LTspice IV Basic Lab Class

Presented by Thomas Mosteller ADI FAE

LINEAR TECHNOLOGY | NOW PART OF ANALOG DEVICES

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 2500 macromodels of Linear Technology products
- 1400+ power products

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 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 for Windows** (Updated November 12, 2013) • **Download LTspice IV for Mac OS X 10.7+** • LTspice Information Flyer & Shortcuts • Mac OS X Shortcuts • LTspice Getting Started Guide • LTspice Blog • LTspice Demo Circuit Collection • View Upcoming LTspice Seminars Follow LTspice on Twitter! View the LTspice Video Channel

How Do I Get Started using LTspice?

How Do I Get Started Using LTspice?

- *Demo Circuits*: 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 or browse through the part's webpage (right column)
- *JIG Files*: 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
- *Blank*: 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 or …

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 for Windows (Updated November 12, 2013) • Download LTspice IV for Mac OS X 10.7+ • LTspice Information Flyer & Shortcuts • Mac OS X Shortcuts • LTspice Getting Started Guide • LTspice Blog • LTspice Demo Circuit Collection • View Upcoming LTspice Seminars Follow LTspice on Twitter! View the LTspice Video Channel

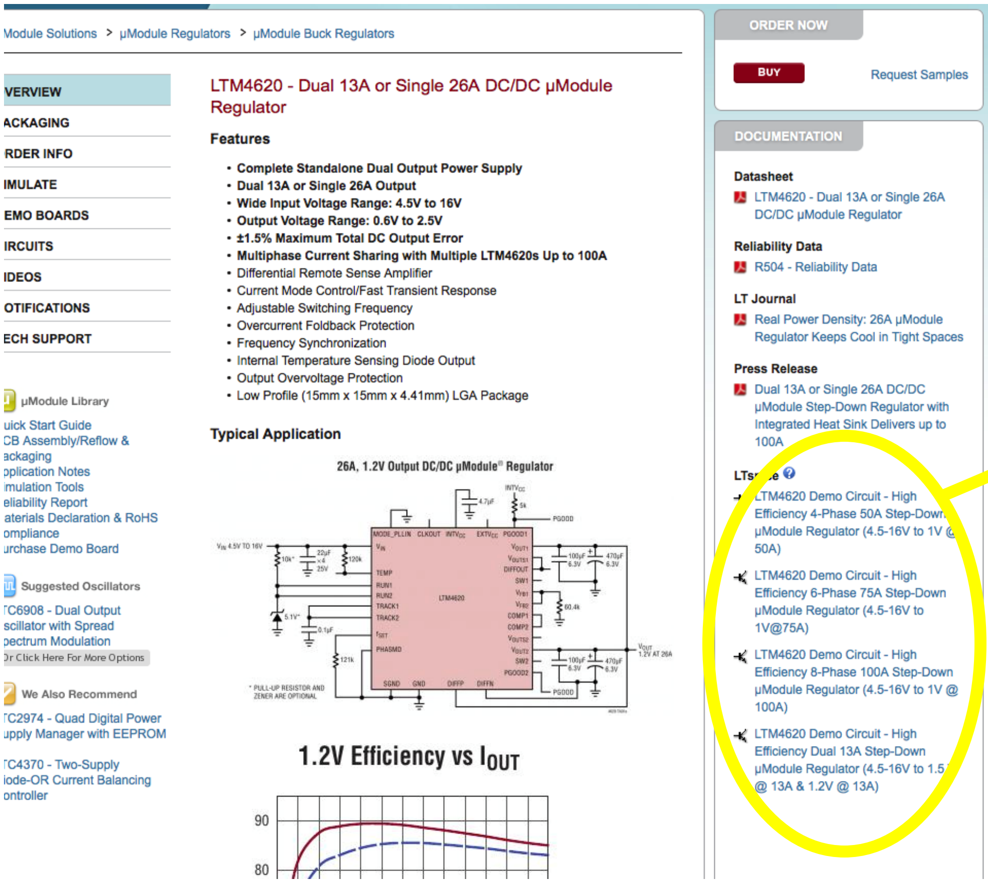
LTC3788-1 Demo Circuit - High Efficiency Dual 12V/24V Boost Converter with Rsense Current Sensing (4.5-24V to 24V @ 5A & 12V @ 10A)	2013-11-18	LTC3788-1
LTC6090 Demo Circuit - Wide Dynamic Input-Output Range High Voltage Integrator	2013-11-13	LTC6090
LT6105 Demo Circuit - Current Sense Amplifier Monitors Both +15V and -15V Supplies	2013-11-11	LT6105
LT8302 Demo Circuit - µPower No-Opto Isolated Flyback Converter (10-30V to 5V @ 2.2A)	2013-11-07	LT8302
LT3090 Demo Circuit - Negative Linear Regulator with Current Monitor (-5V to -1.25V @ 600mA)	2013-11-05	LT3090
LTC1625 Demo Circuit - High Efficiency Step-Down Converter (5-28V to 3.3V @ 4.5A)	2013-10-28	LTC1625
LTC2053 Demo Circuit - Unidirectional Current Sense Circuit for 1V Supply (0A to 10A)	2013-10-28	LTC2053

What if I'm browsing the part's webpage?

Demo Circuits on linear.com (cont.)

Go to the part's webpage

http://www.linear.com/product/LTM4620



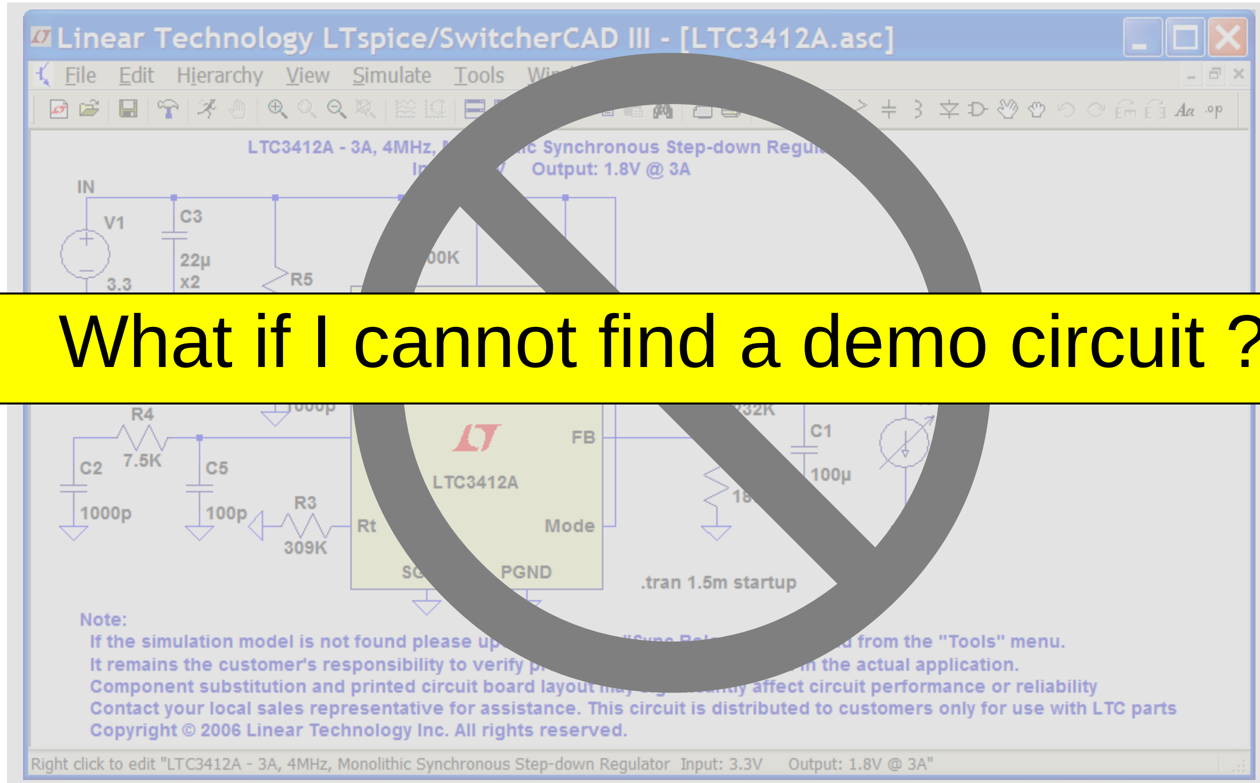
LTspice

- LTM4620 Demo Circuit - High Efficiency 4-Phase 50A Step-Down µModule Regulator (4.5-16V to 1V @ 50A)
- LTM4620 Demo Circuit - High Efficiency 6-Phase 75A Step-Down µModule Regulator (4.5-16V to 1V@75A)
- LTM4620 Demo Circuit - High Efficiency 8-Phase 100A Step-Down µModule Regulator (4.5-16V to 1V @ 100A)
- LTM4620 Demo Circuit - High Efficiency Dual 13A Step-Down µModule Regulator (4.5-16V to 1.5 V @ 13A & 1.2V @ 13A)

NOW PART OF

What are Demo Circuits ?

Circuits designed and reviewed by factory apps group



What if I cannot find a demo circuit ?

- 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

NOW PART OF

How Do I Get Started Using LTspice?

