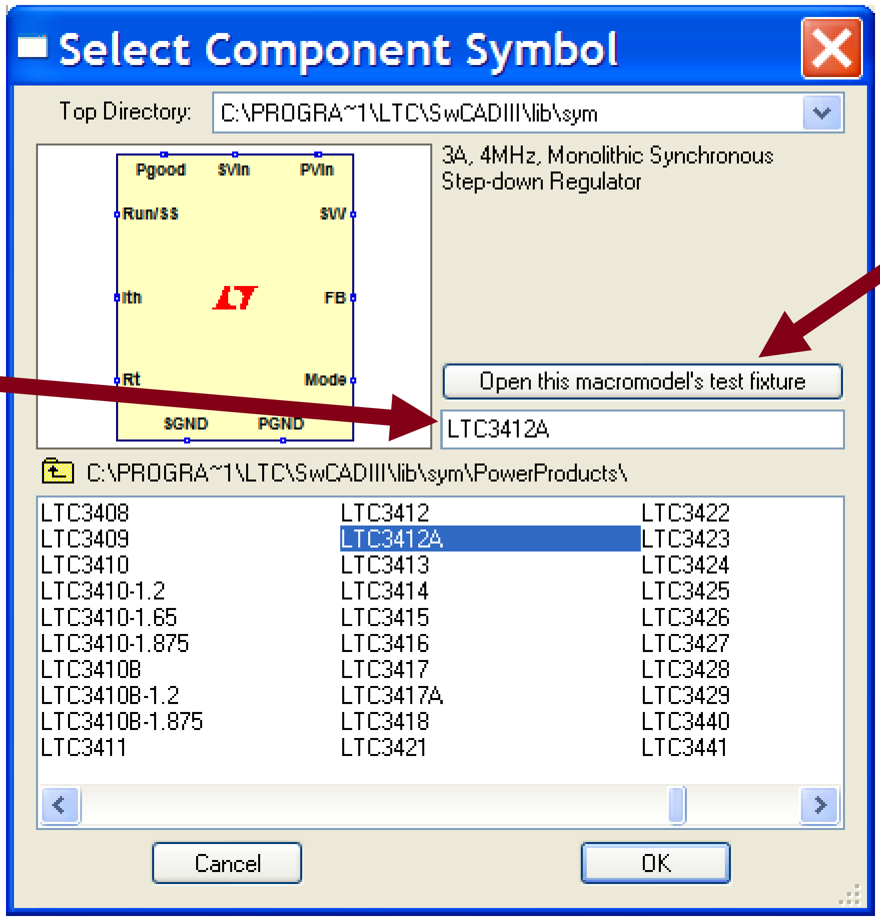
Take a moment to review all of the components!

Selecting a Model & Opening Test Fixture

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

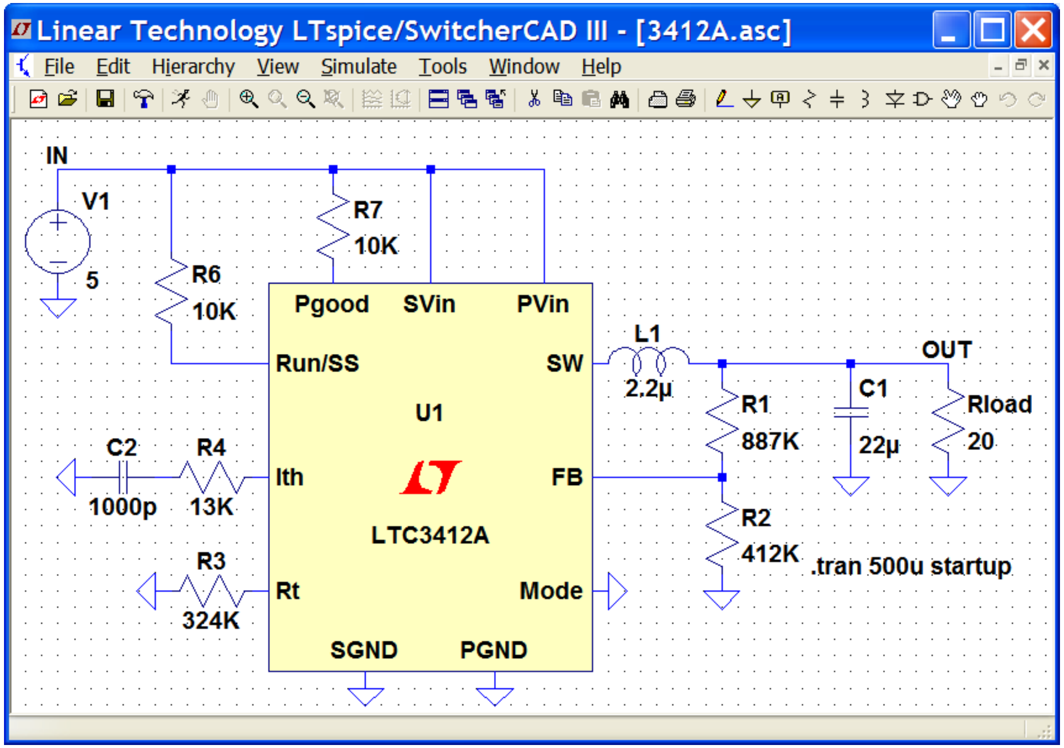
2. Select

1. Enter 3412A



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

How Do I Get Started using LTspice?

1. 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
2. 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
3. Use simulation circuits posted on the LTspice Yahoo! User's Group. URL = http://tech.groups.yahoo.com/group/LTspice
   - Also contains many very helpful discussion threads and tutorials
4. Use the schematic editor to create your own design
   - LTspice contains models for most LTC power devices and many more

LTspice Yahoo! User's Group Web Page

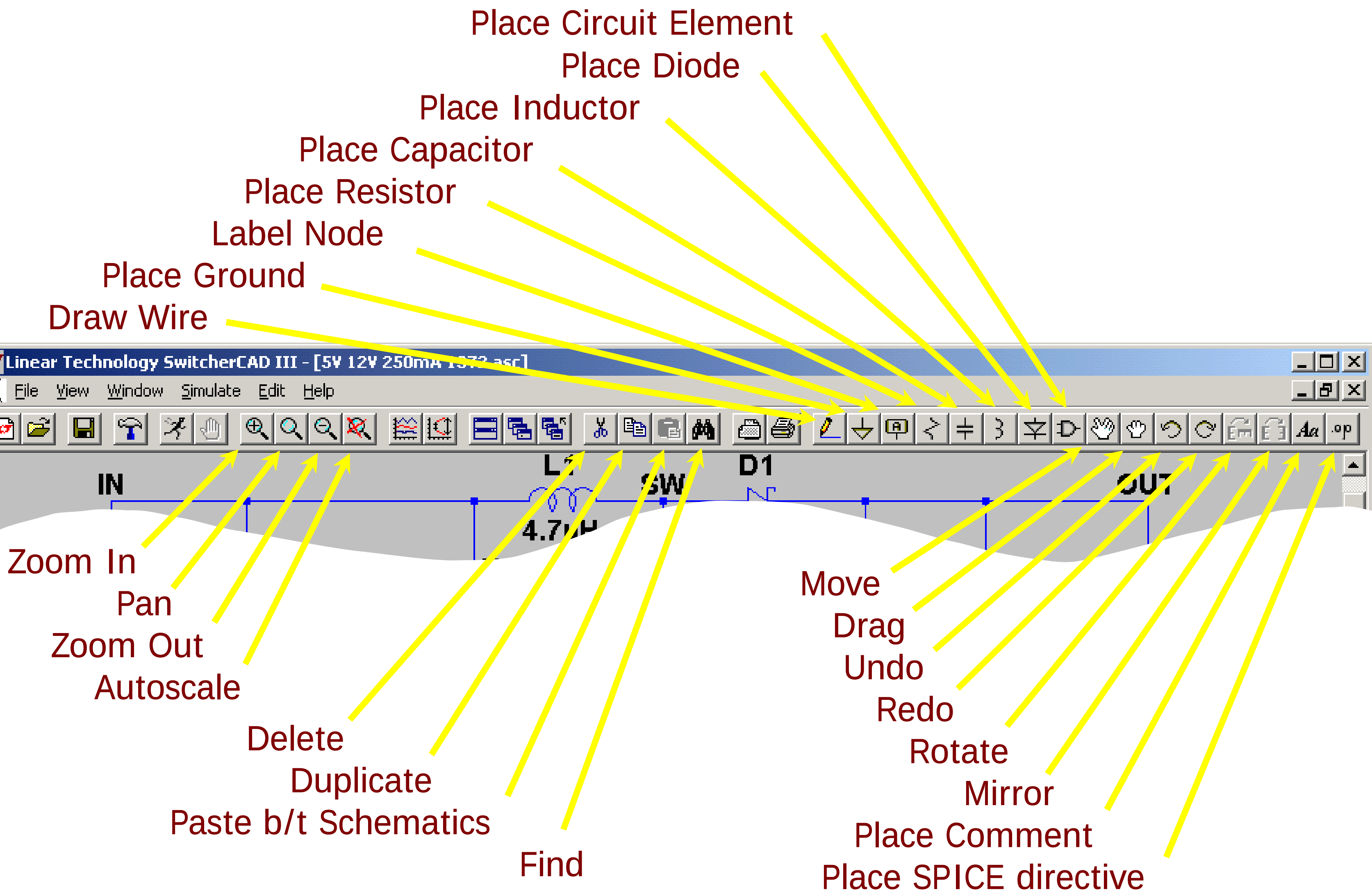
URL

Join the group here. As of May 11 2009, there are 17,630 members!

How Do I Get Started using LTspice?

1. 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
2. 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
3. Use simulation circuits posted on the LTspice Yahoo! User's Group. URL = http://tech.groups.yahoo.com/group/LTspice
   - Also contains many very helpful discussion threads and tutorials
4. Use the schematic editor to create your own design
   - LTspice contains models for most LTC power devices and many more

Schematic Editing



Using Labels to Specify Units for Component Attributes

(industry standard SPICE unit labels)

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

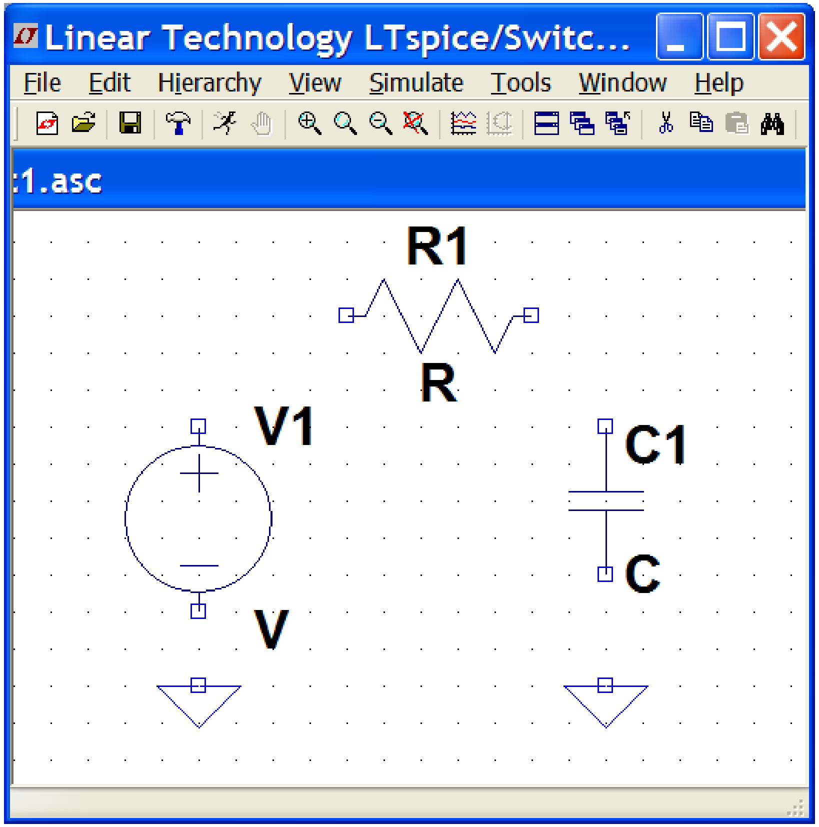
- M = m = milli = 10-3
- U = u = micro = 10-6
- N = n = nano = 10-9
- P = p = pico = 10-12
- F = f = femto = 10-15

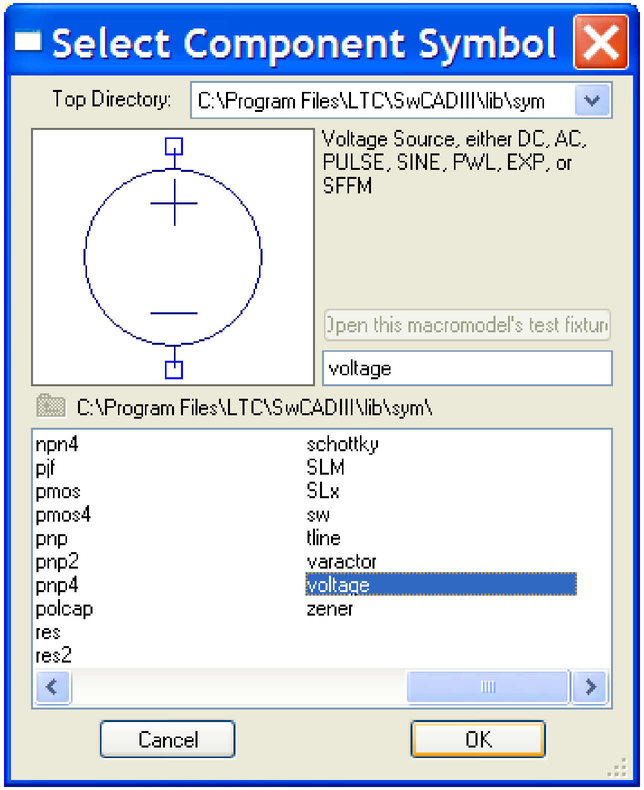
Hints

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

Wiring up a Simple RC Circuit

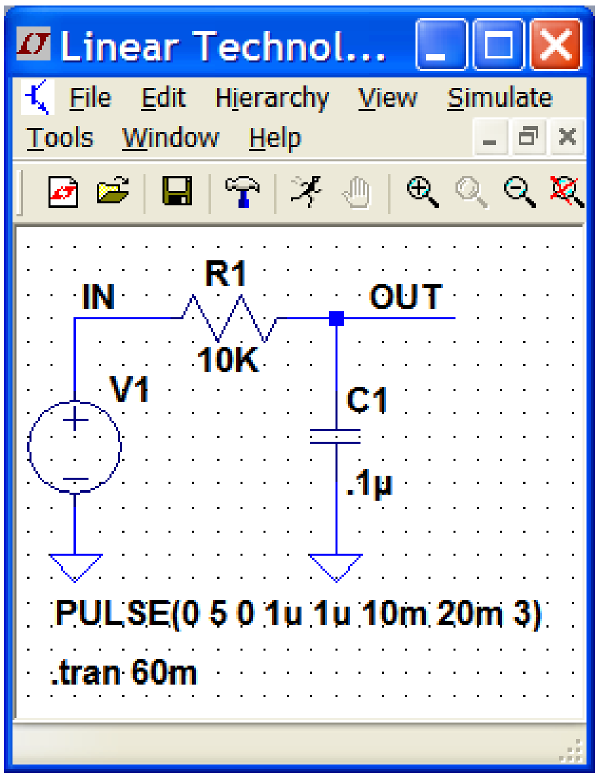
- Using the toolbar, select New Schematic
- Using the toolbar, select a Resistor, Capacitor and Ground. Place these on the schematic as shown below.
- Using the toolbar, select Component. From the component window, type “voltage” in the dialog box, and click “OK” to place a voltage source

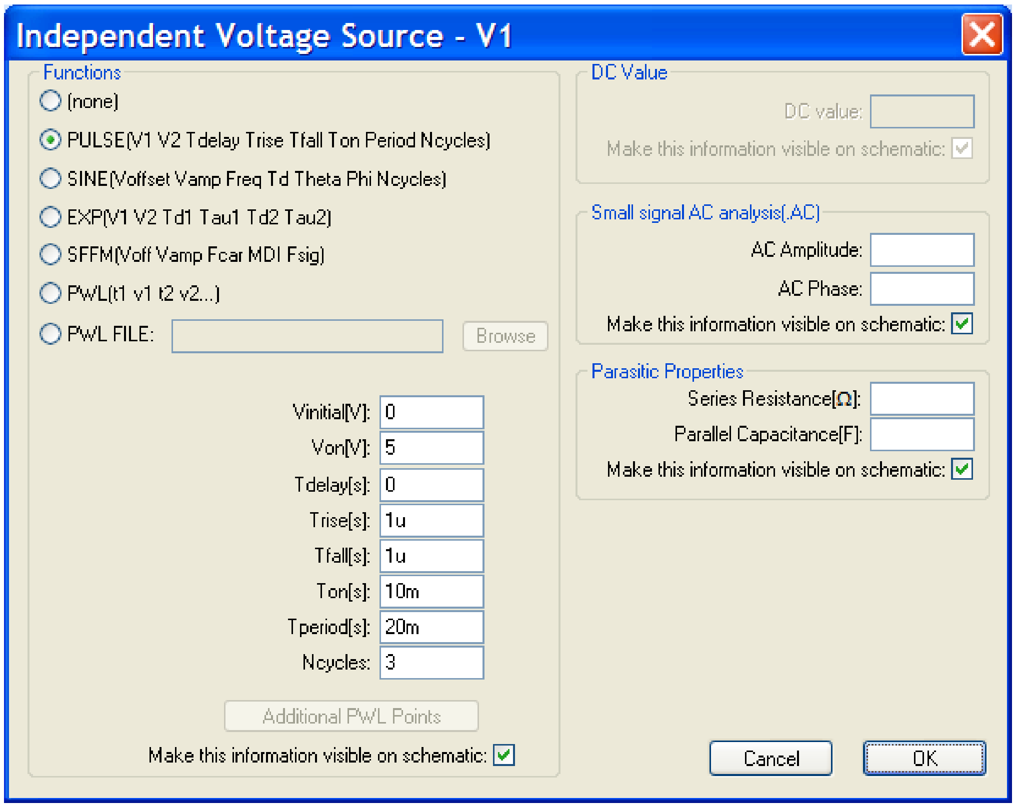




Wiring up a Simple RC Circuit

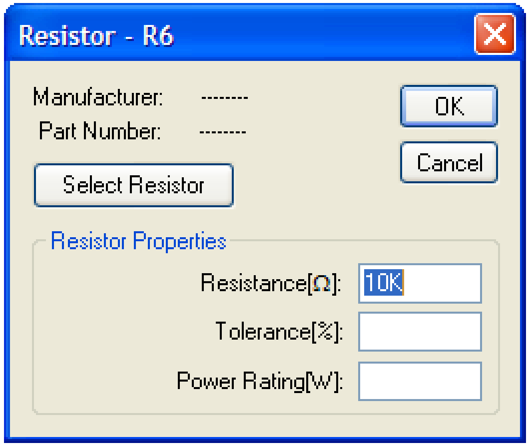
- Using the toolbar, select Wire. Wire up the RC circuit as shown below.
- Using the toolbar, select Label Net. Label the input/output nodes as shown below
- Right-click on each component to change its value as shown below
- Right-click on the voltage source and enter the parameters shown below under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

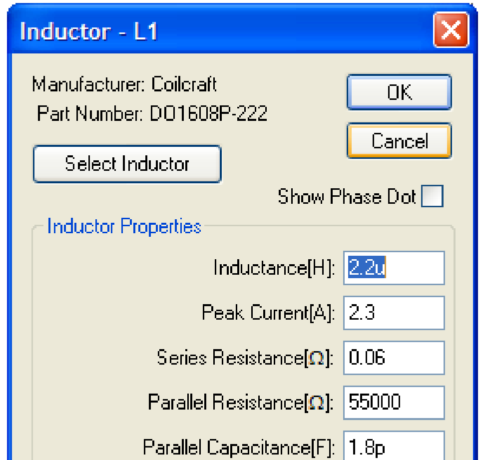


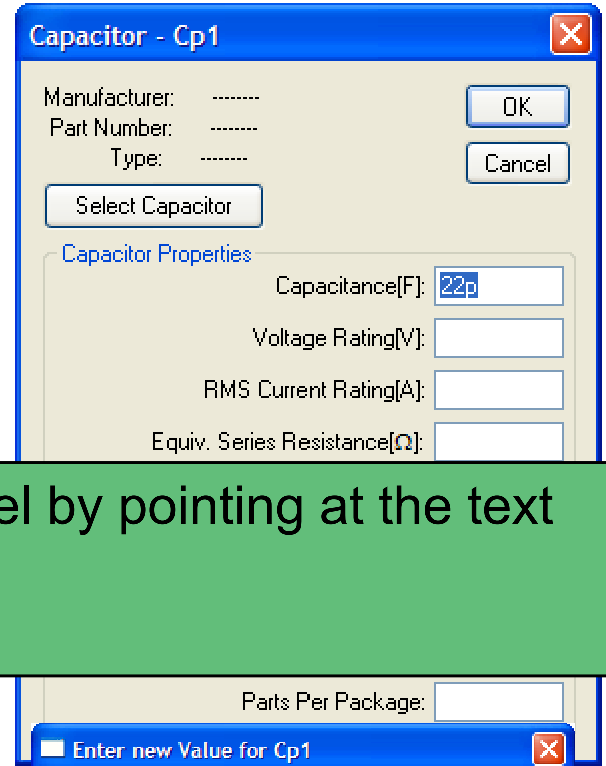


Editing Components

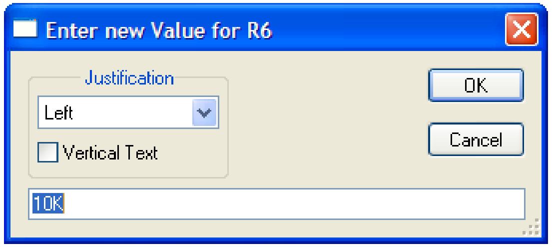
- Component attributes can be edited by pointing at the component with the mouse and Right-Clicking

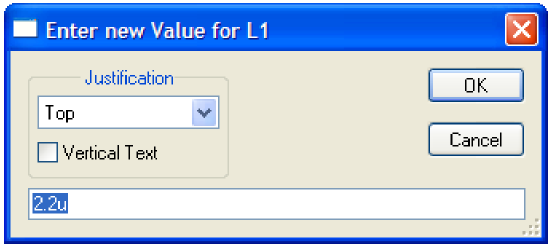


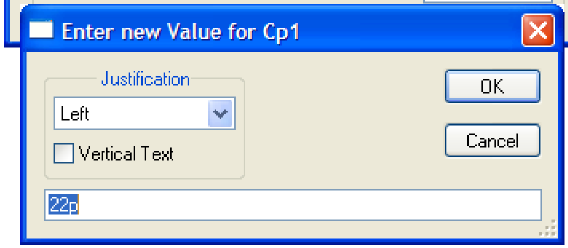




- You can also edit the visible attribute and label by pointing at the text with the mouse and then right clicking
  - Mouse cursor will turn into a text caret



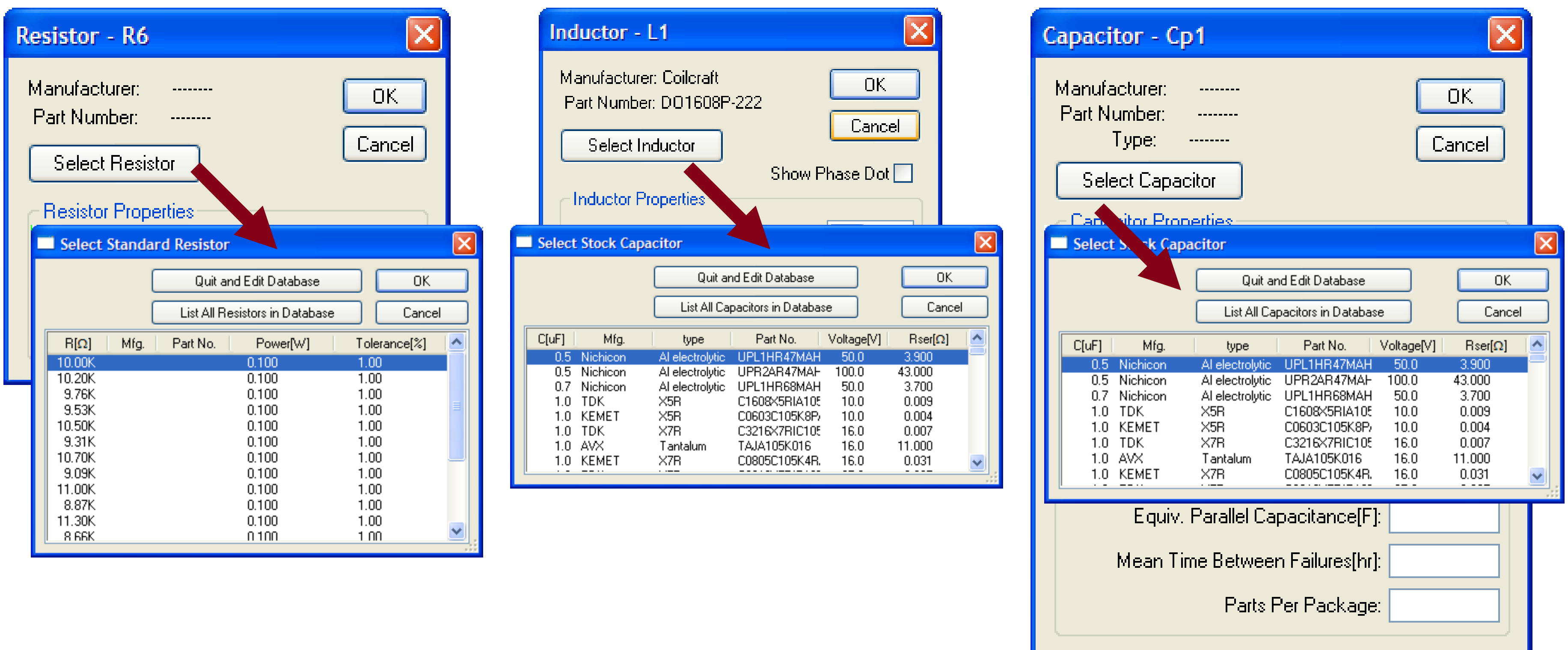




Component Database

- Components such as
  - Resistors, capacitors, inductors, diodes,
  - Bipolar transistors, MOSFET transistors, JFET transistors
  - Independent voltage and current sources

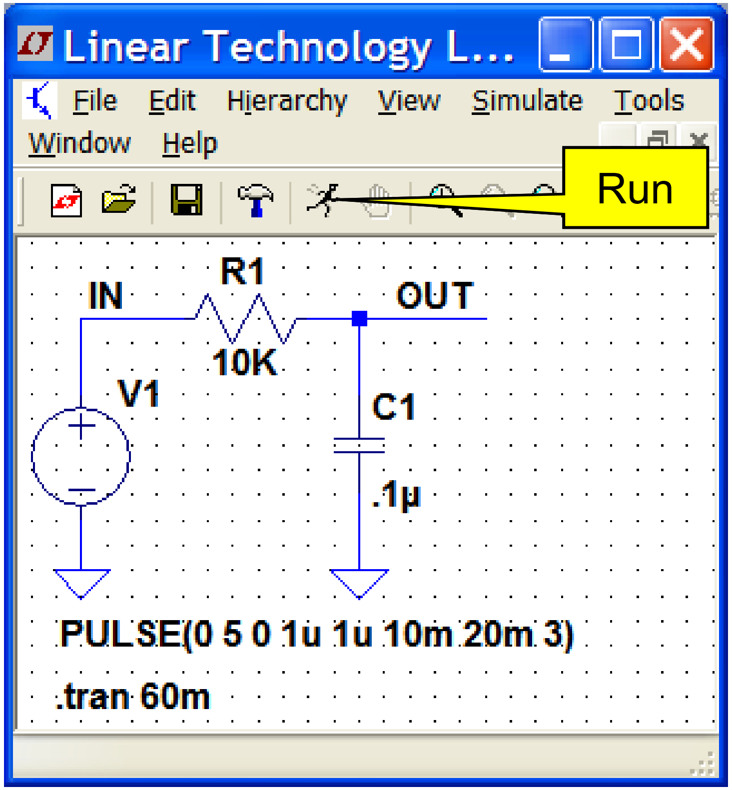
<u>You can access a database of known devices</u>

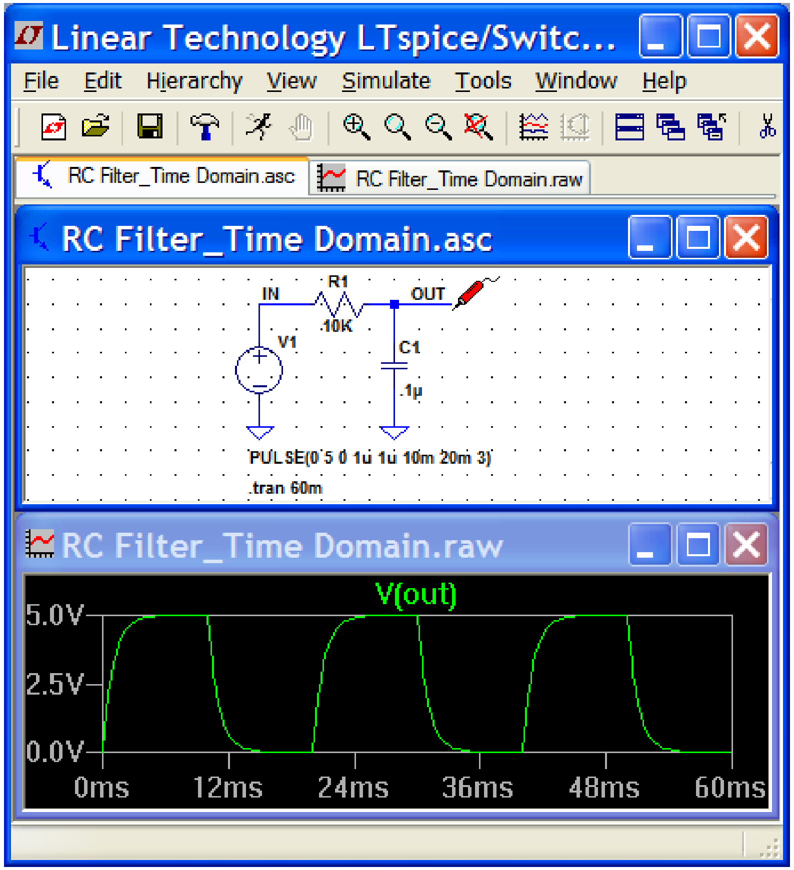


How do you run and probe a circuit in LTspice?