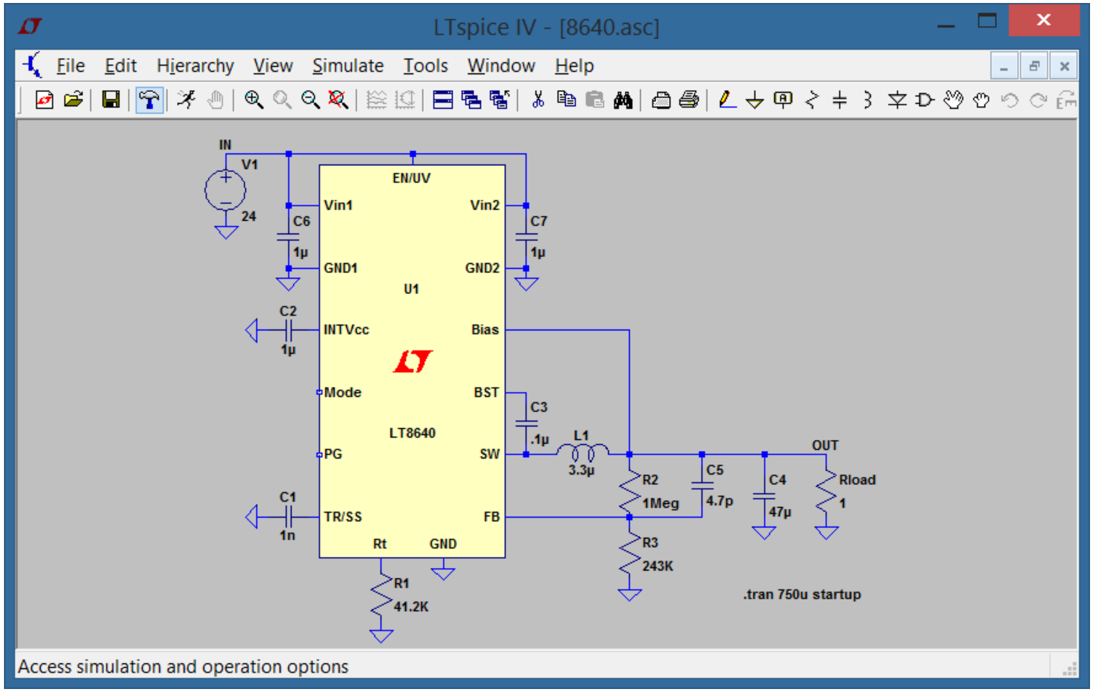
- *Demo Circuits*: Use one of the 100's demo circuit available on linear.com
  - Designed and Reviewed by Factory Apps Group
  - Go to http://www.linear.com/software or browse through the part's webpage (right column)

- *JIG Files*: 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

- *Blank*: Use the schematic editor to create your own design
  - LTspice contains models for most LTC power devices and many more

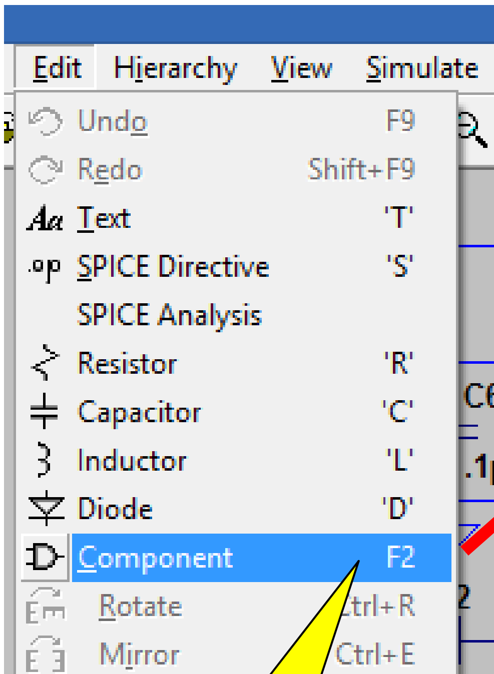
Pre-drafted Test Fixture

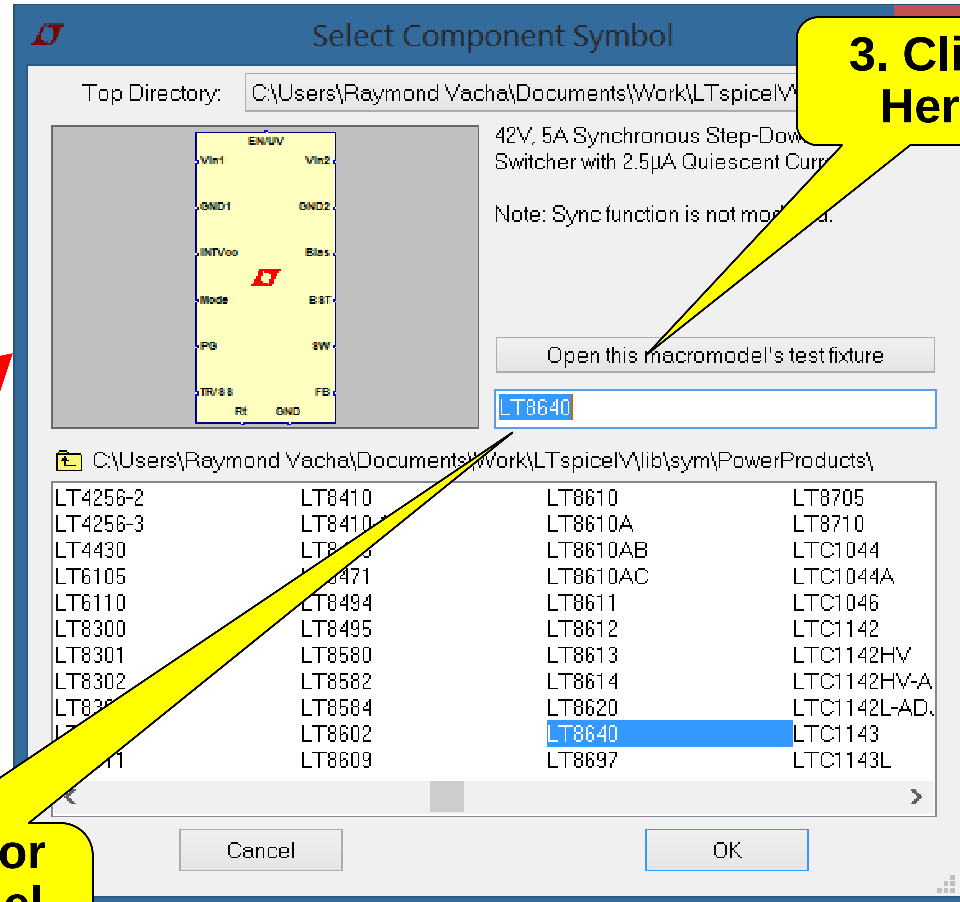
- These simulations / designs are not production-ready
- Used to prove out part models, and are not complete designs.
- Components are typically “ideal” components and will need to be modified based on your operating conditions