Running the RC Circuit Simulation

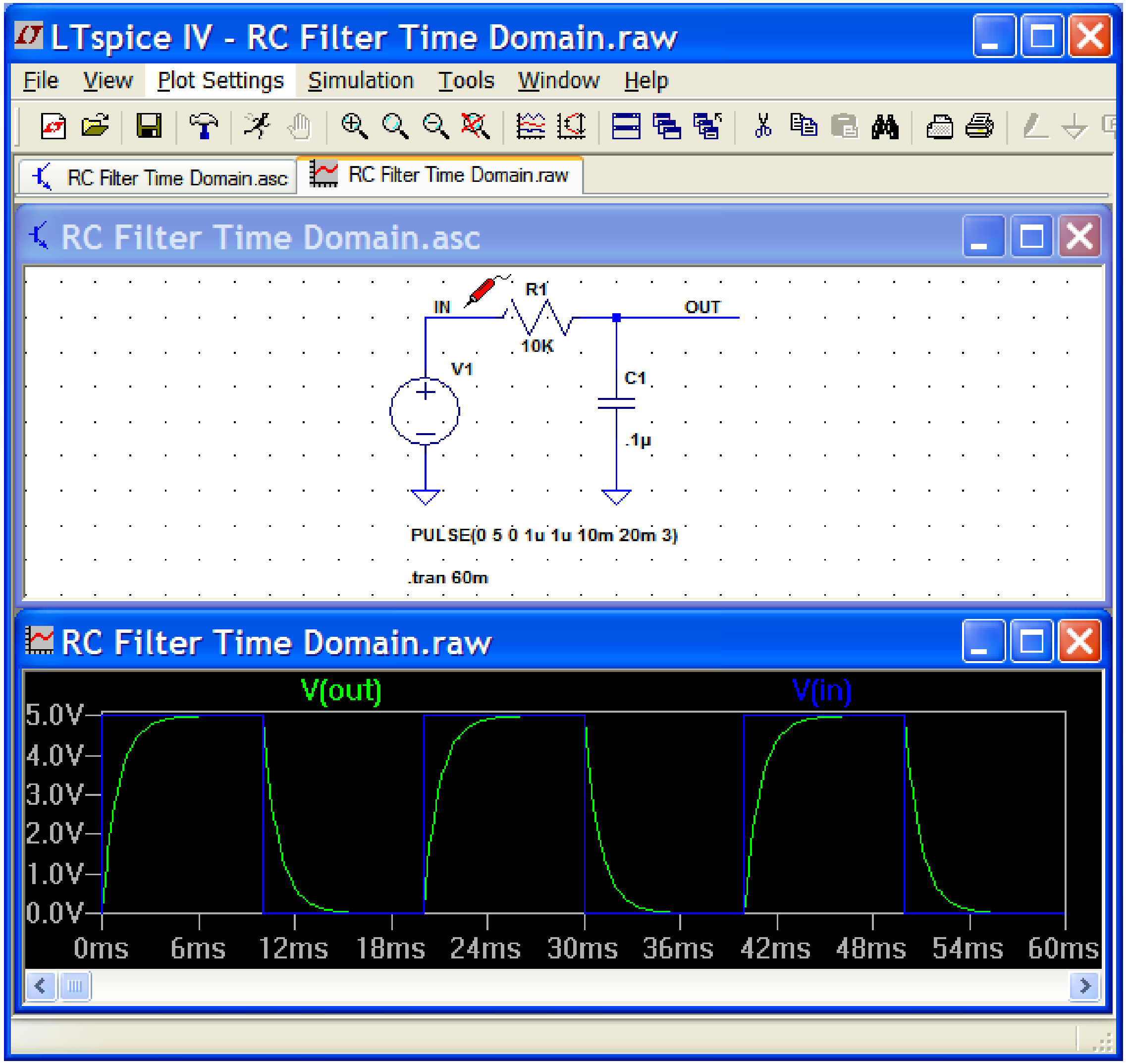
- With the RC circuit in the active window, click on the "Running Person" button on the tool bar
- The Edit Simulation Command window will appear. Set the Stop Time for 60msec, and click "OK"
- Using the mouse, click on the "OUT" node to display the output voltage waveform





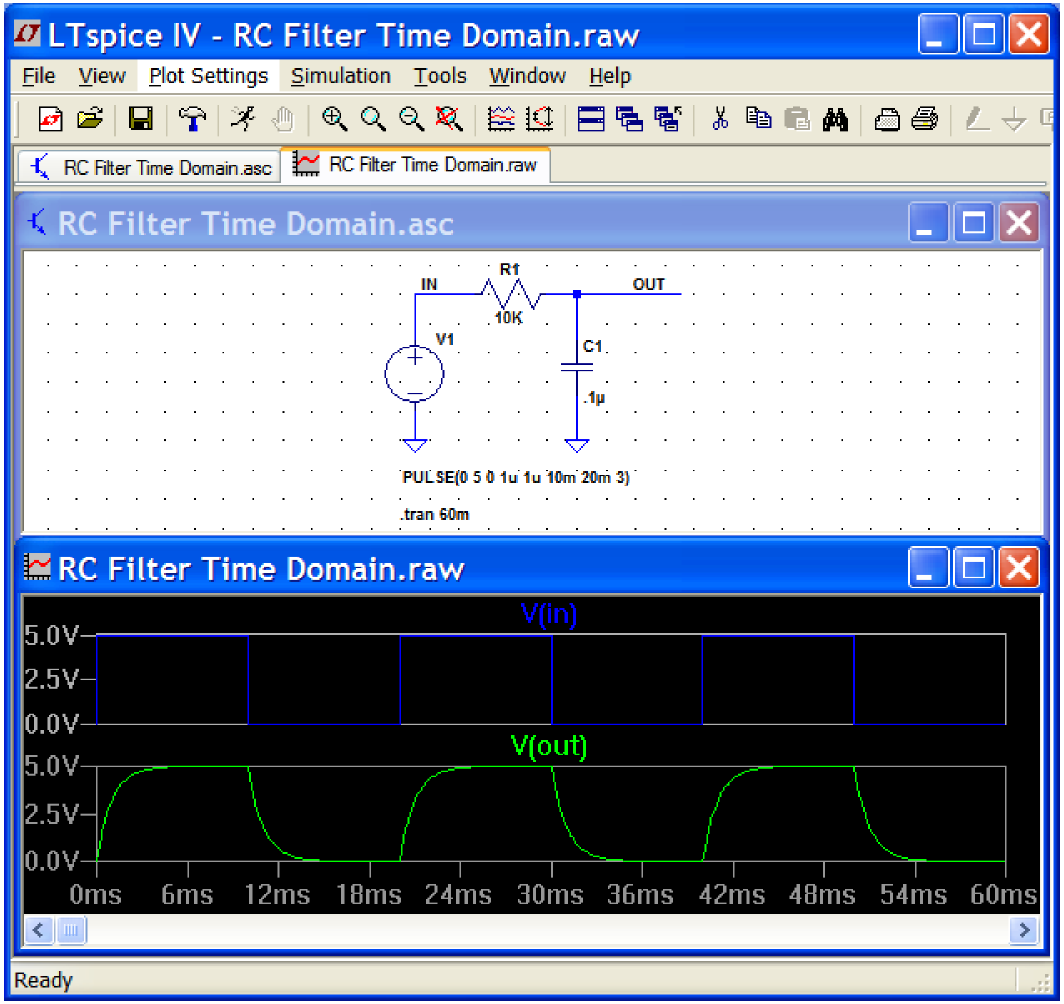
Running the RC Circuit Simulation

- Using the mouse, click on the "IN" node to display the input voltage waveform also



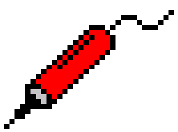
Running the RC Circuit Simulation

- Split the plot pane by selecting "Add Plot Pane" under the Plot Settings pull down menu
- Drag and drop the V(in) waveform title into the new plot pane



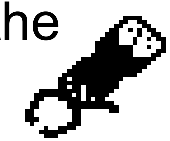
Waveform Viewer

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



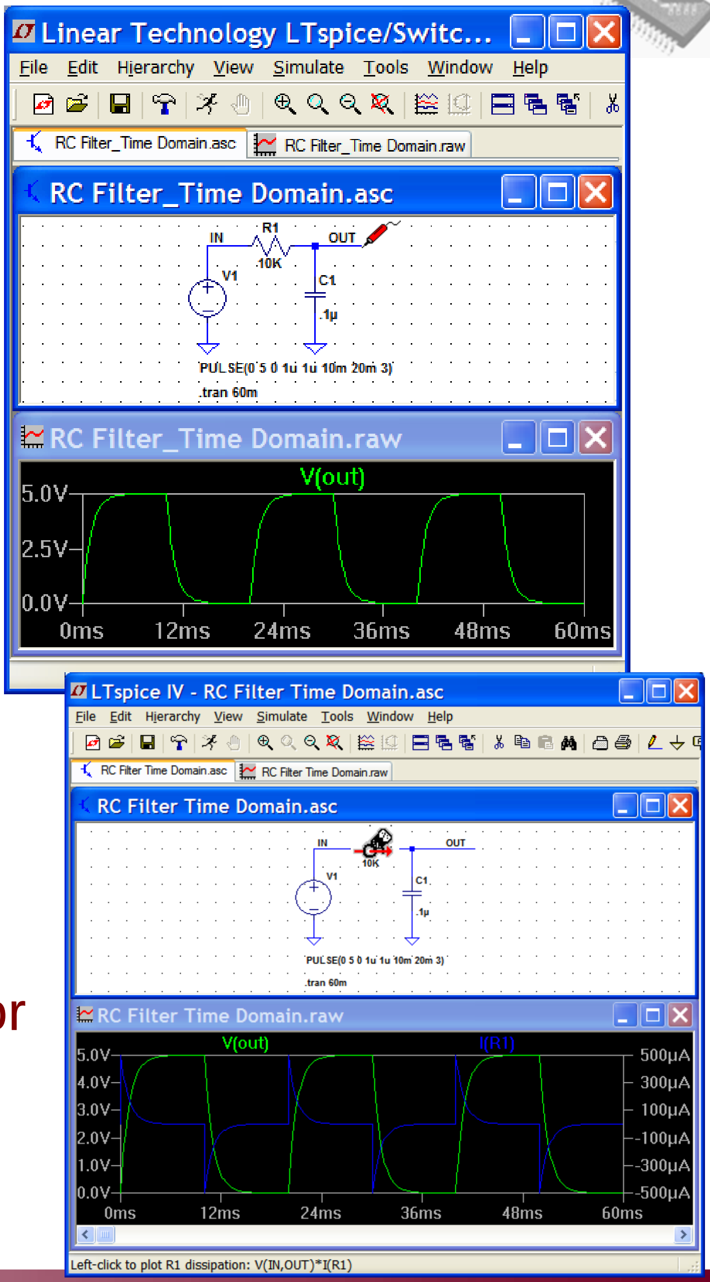
Voltage probe cursor

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



Current probe cursor

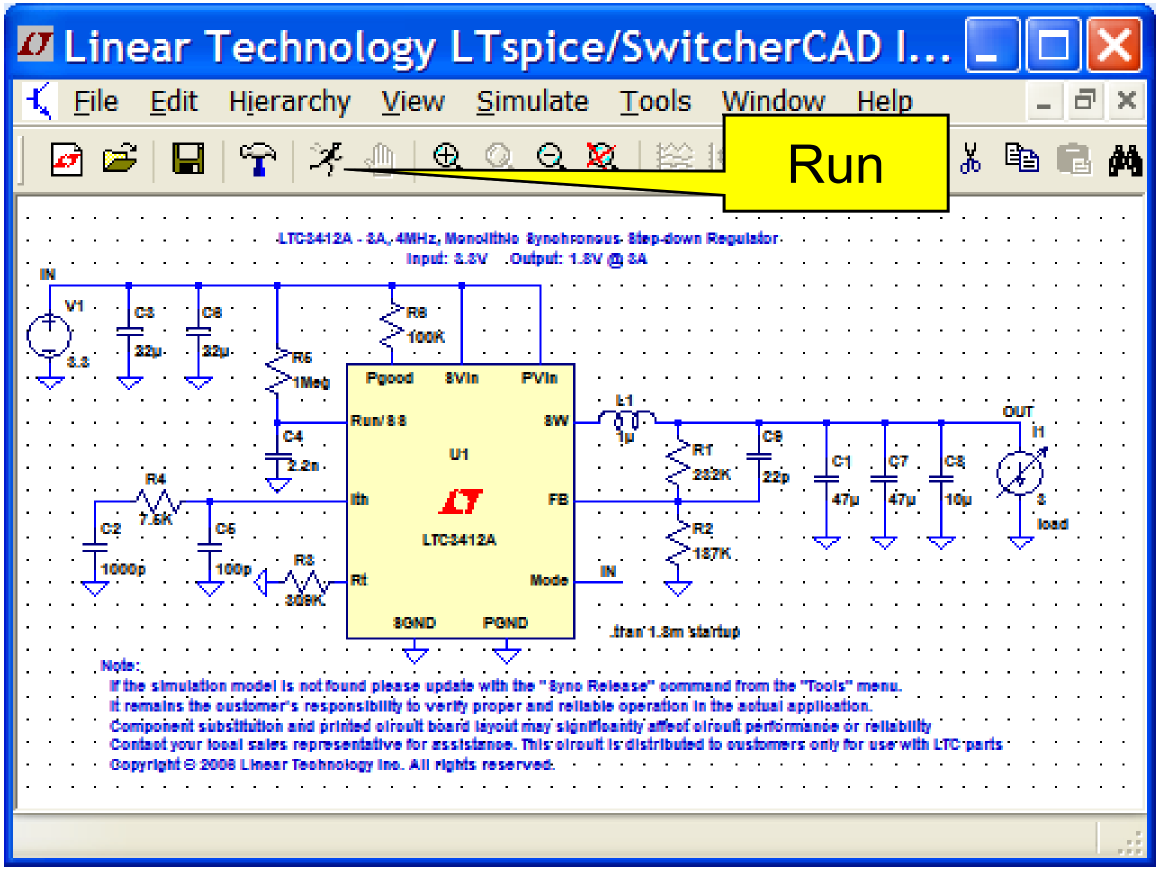
     - R, C, L
     - Convention of positive current is from netlist pin #1 to pin #2



Running a Demo Circuit

- Access the LTC3412A demo circuit located in the “LTspice Training Files” folder on your desktop
  - Click File ---> Open, and navigate to the LTspice Training Files folder on your desktop. Look for the file titled “LTC3412A.asc”

Running a Demo Circuit

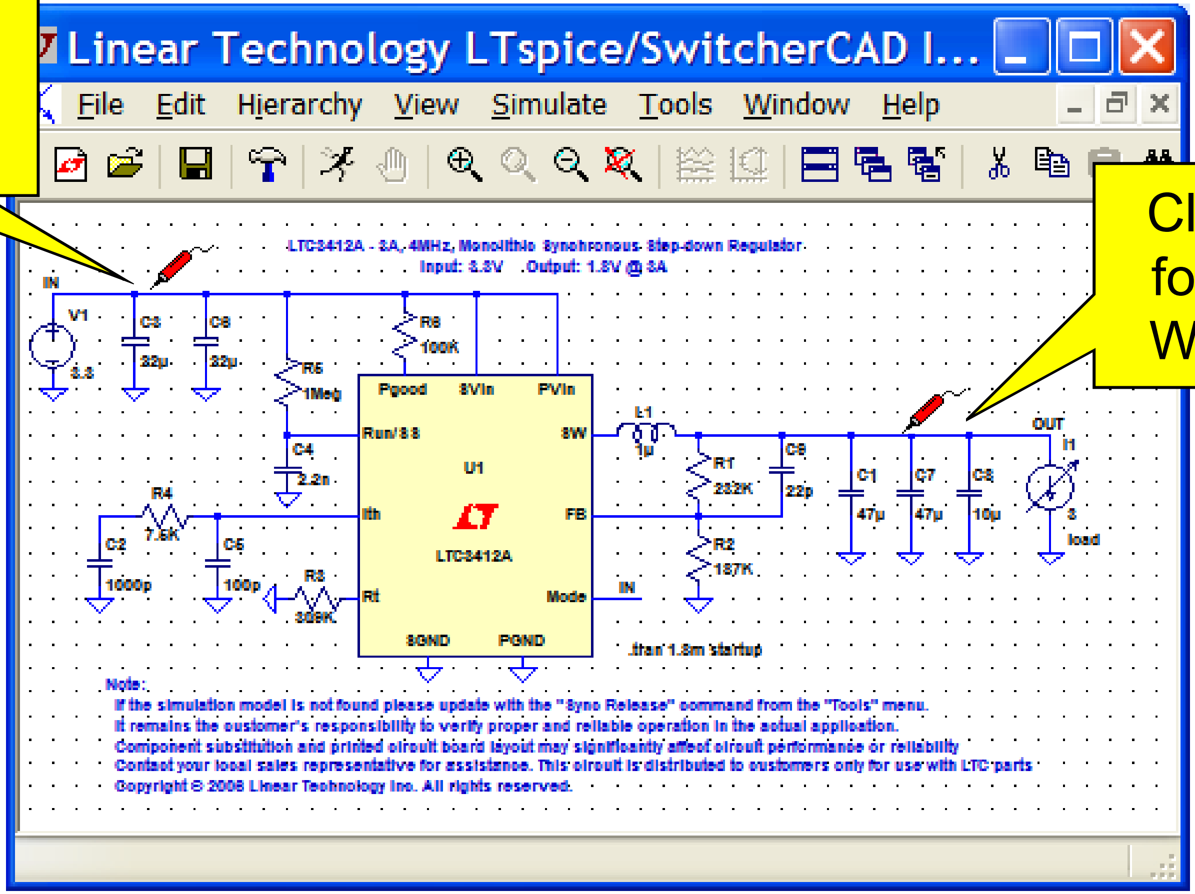


Note: A simulation can be paused by selecting "pause" under the Simulate pull-down menu.

- 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

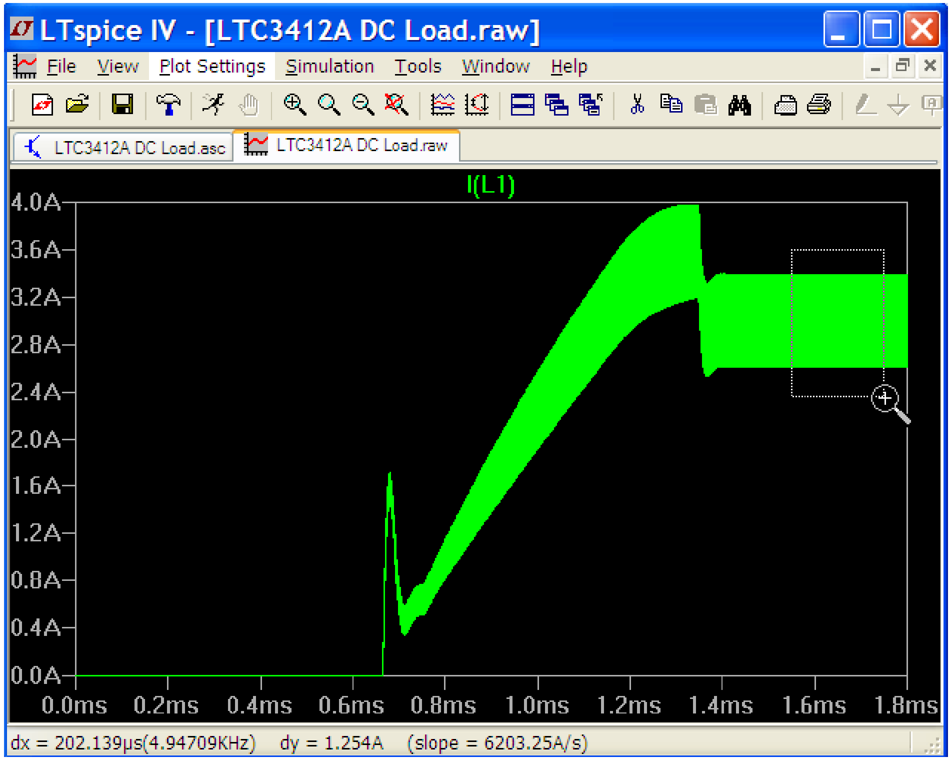
Click Here for Input Waveform



Click Here for Output Waveform

- All Demo Circuits have INs and OUTs clearly labeled to help you quickly select them
- Display the waveforms for IN and OUT by clicking on the IN and OUT nodes

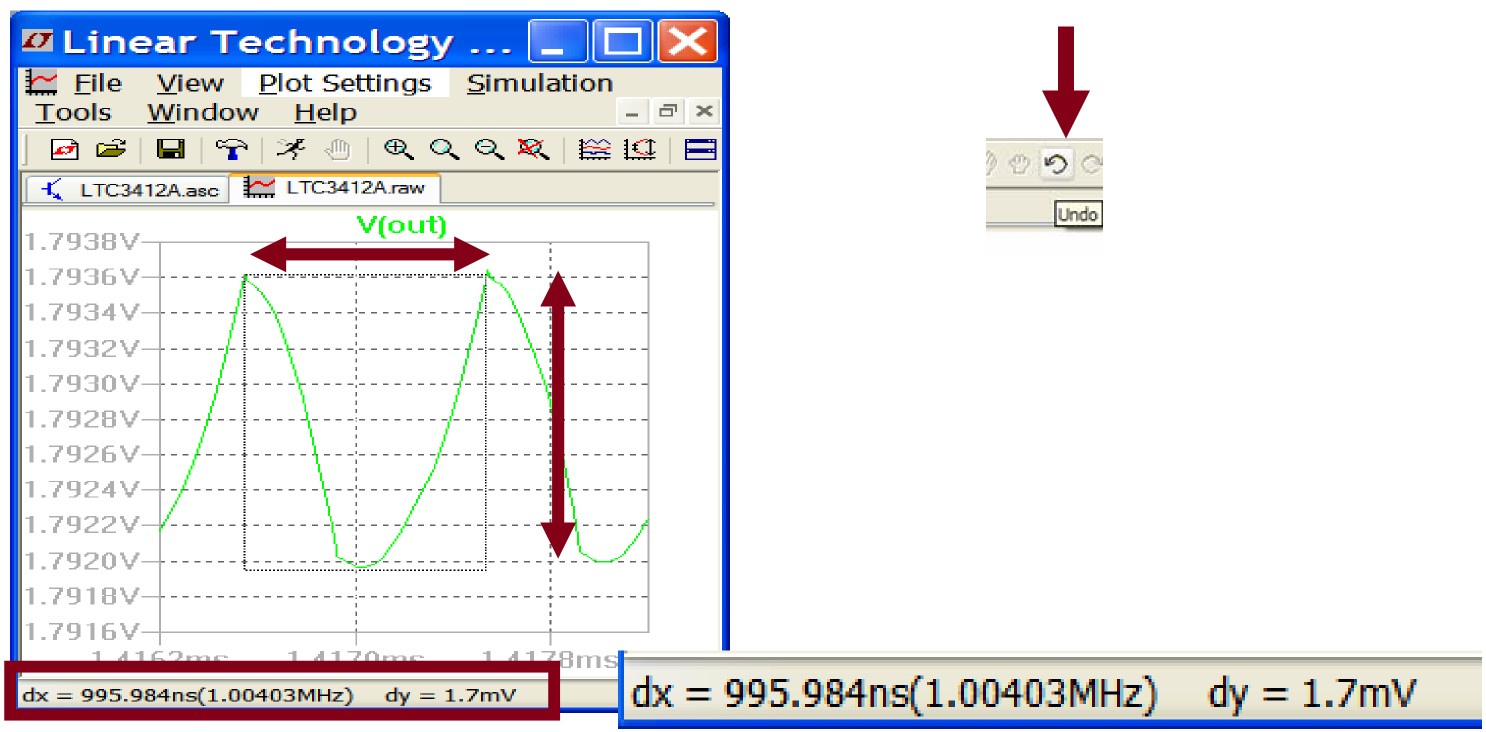
Zooming In and Out on a Waveform



- Using the mouse, click on inductor L1 to display the inductor current waveform. When the mouse cursor is on L1, an ammeter symbol appears.
- 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 for the Waveform (Measurement Using Zoom)

1. Drag a box about the region you wish to measure
   - Left-click, drag, and hold (do not let go of the left mouse key)
2. View the lower left corner of the window for the status bar. The dx and dy measurement data is displayed here.
3. Use the undo button on the toolbar to return to original waveform

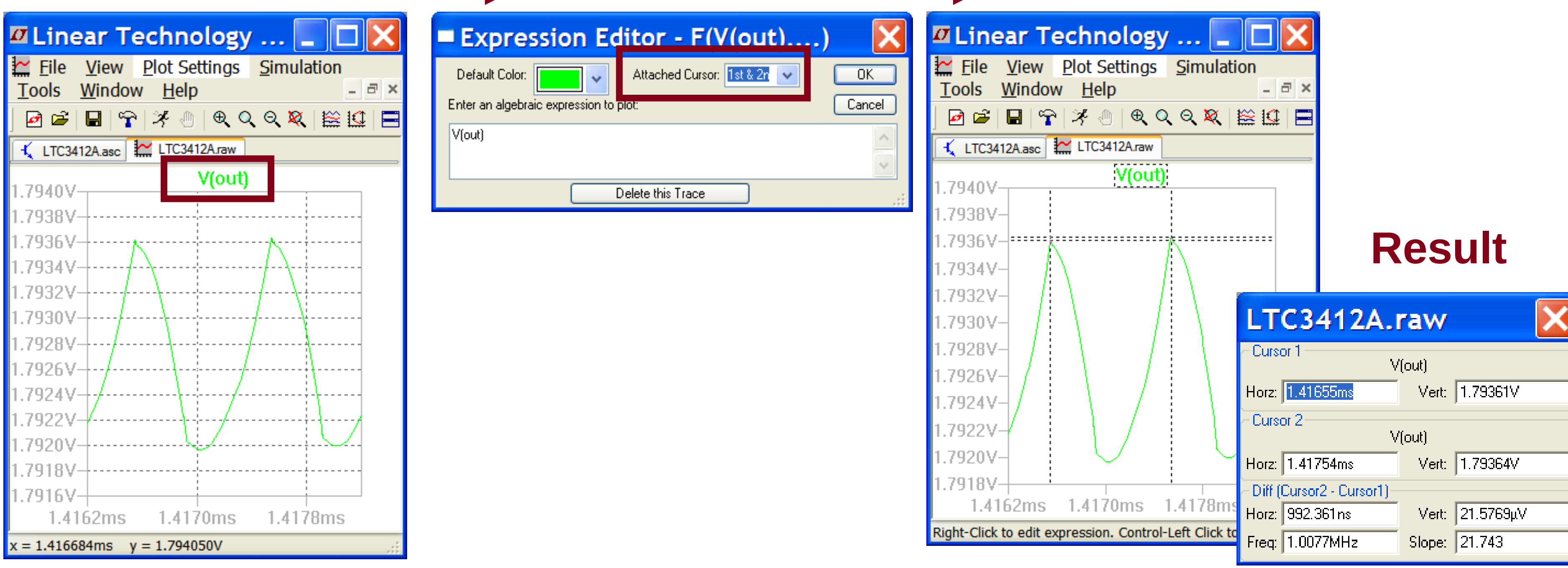


Measuring V, I and Time for the Waveform (Measurement Using Cursors)

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.

1. → 2. → 3.