- 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

Opening a Test Fixture





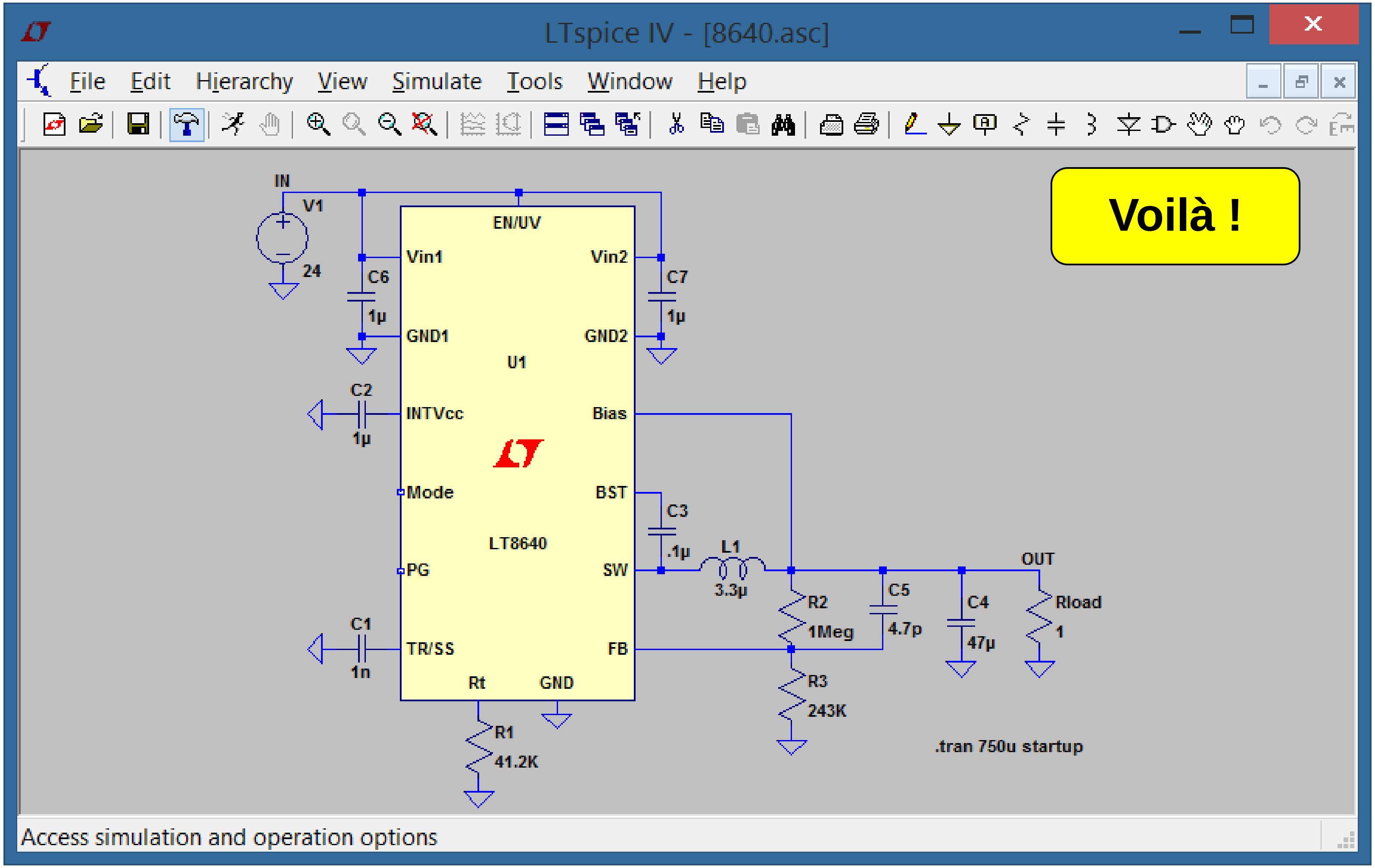
3. Click Here

1. Edit menu, select "Component"

2. Search for macromodel (ex. LT8640)

NOW PART OF ANALOG DEVICES

Opening a Test Fixture



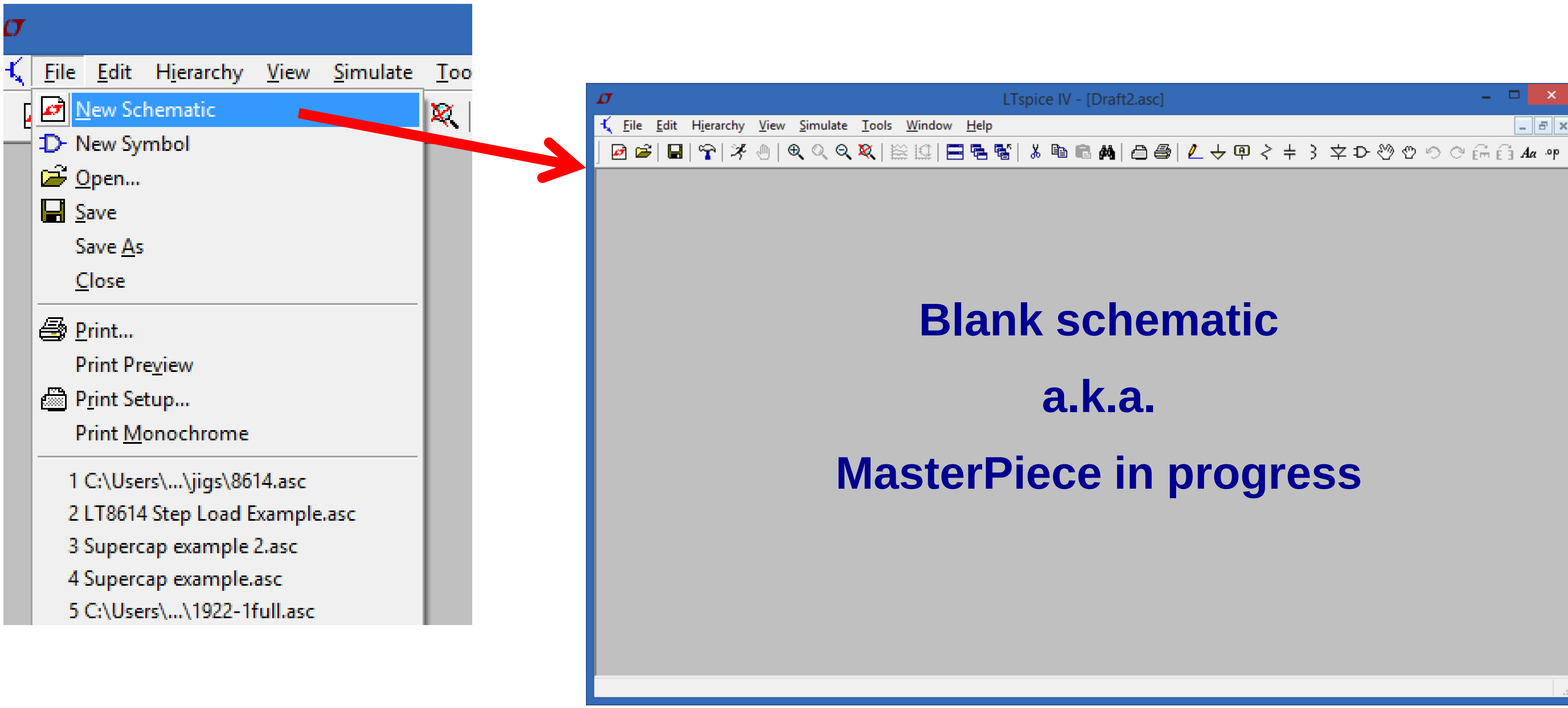
NOW PART OF

How Do I Get Started Using LTspice?

- *Demo Circuits*: 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 or browse through the part's webpage (right column)
- *JIG Files*: 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
- *Blank*: 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

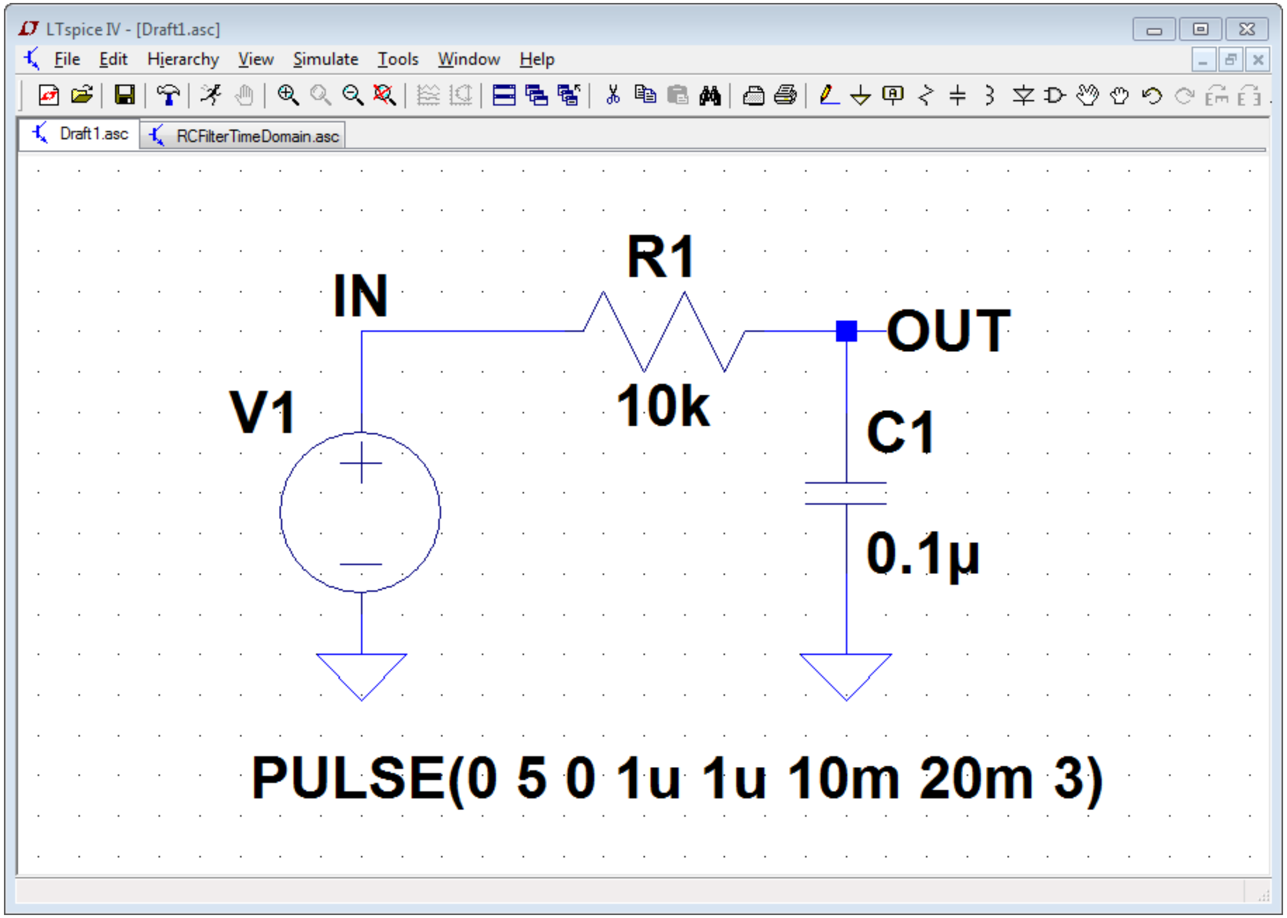
- To open up a blank schematic screen select "*File*" Menu and "*New Schematic*"



Using the Schematic Editor in LTspice

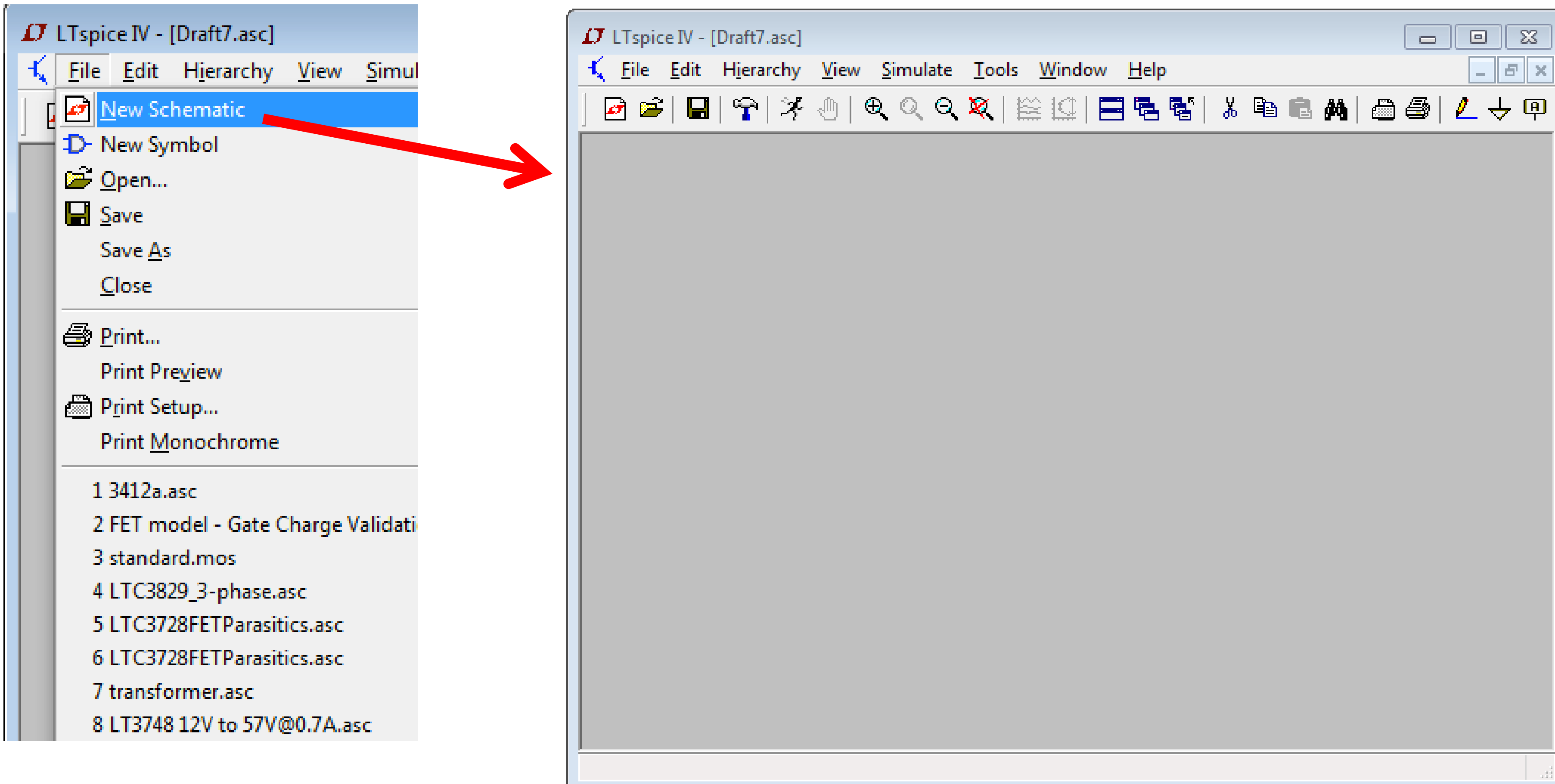
How to Wire up a Simple RC Circuit

- The completed exercise:



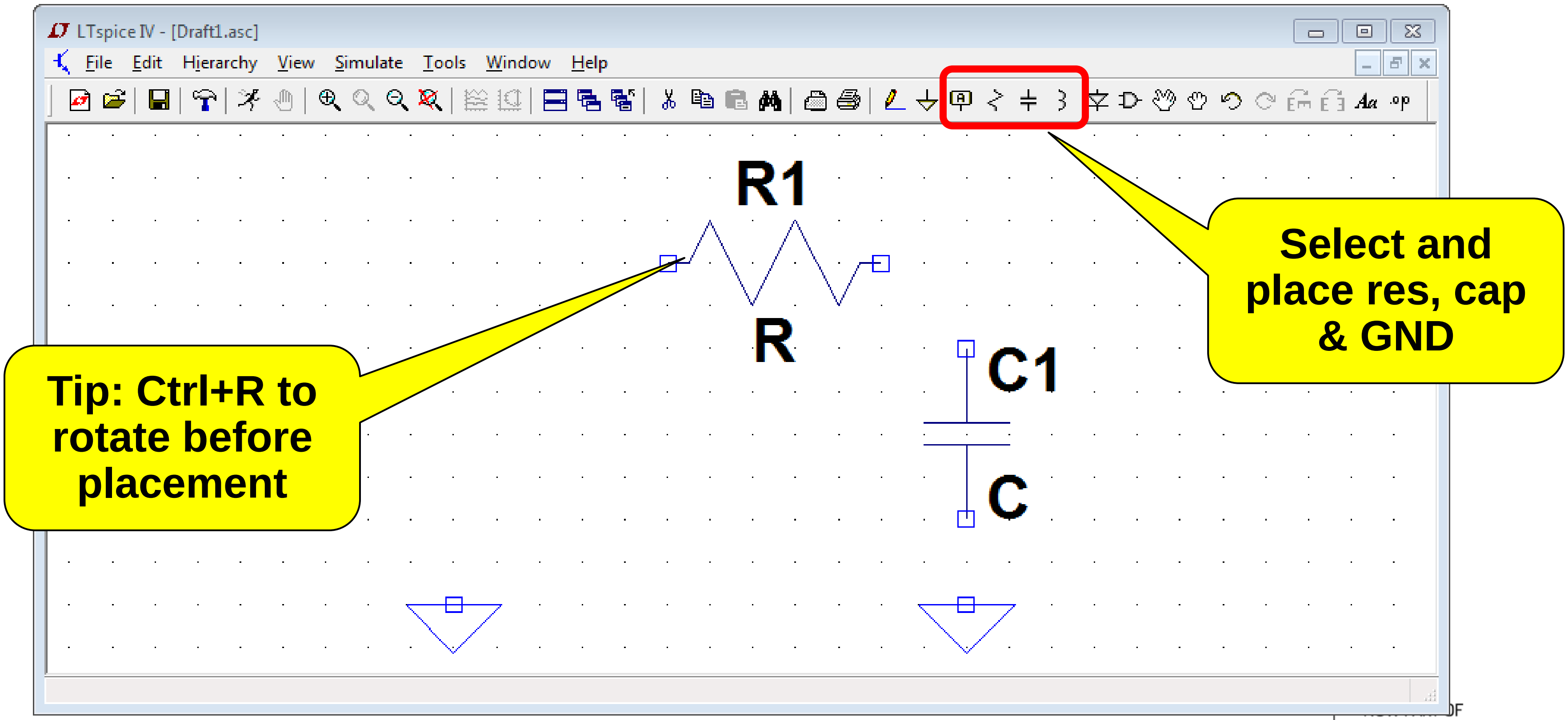
How to Wire up a Simple RC Circuit (cont.)

- Step 1: Open up a blank schematic screen
  - Select “*File*” Menu and “*New Schematic*”



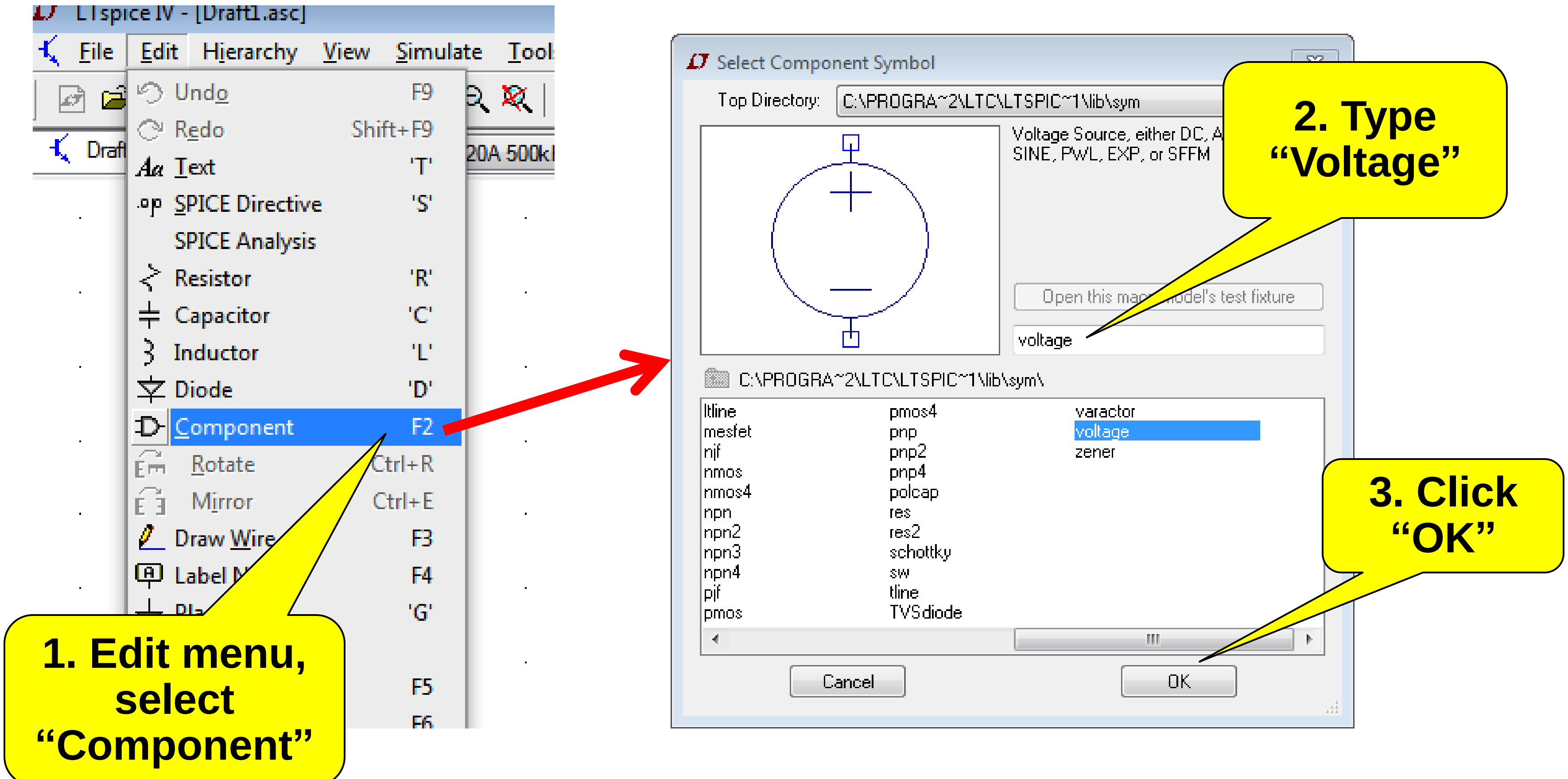
How to Wire up a Simple RC Circuit (cont.)

- Step 2: Add the passives and grounds
  - Using the toolbar, select Resistor, Capacitor and Ground. Place these symbols on the schematic as shown below. Use Ctrl+R to rotate before placement.



How to Wire up a Simple RC Circuit (cont.)

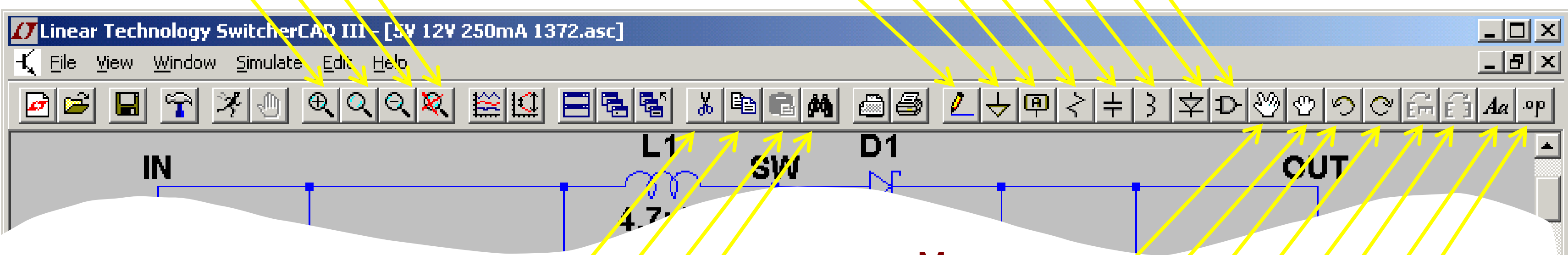
- Step 3: Add the voltage source
  - Select "*Edit*" Menu and "*Component*". *From the component window, start typing "voltage" in the dialog box, and click "OK"*



Toolbar and Keyboard Shortcuts

Zoom In
Pan
Zoom Out
Autoscale

Place Circuit Element [F2]
Place Diode [D]
Place Inductor [L]
Place Capacitor [C]
Place Resistor [R]
Label Node [F4]
Place Ground [G]
Draw Wire [F3]