Result

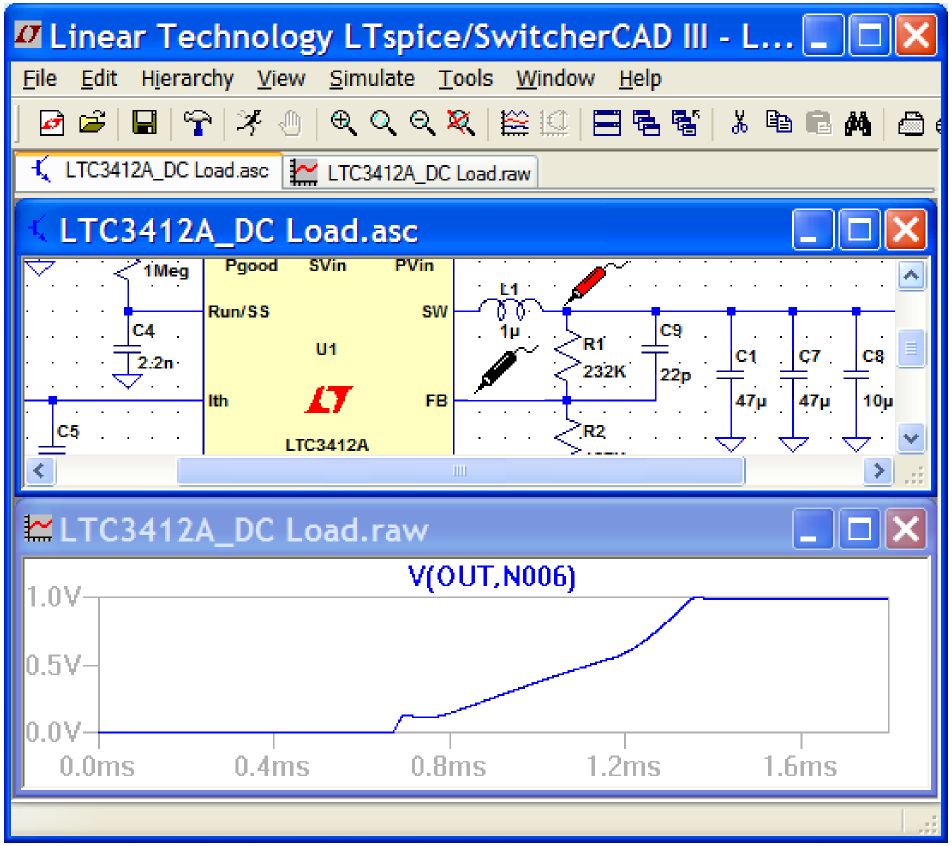


Voltage Differences Across Nodes

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

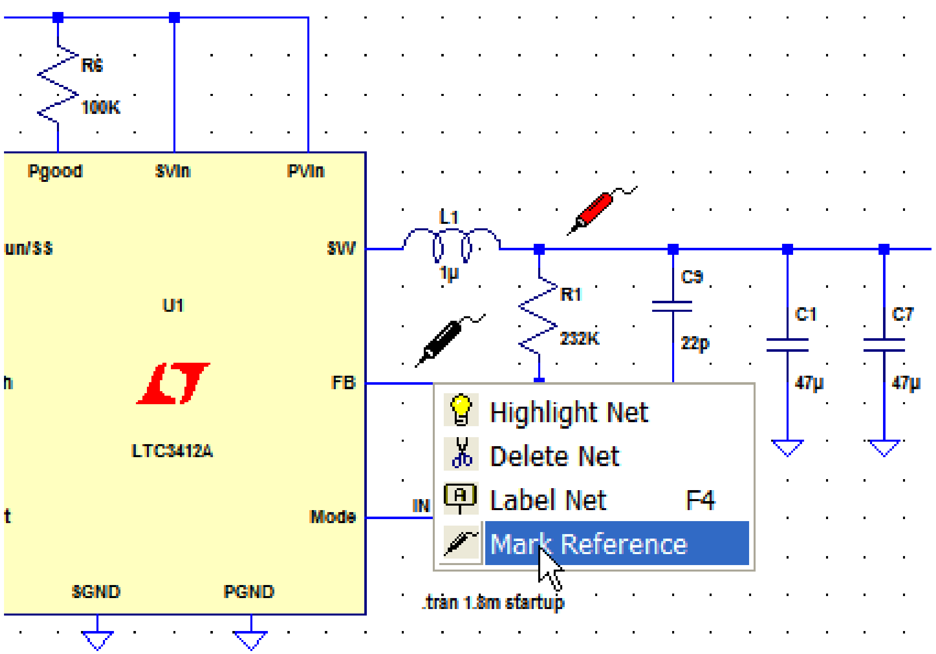
Example:

Measure across LTC3412A top resistor in feedback divider



Voltage Differences Across Nodes

- To create a measurement reference node, right-click on the desired node and select “Mark Reference”
  - A black voltage probe is anchored to the selected node
- All measurements in the circuit are now referenced to the node with the black probe

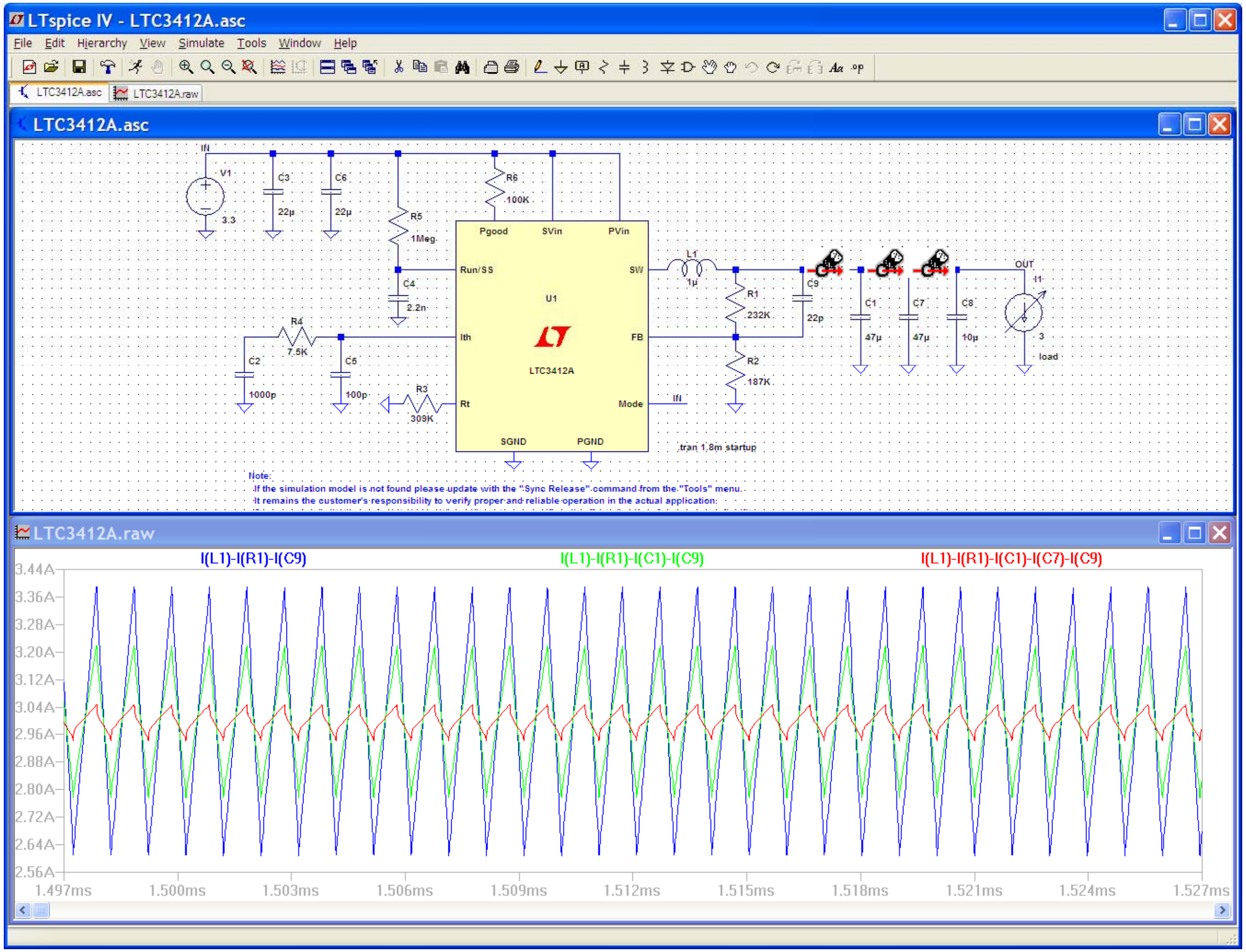


Measuring the Current in a Wire

- To measure the current in a wire, hold down the ALT key and left click on the wire of interest
  - An ammeter will appear to indicate that the wire current will be displayed

Example:

Probe the LTC3412A circuit in the three locations shown to display the different current waveforms.

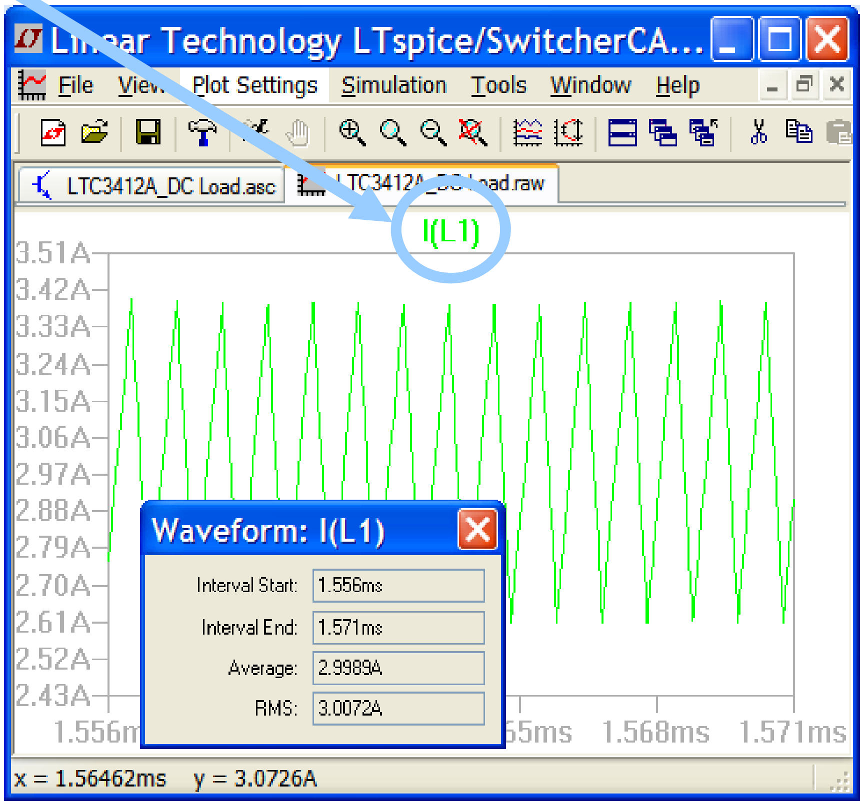


Average & RMS Calculations

- Average & RMS Current, Voltage, or Power Dissipation for selected time window
- Click on inductor L1 to display the inductor current waveform
- Zoom in on the waveform at steady state to display 10-20 cycles
- CTRL key + Left Click the I(L1) trace label in the waveform window
  - Waveform summary window will appear which shows the average and RMS inductor current

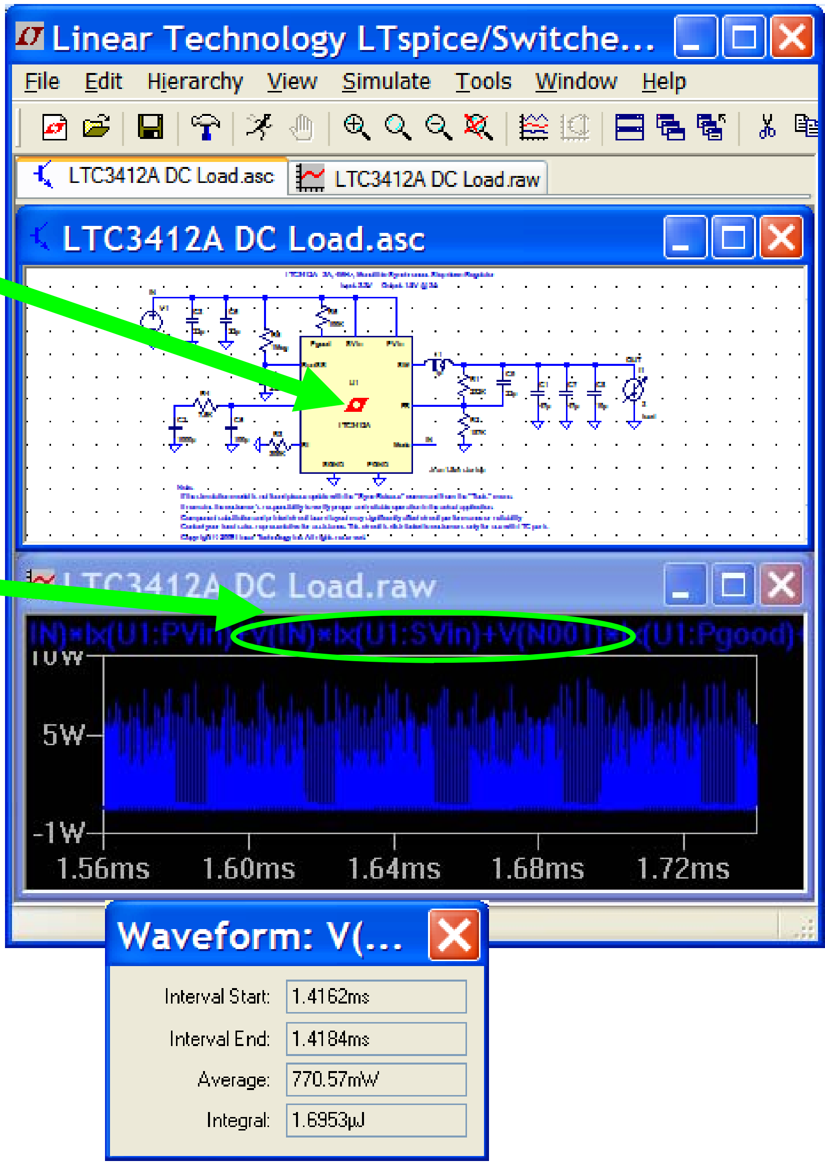
Example:

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 the waveform at steady state
  - **CTRL+ left click** on the Power Dissipation Trace Label in the waveform view
  - Waveform summary window will appear which shows the power dissipation in the IC and the integral of power (Joules)