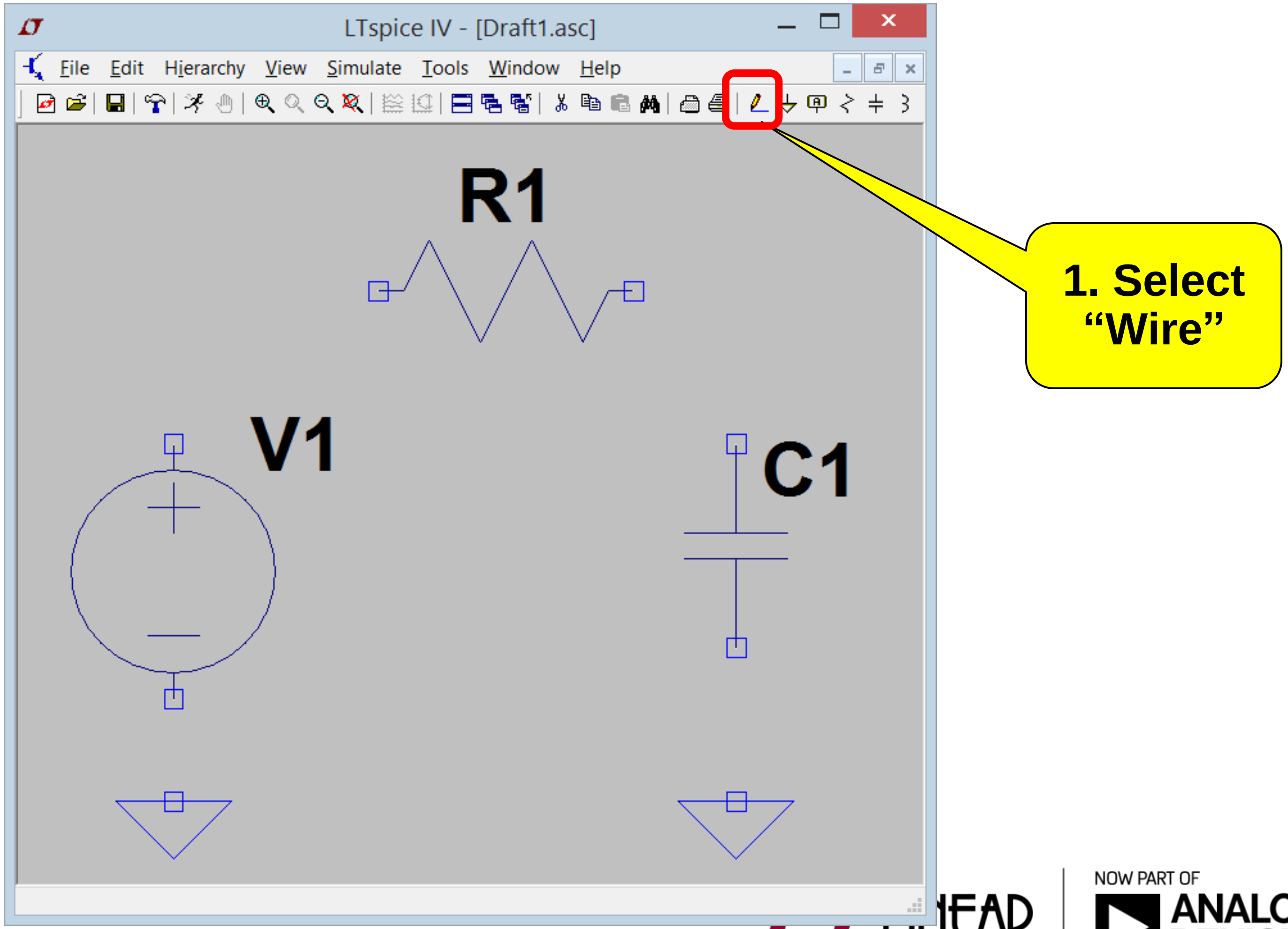


Delete [Del]
Duplicate [Ctrl+C]
Paste b/t Schematics [Ctrl+V]
Find [Ctrl+F]

Move [F7]
Drag [F8]
Undo [F9]
Redo [Shift+F9]
Rotate [Ctrl+R]
Mirror [Ctrl+E]
Place Comment/text [T]
Place SPICE directive [S]

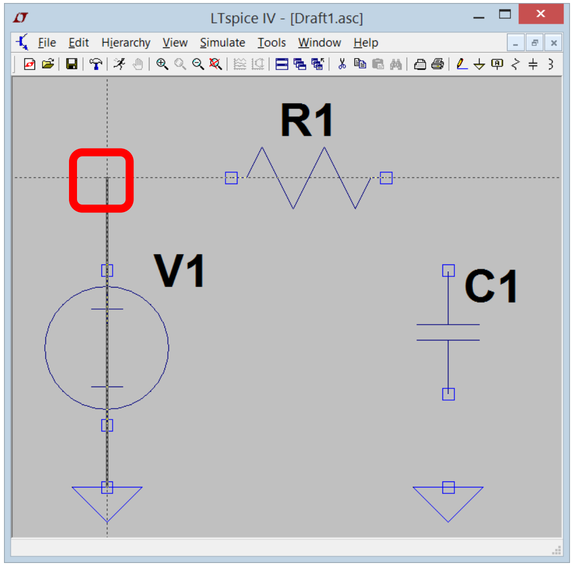
How to Wire up a Simple RC Circuit (cont.)

- Step 4: Wire up the circuit
  - Using the toolbar, select Wire



How to Wire up a Simple RC Circuit (cont.)

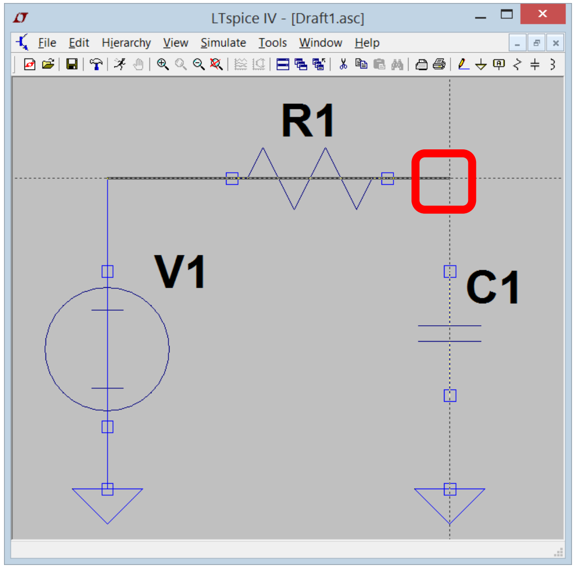
- Step 4: Wire up the circuit (cont.)



Left-Click ground

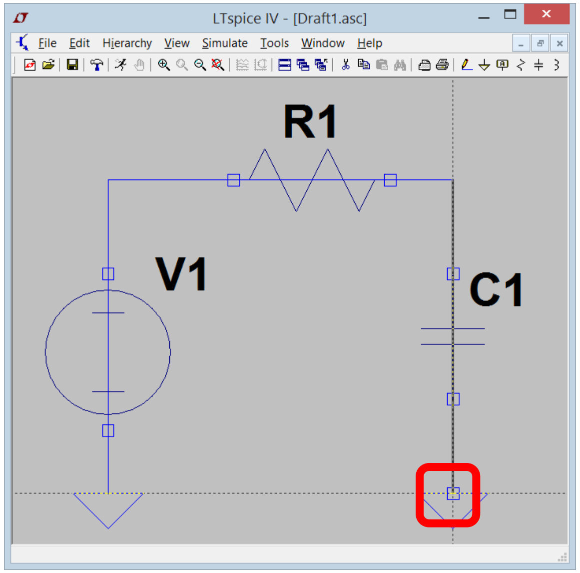
“Pull” wire up through the source

Left-Click here ☐ to anchor



“Pull” wire through the resistor

Left-Click here ☐ to anchor



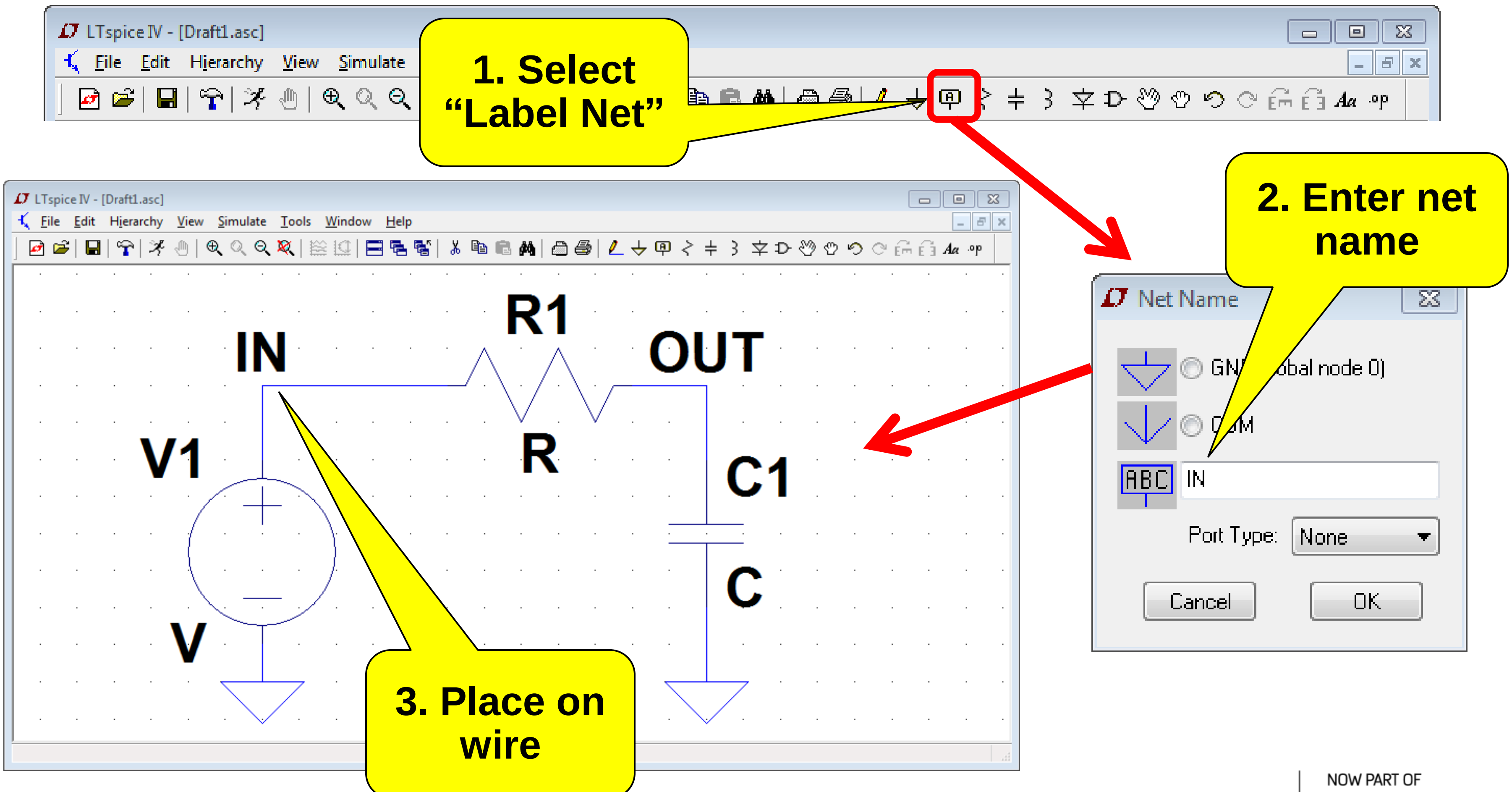
“Pull” wire down through the capacitor

Left-Click here ☐ to anchor & finish

Hint: Press the ESC key at any time to clean up the schematic

How to Wire up a Simple RC Circuit (cont.)

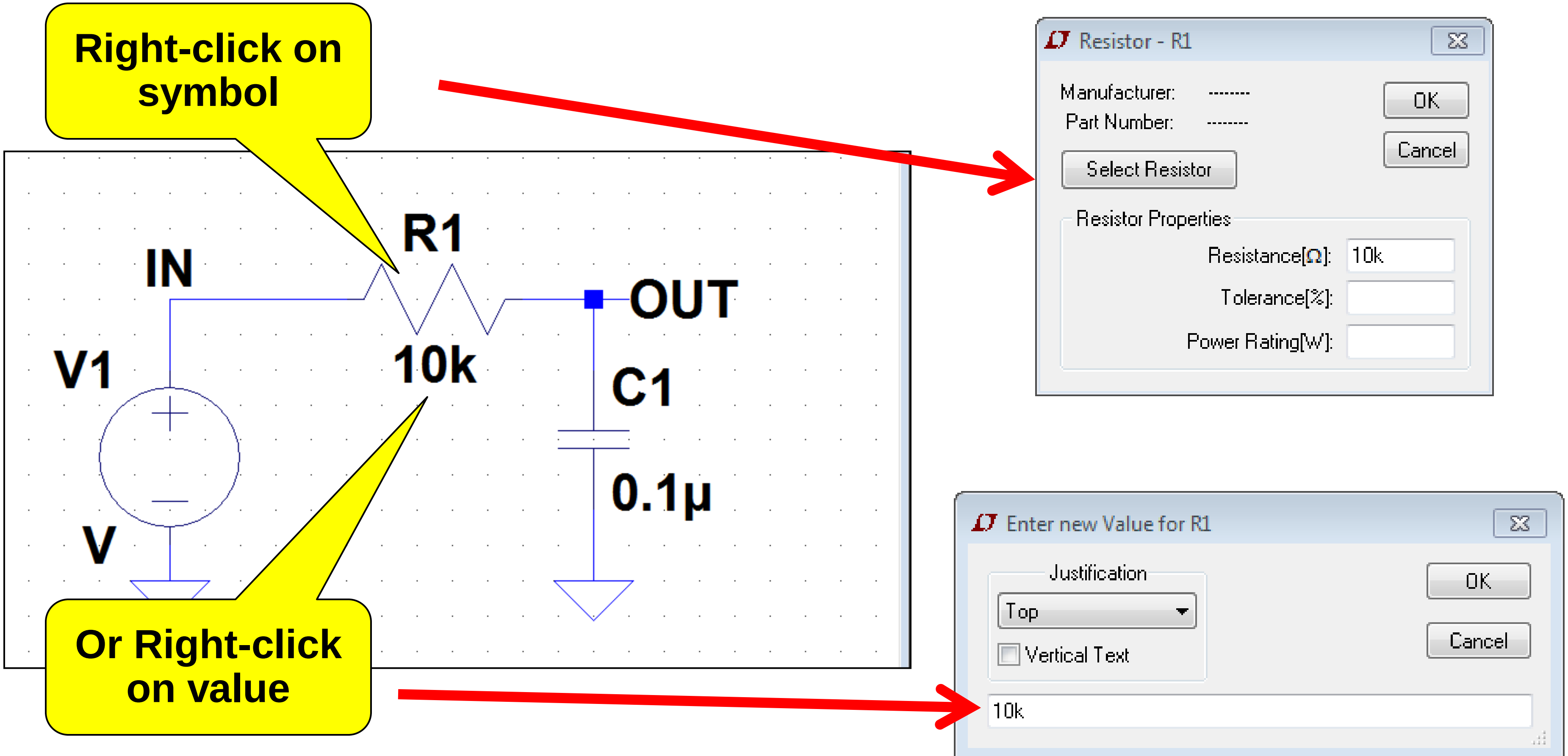
- Step 5: Add net labels
  - Using the toolbar, select Label Net. Label the input/output nodes as shown below



NOW PART OF ANALOG DEVICES

How to Wire up a Simple RC Circuit (cont.)

- Step 6a: Component values
  - **Right-Click** on each component symbol to change its value as shown below



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}

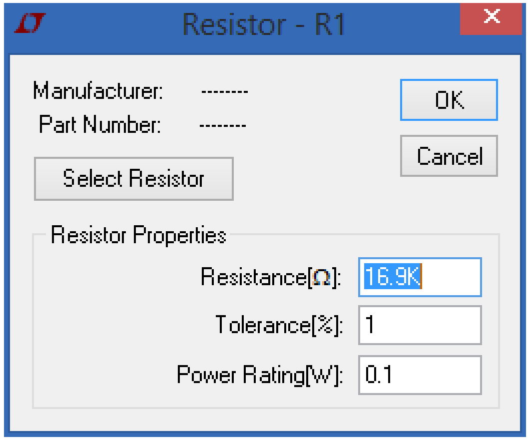
- M = m = milli = 10-3
- U = u = micro = 10-6
- N = n = nano = 10-9
- P = p = pico = 10-12
- F = f = femto = 10-15

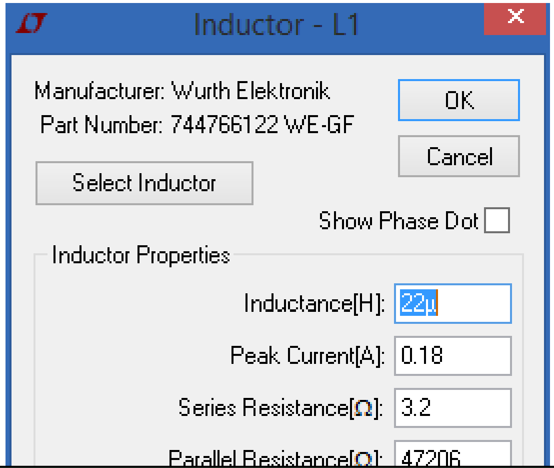
Hints

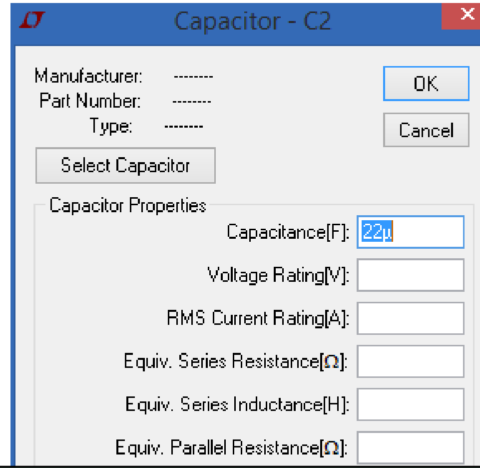
- **Use *MEG (or meg)* to specify 10^6, not *M***
- **Enter *1* for 1 Farad, not *1F***

Editing Components

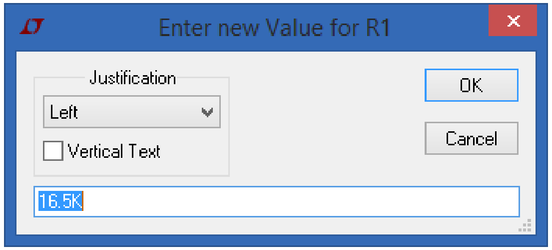
- **Right-Click** on the component to edit attributes

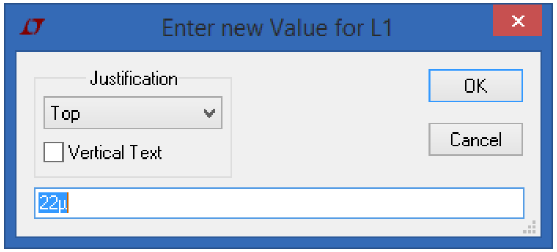


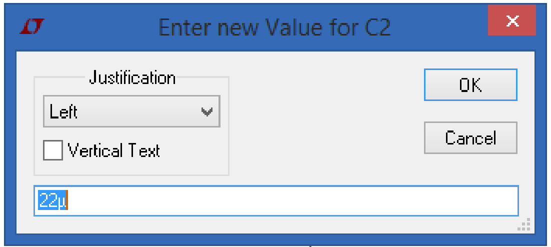




- You can also edit the visible attribute and label by pointing at the text with the mouse and then right-clicking
  - Mouse cursor will turn into a text caret