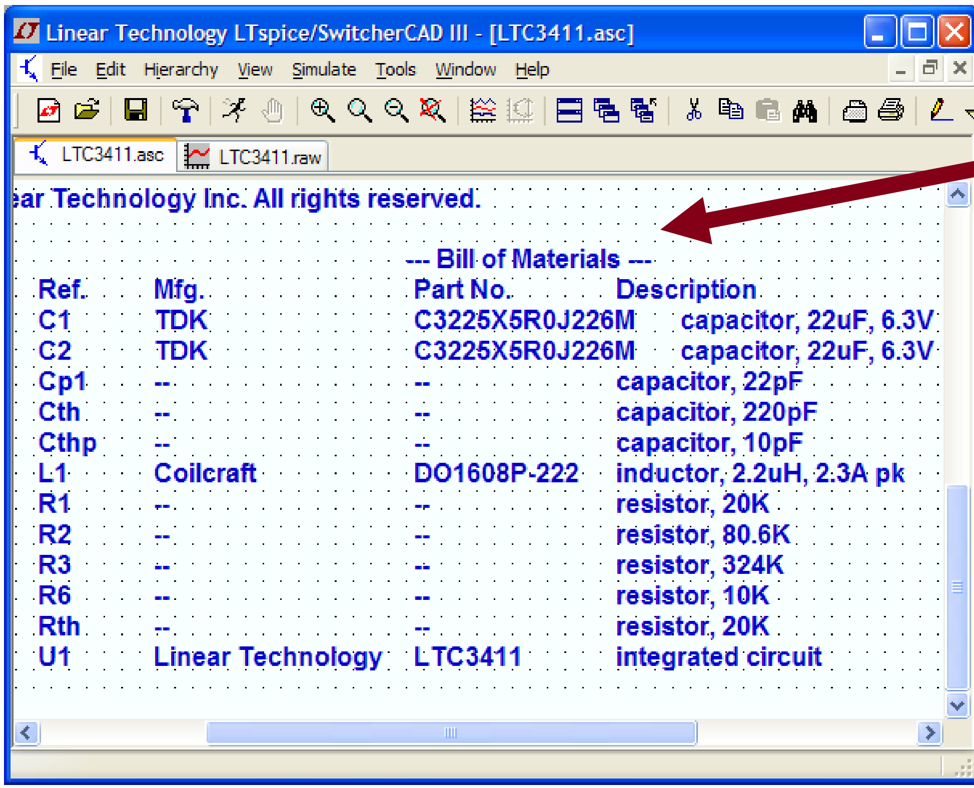
Example:

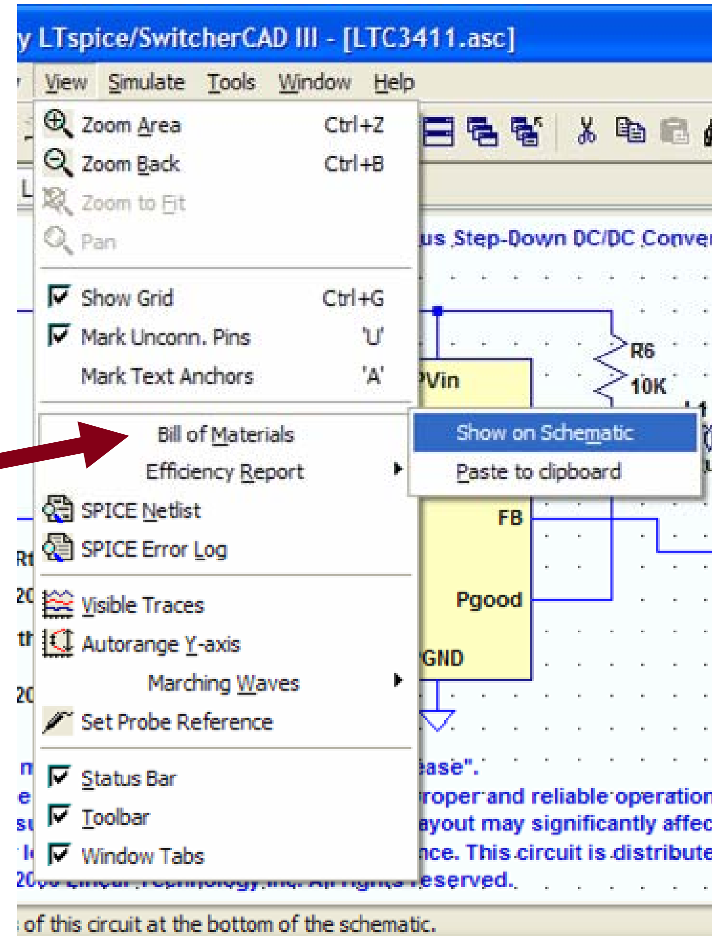
Measure the power dissipation in the LTC3412A IC

Generating a BOM and Efficiency Report

BOM

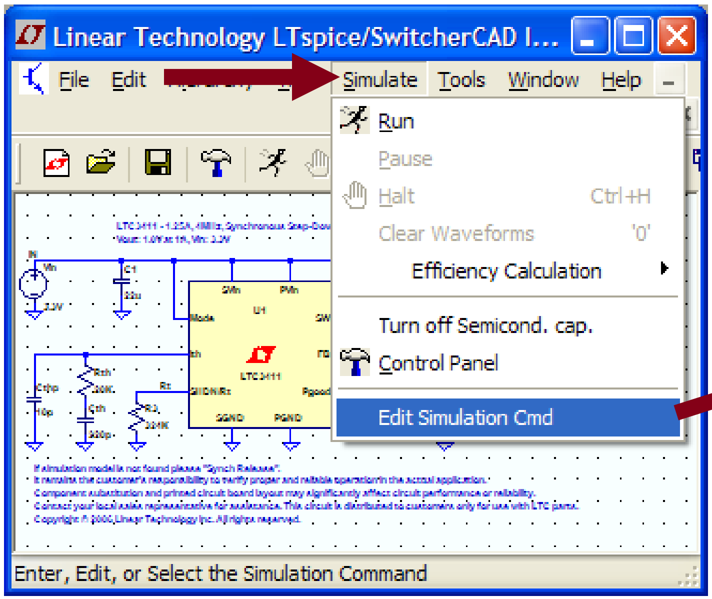
- Under View, select Bill of Material:
  - “Show on Schematic”
  - “Paste to Clipboard” (can paste into Excel and columns will be delimited)

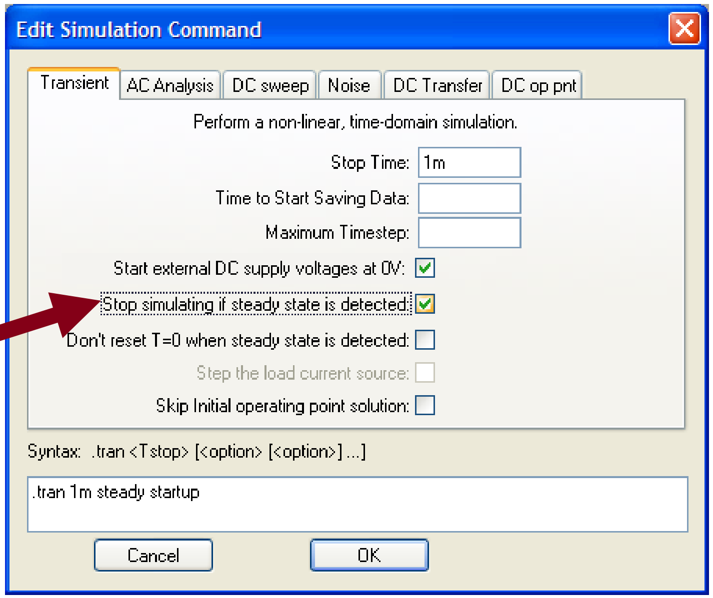




Computing Efficiency & Dissipation

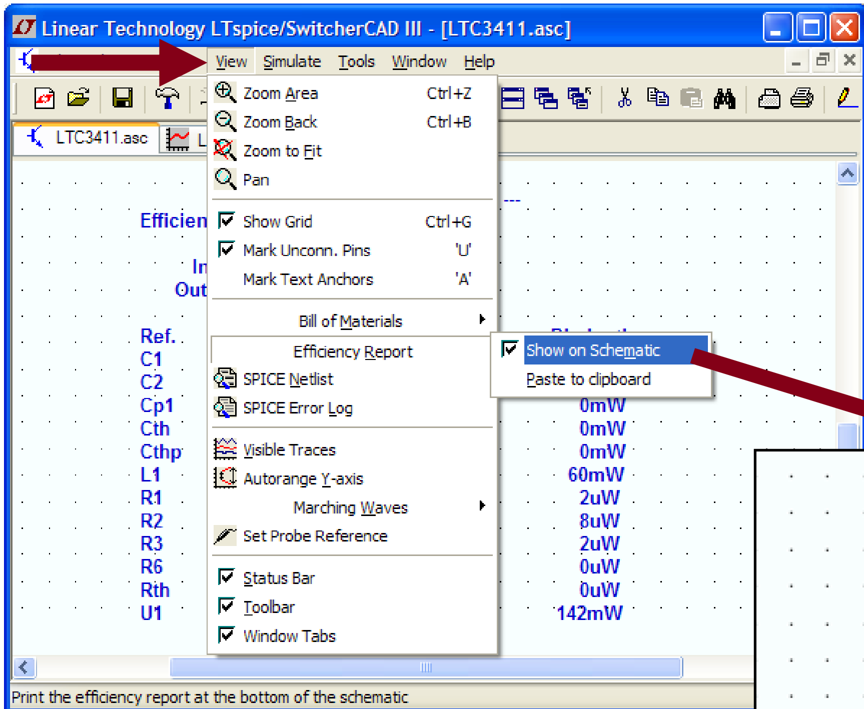
- To compute efficiency of SMPS circuits:
  - Check the "Stop simulating if steady state is detected" on the Edit Simulation Command editor
  - Rerun simulation
  - Use the menu command View=>Efficiency Report



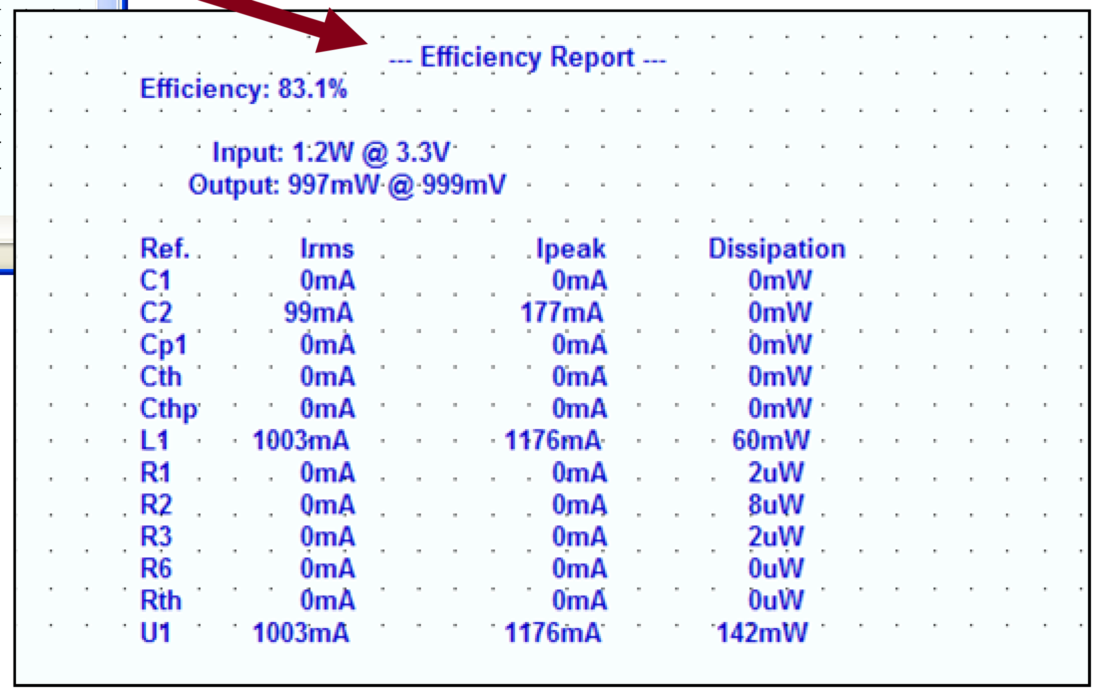


Automatic detection of steady state may not always work – criteria for steady state detection may be too strict or too lenient

Viewing Efficiency Report



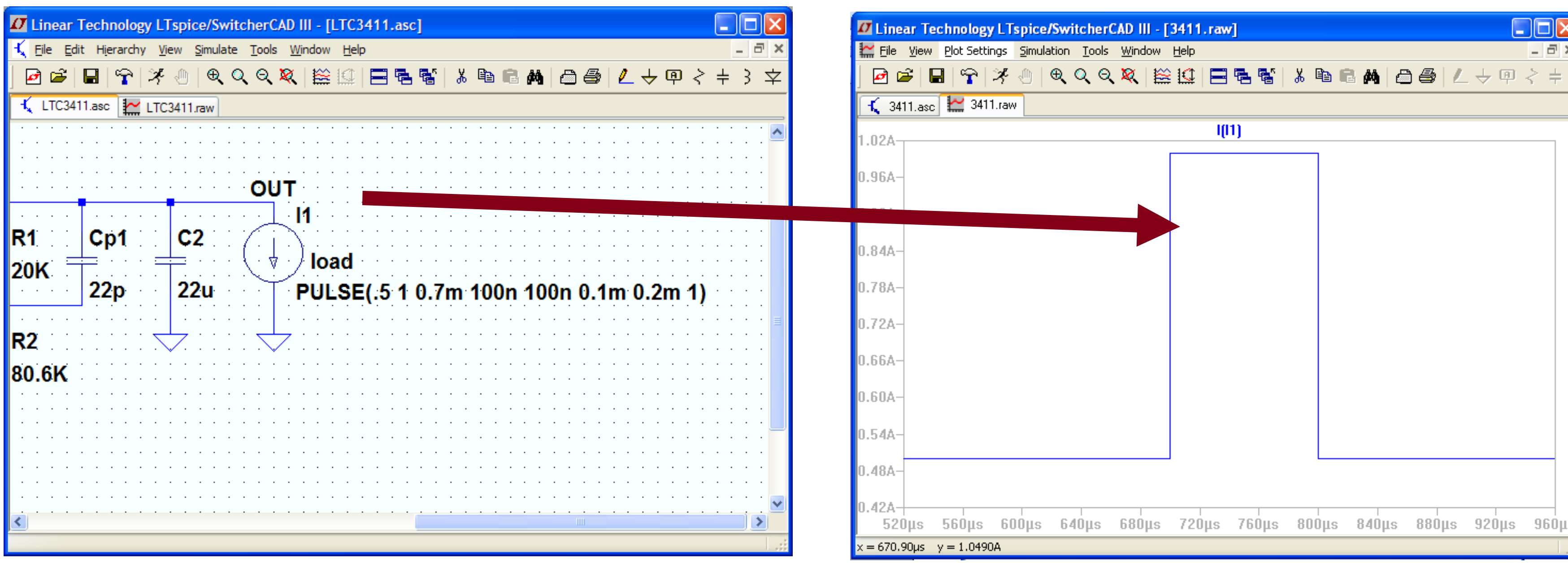
- Under View, select Efficiency Report
  - Show on Schematic
  - Paste to Clipboard (can paste into Excel and columns will be delimited)