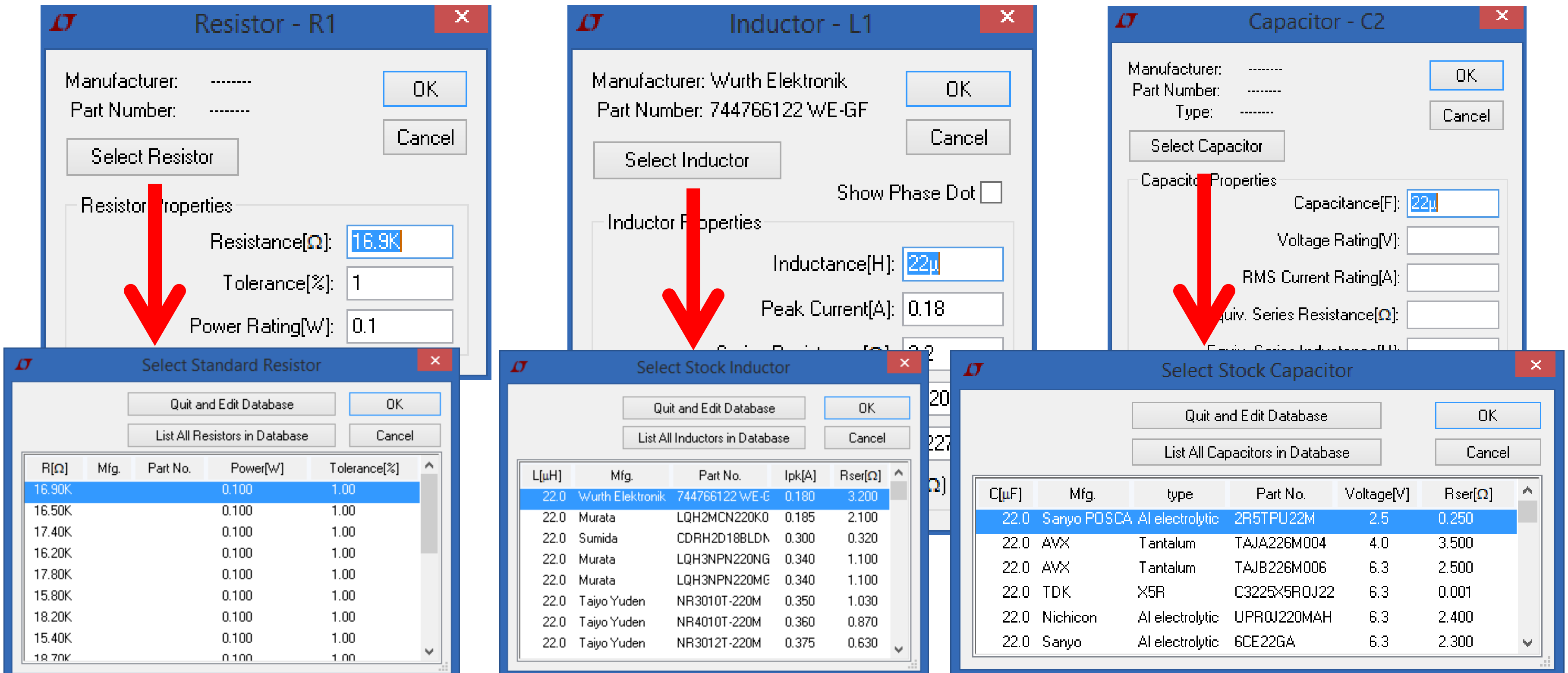




LINEAR TECHNOLOGY NOW PART OF ANALOG DEVICES

Component Database

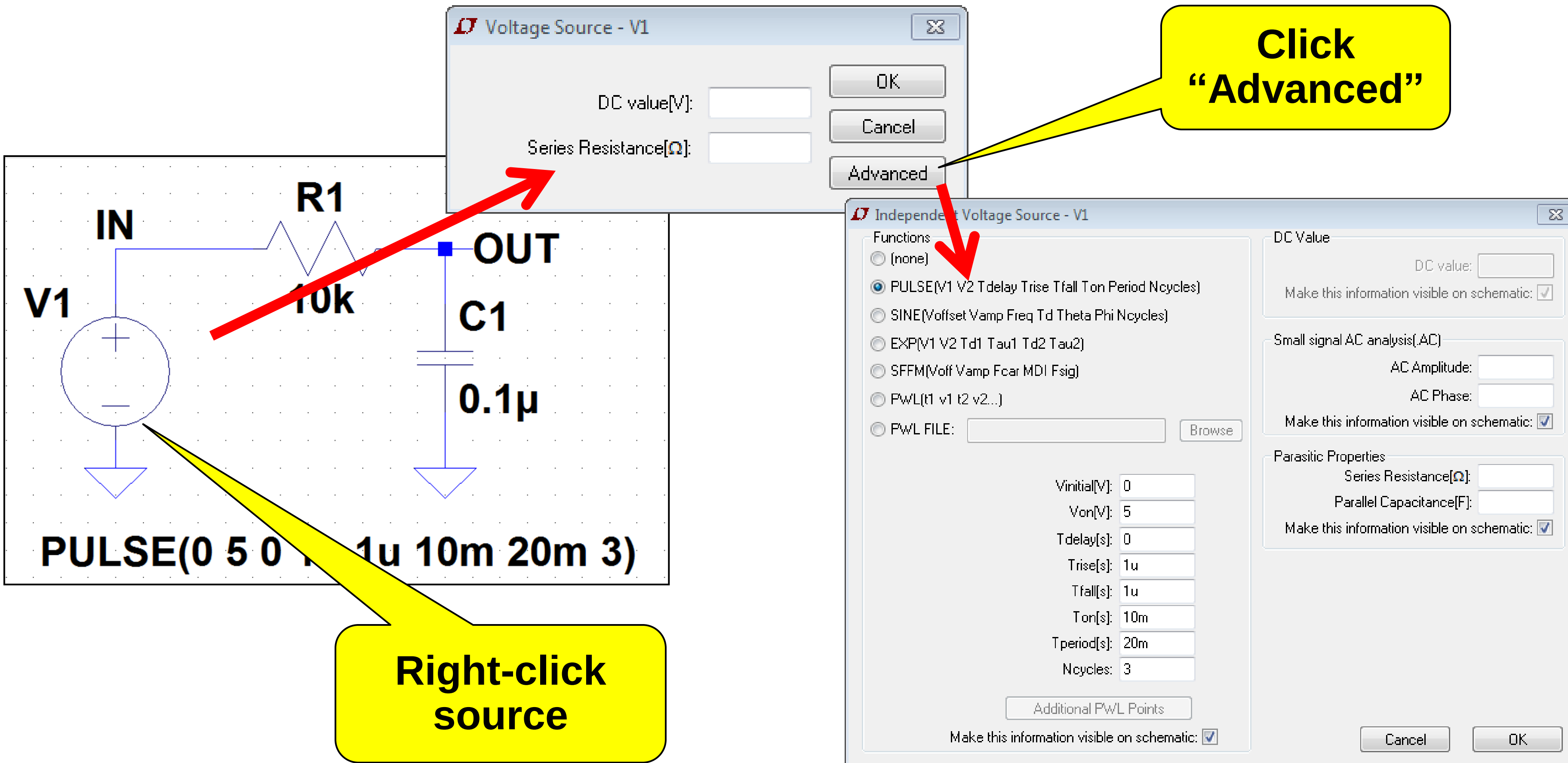
- Resistors, capacitors, inductors, diodes, Bipolar transistors, MOSFET transistors, JFET transistors, Independent voltage and current sources
- <u>You can access a database of known devices</u>



NOW PART OF

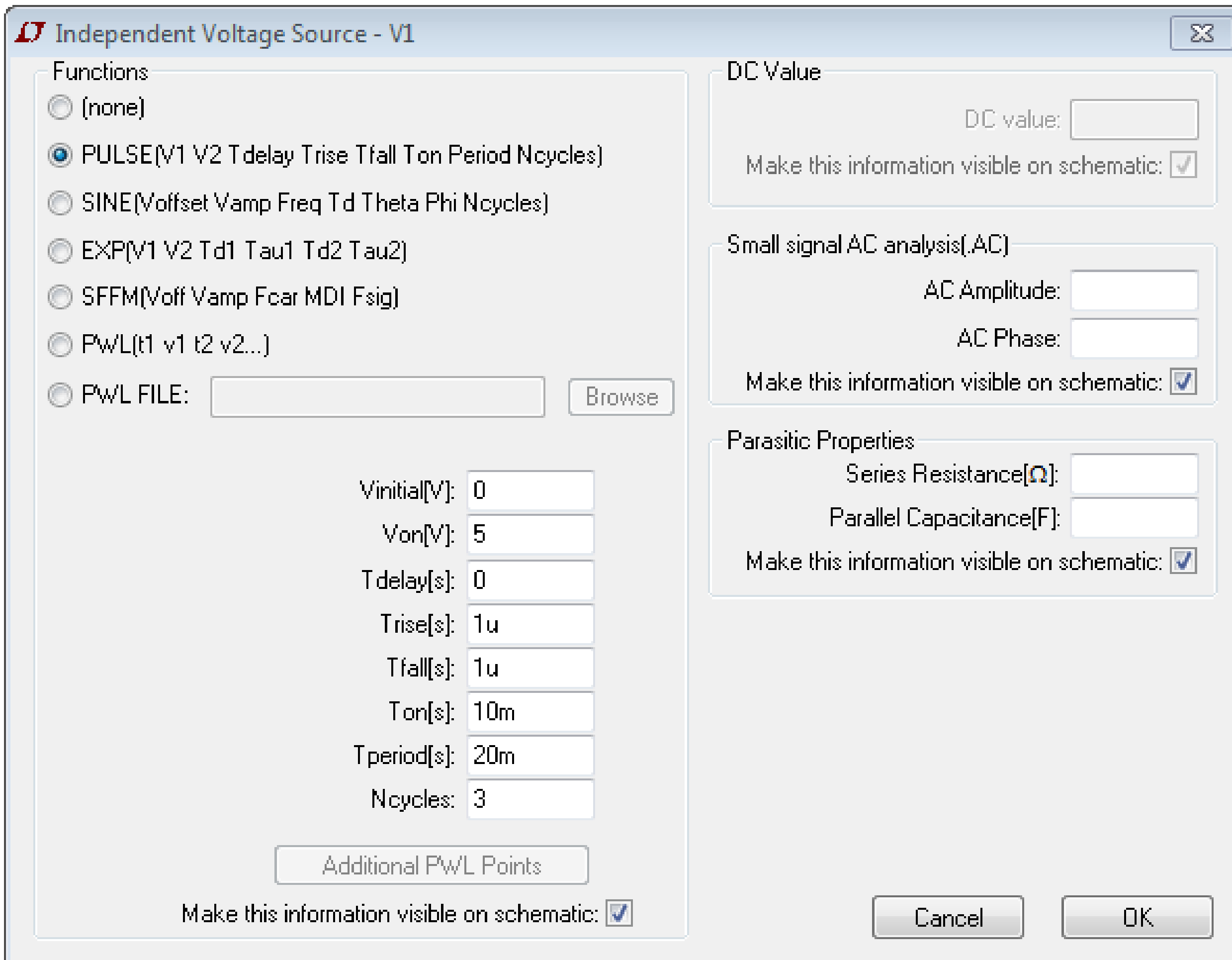
How to Wire up a Simple RC Circuit (cont.)

- Step 6b: Source parameters
  - **Right-Click** on the voltage source and enter the parameters shown below under the “Advanced” tab.



How to Wire up a Simple RC Circuit (cont.)

- Step 6b: Source parameters
  - **Right-Click** on the voltage source and enter the parameters shown below under the “Advanced” tab.