Simulating a Transient Response

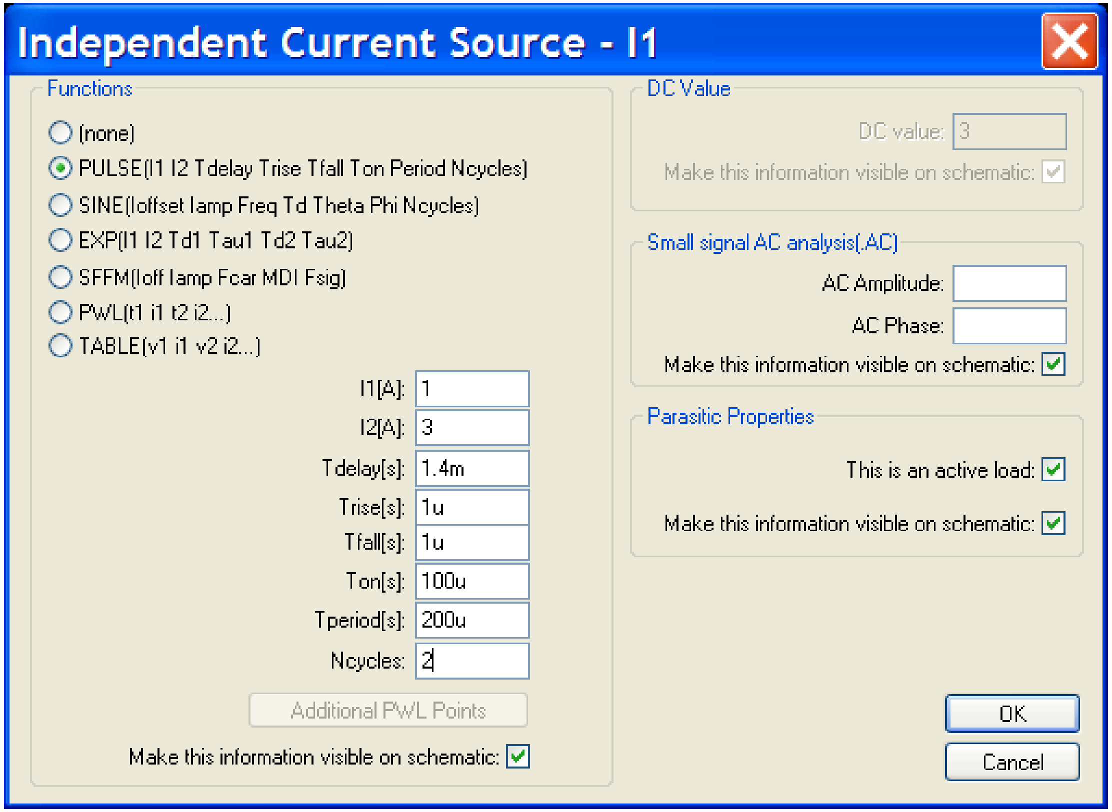
Current Load and Pulse Function

- You can simulate a load with a resistor or current load
- In particular, the pulse function for a current load is helpful for transient response and control loop stability analyses
  - Steps a current load from one value to another value



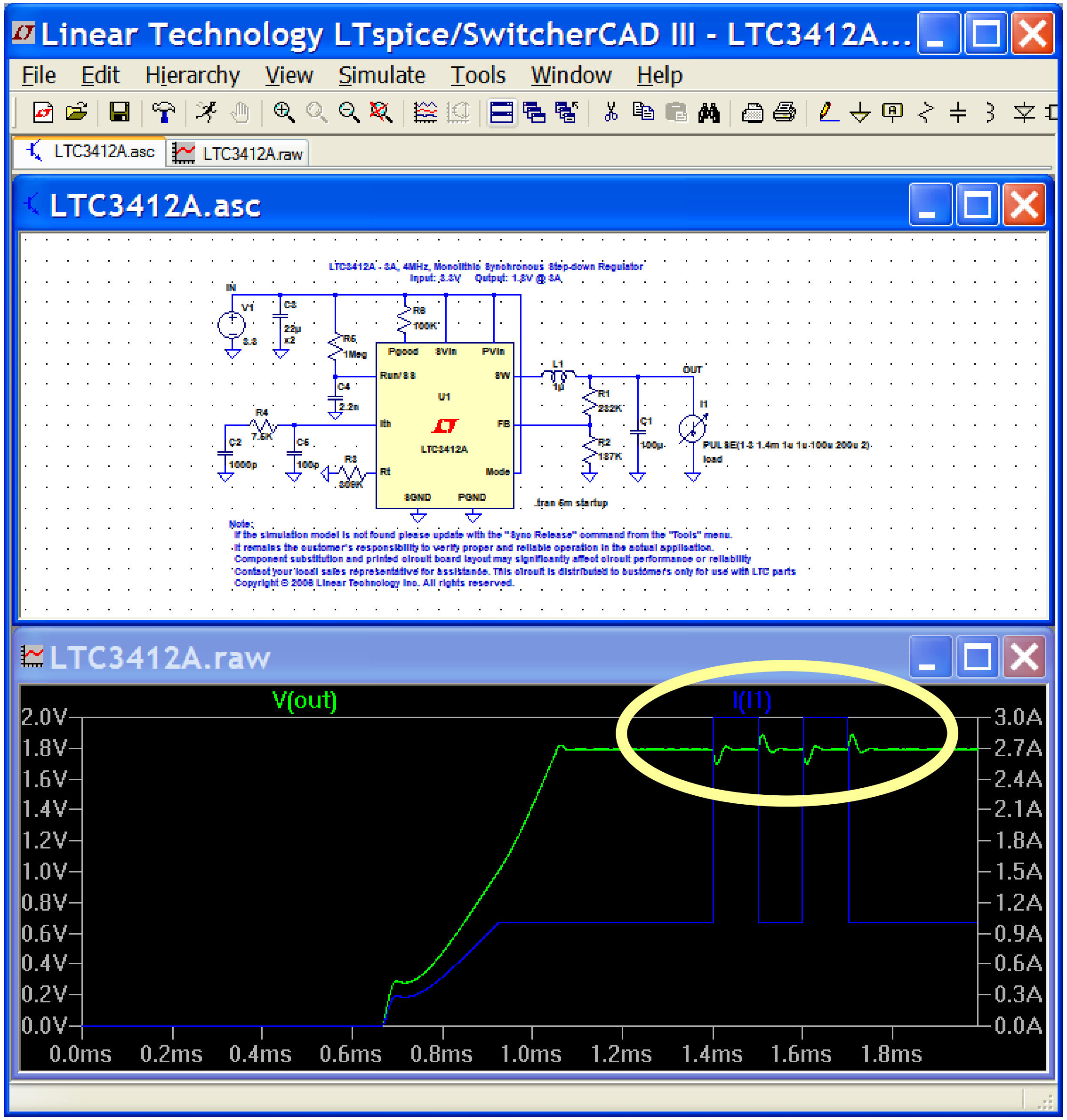
Change the DC Current Load to a Pulse Load

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



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



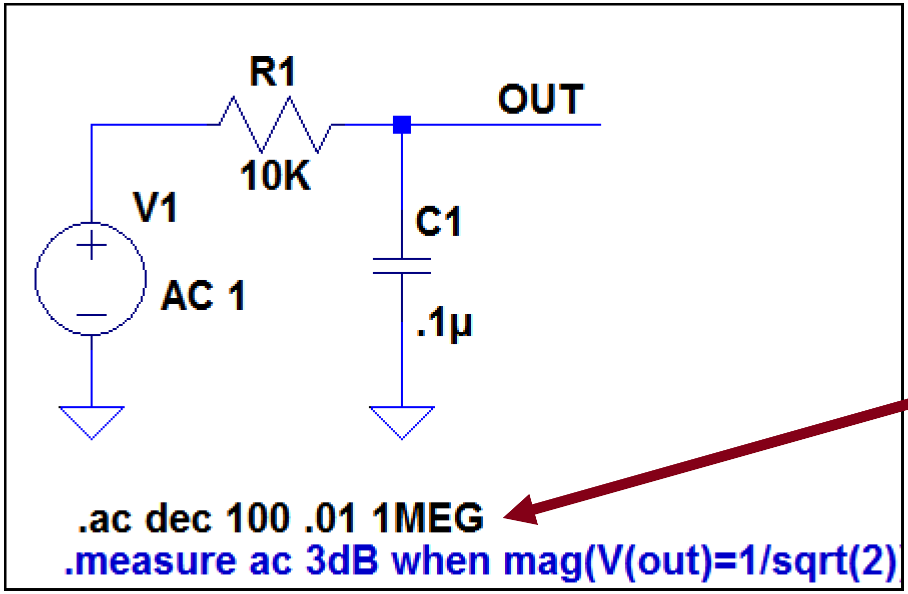
AC Analysis

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

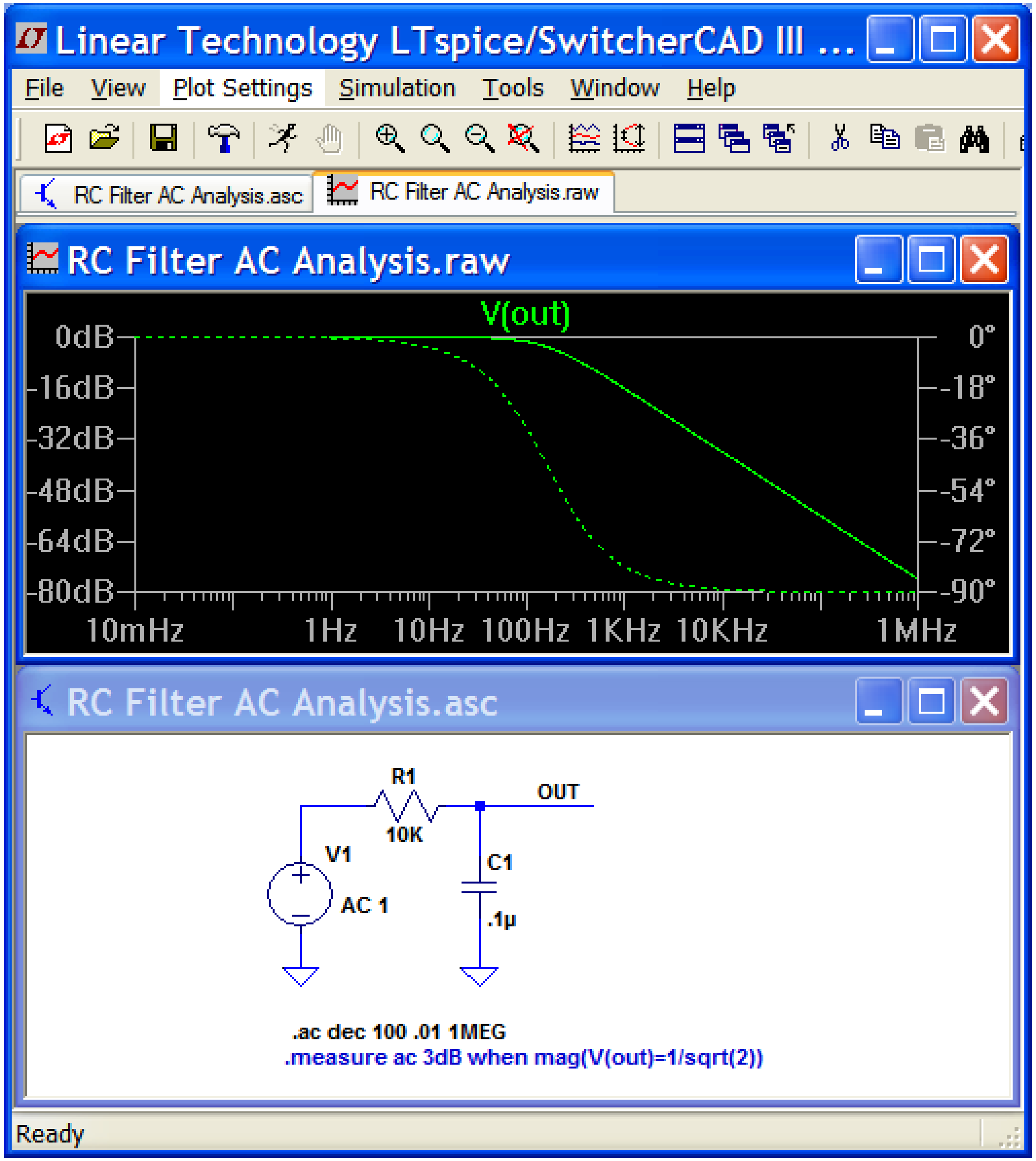
- Single pole filter using RC network
- Syntax: .ac <oct, dec, lin> <Nsteps> <StartFreq> <EndFreq>
- Example: RC network and .ac dec 100 .01 1MEG



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

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

Simulating AC Analysis – RC Result



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 (RightClick on the device name, and modify accordingly)

Note: The contents of the spice model file can be pasted into the schematic as a spice directive. When this is done, the .include spice directive is not needed and the spice model file is no longer needed.

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! The file name syntax is 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:

1N5244B.mod - Notepad

File Edit Format View Help

```
* 1N5244B Zener Diode
* ---------------------------
.MODEL 1N5244B1 D
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
+ NR = 3.907
+ BV = 14.00
+ IBV = 0.001
```

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:

Joe.txt - Notepad

File Edit Format View Help

```
* 1N5244B Zener Diode
* ---------------------------
.MODEL Everest D
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
+ NR = 3.907
+ BV = 14.00
+ IBV = 0.001
```

File name = Joe.txt

Model name = Everest

Importing Third-Party Spice Models

Hands-on Exercise:

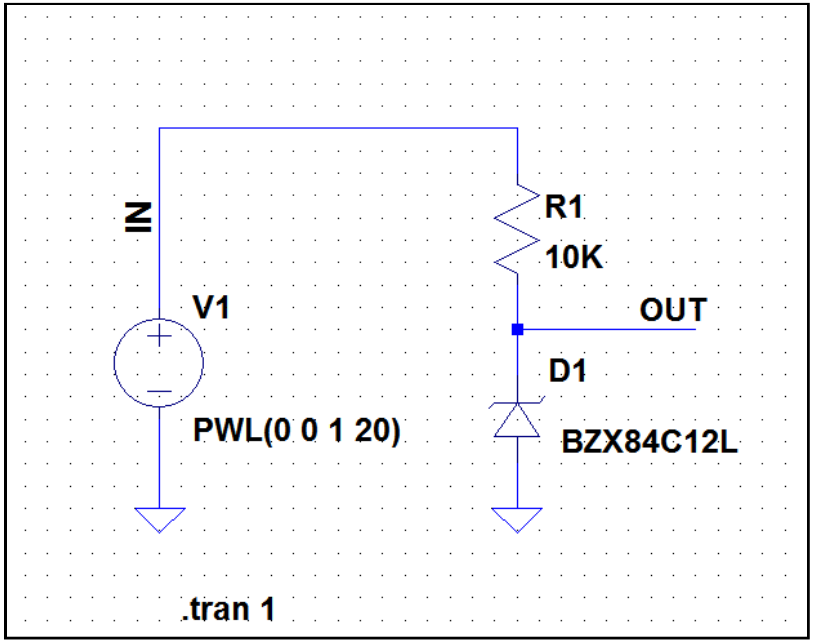
1.) Navigate to the LTSpice Training Files folder

2.) Open up the simulation file titled “Zener Import Example.asc”

3.) Open up the SPICE model file titled “1N5244B.mod” 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



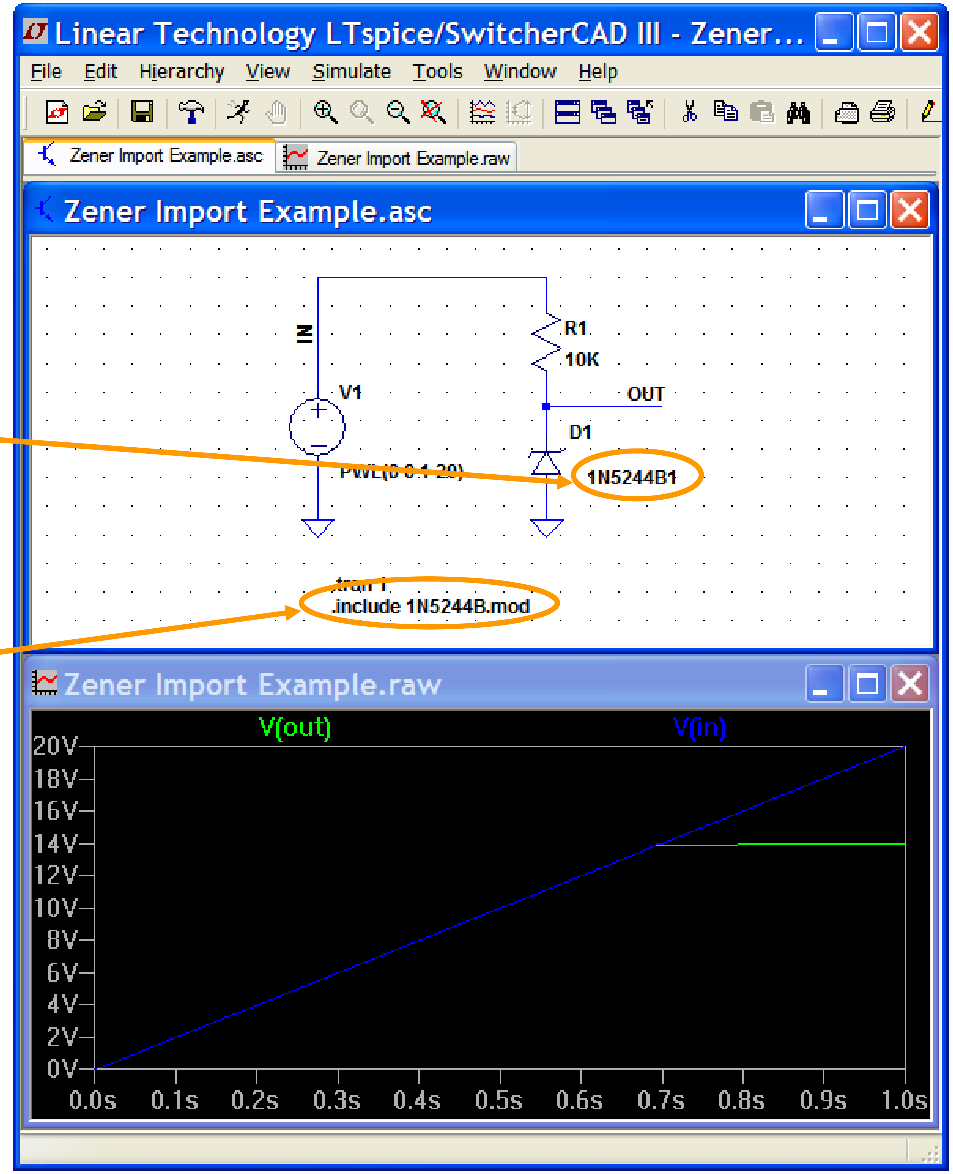
Importing Third-Party Spice Models

Solution:

1.) Zener name changed to 1N5244B1 to match model name in the SPICE model file. RightClick 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.

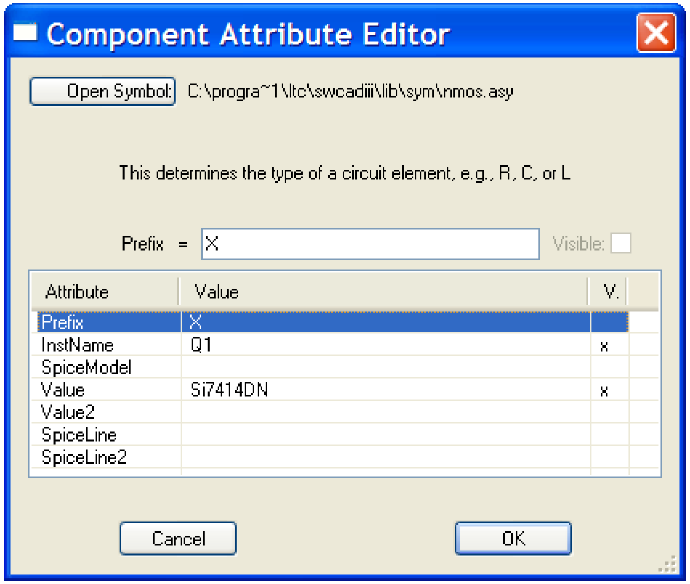
3.) Result after clicking on the Running Person symbol on the toolbar and probing the IN and OUT nodes.



Importing Third-Party Spice Models

Types of SPICE Models (open up the SPICE model file to determine)

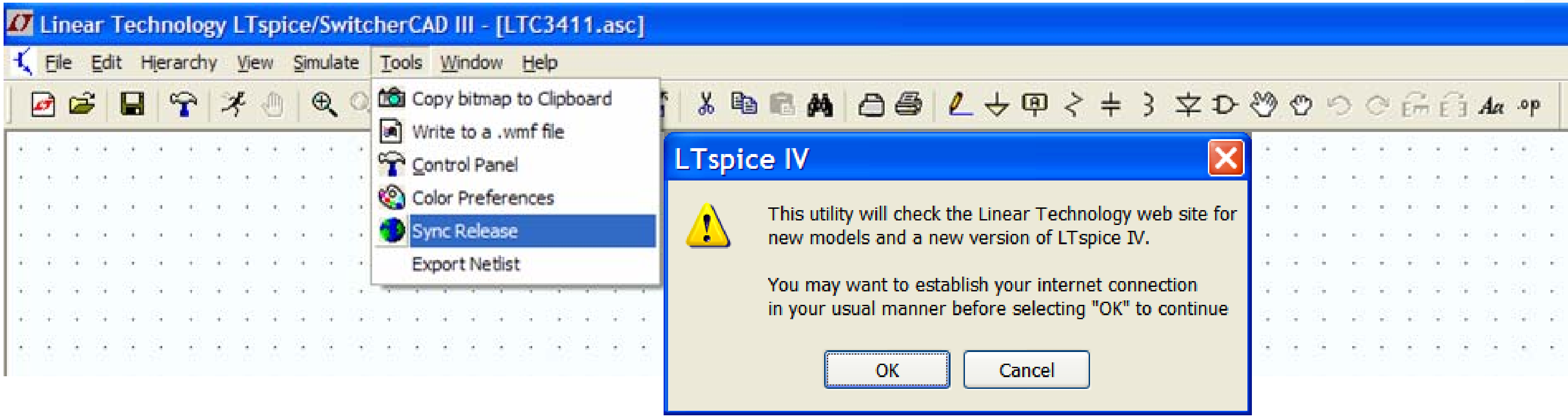
- .MODEL definition (as covered in the previous Zener example)
  1. Change the device name in the simulation schematic to match the device name in the SPICE model file
  2. Add the SPICE directive to the schematic ".include spice_model_file.abc"
- .SUBCKT definition
  1. Same as above
  2. Same as above
  3. Must CTRL+RightClick on the device and change the Prefix to "X".



For additional information, search LTspice Help using the key words "third party model".

More Information and Support

Reminder to Periodically Synch Release



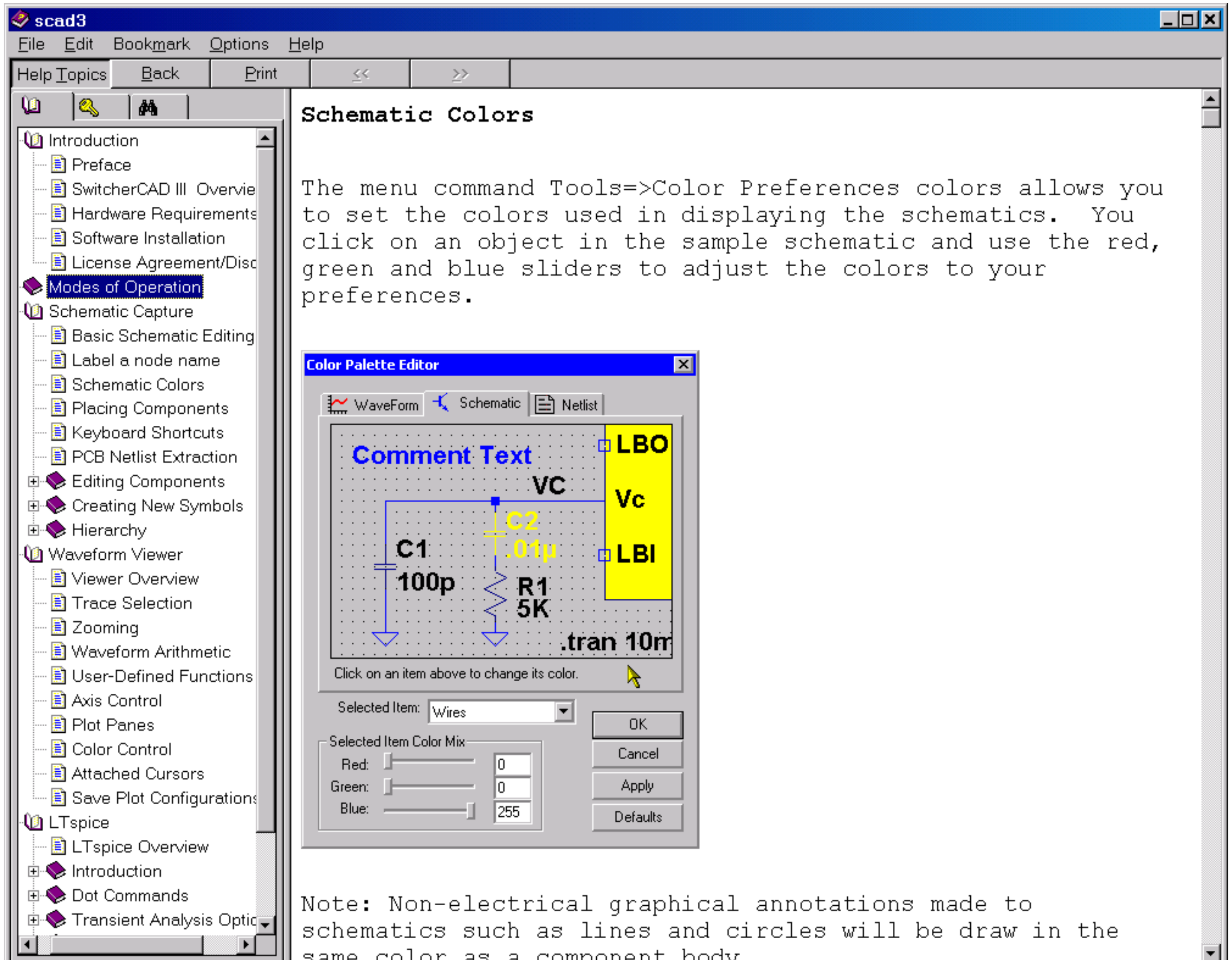
- It is important to synch your release of LTspice once a month to get the latest updates:
  - Software updates and bug fixes
  - New models
  - Updated libraries (your library updates will be preserved)
  - Sample circuits and examples

<u>Synch Release</u> changes are captured in the changelog.txt file located under the following path:

C:\Program Files\LTC\SwCADIII
--or--
C:\Program Files\LTC\LTspiceIV

Built-in Help System

- Select F1 for help menu



PDF User's Guide

- Download the PDF Users Guide Manual:
  - http://LTspice.linear.com/software/scad3.pdf
  - This is a PDF of the help file database with the addition of a table of contents and an index.

Thank you for attending, and happy simulating!

<u>**Challenge:**</u> **Once you return to the office, go back over the training materials within one week!**

Appendix A – Summary of Special Mouse and Keyboard Commands

Schematic-Based Special Commands:

1. ALT + LeftClick on a wire
   - This will display the waveform for the current flowing in the wire
2. ALT + LeftClick on a component
   - This will display the instantaneous power dissipation in the component
3. CTRL + RightClick on a component
   - Allows you to edit embedded component attributes

Waveform-Based Special Commands:

1. CTRL + LeftClick on a waveform title
   - Displays the average and RMS values for the waveform
2. LeftClick on a waveform title
   - Enables a single measurement cursor

Appendix B – Summary of Additional Features

1. To pause a simulation:
   - “Simulate” pull down menu ---> Pause
2. To zoom in/out using the schematic editor:
   - Just use the wheel on your mouse
3. To pan around a schematic
   - Just left-click the mouse and hold, then drag