Running and Probing a Circuit in LTspice

Running a Circuit Simulation

- Access the LT8640 circuit
  - Click File ---> Open, and navigate to the LTspice Lab folder on your desktop. Look for the file titled "LT8640DCLoad.asc"
  - Or click "c" symbol on the right

LT8640DCLoad.asc C

- Hotlink Nomenclature:

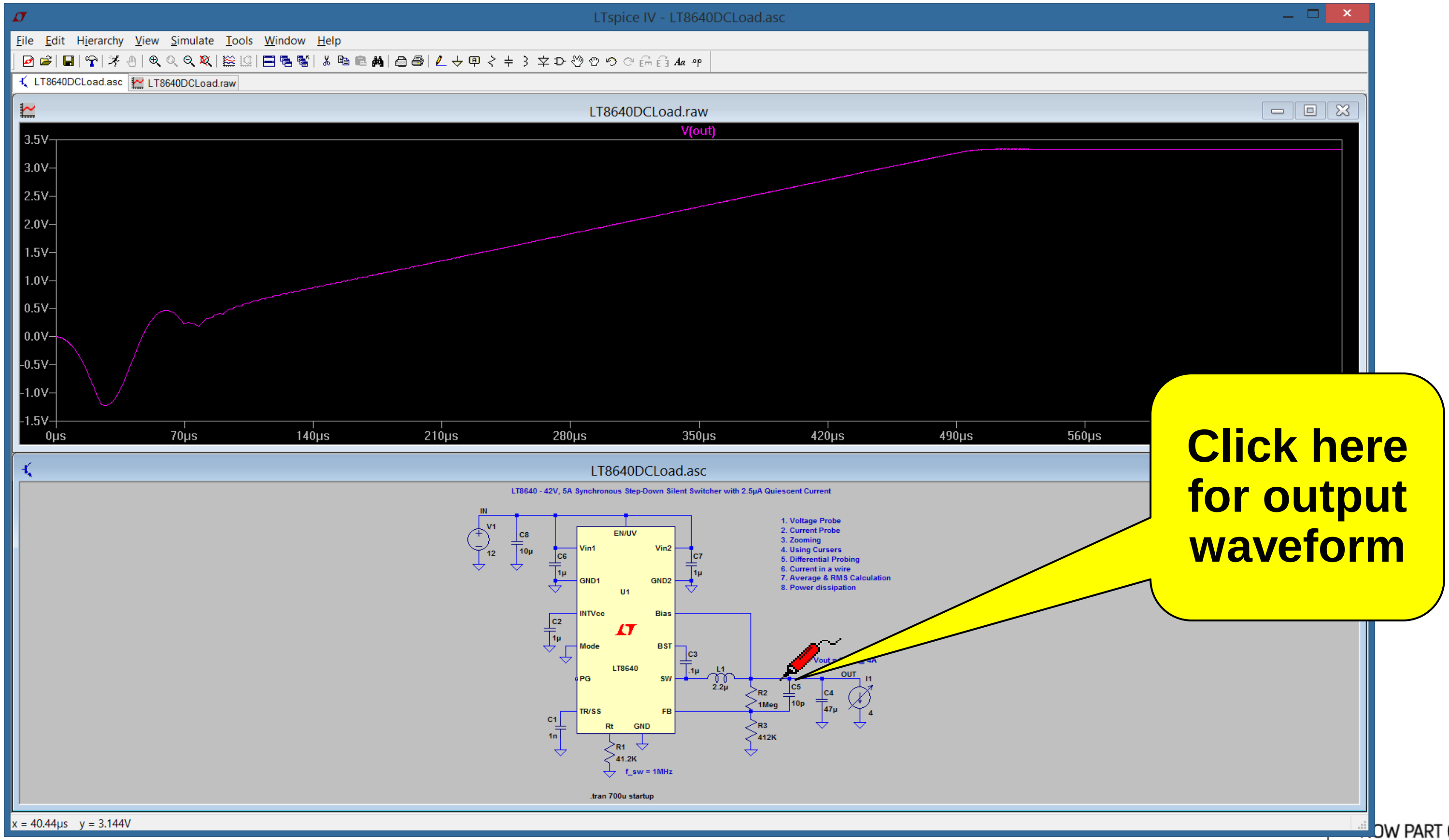
C Class exercise

S Solution to exercise

Circuits to explore at your leisure

Viewing Voltage Waveforms

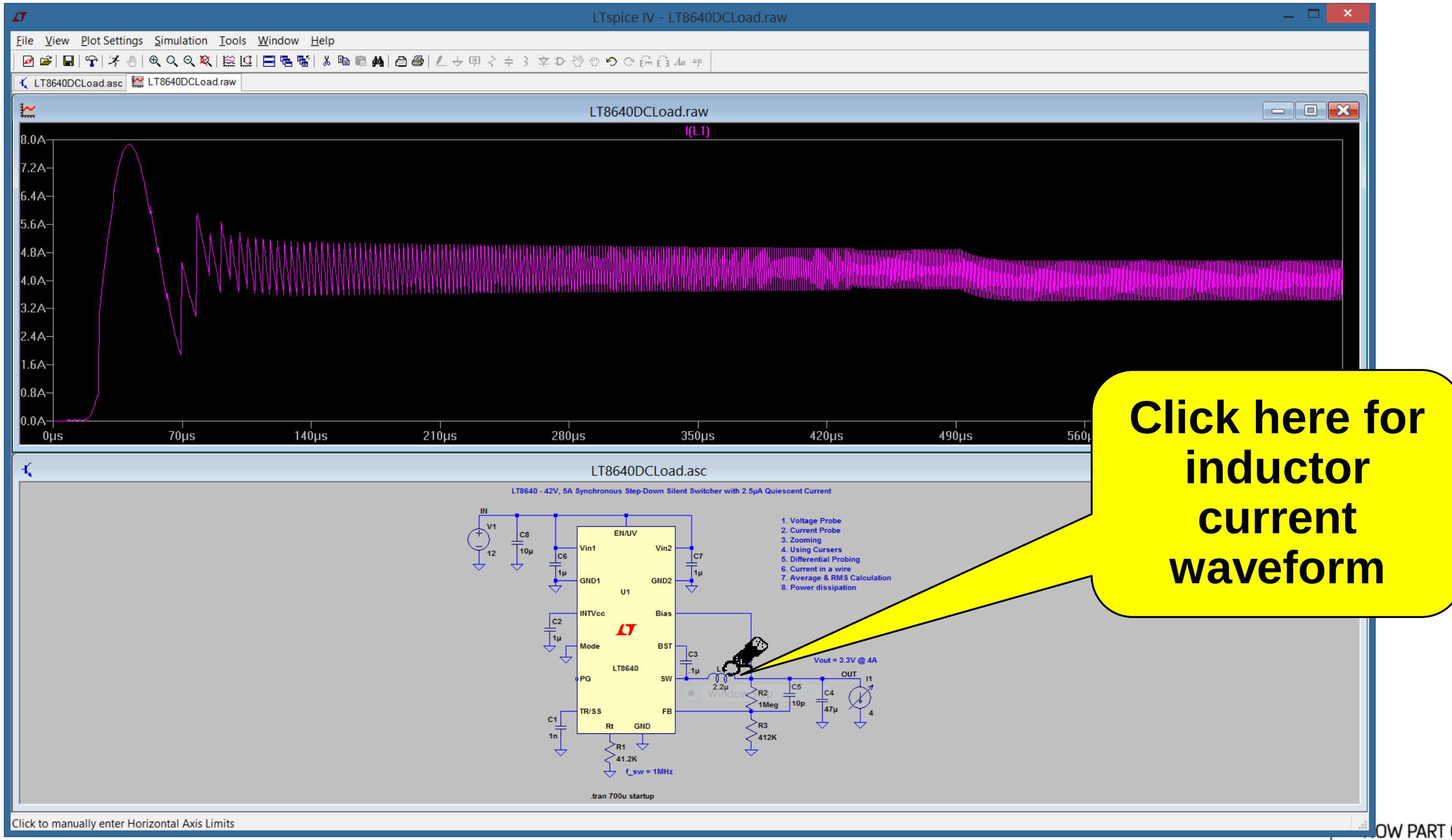
- Plot the voltage on any wire by **Left-Clicking** it
  - Tip: All Demo Circuits have INs and OUTs clearly labeled to help you quickly select them



LT8640DCLoad.asc

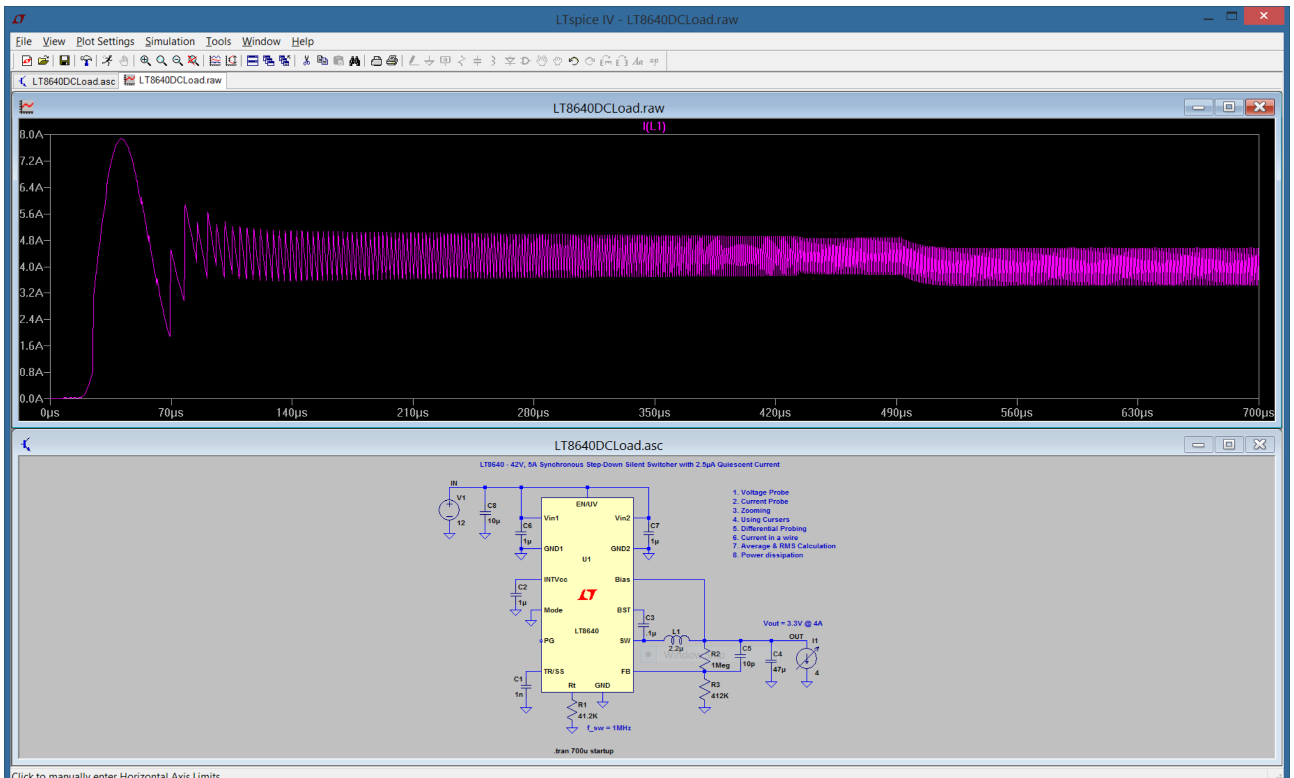
Viewing Current Waveforms

- Plot the current through any component by **Left-Clicking** on the body of the component
  - Current flowing into a node is defined as being positive



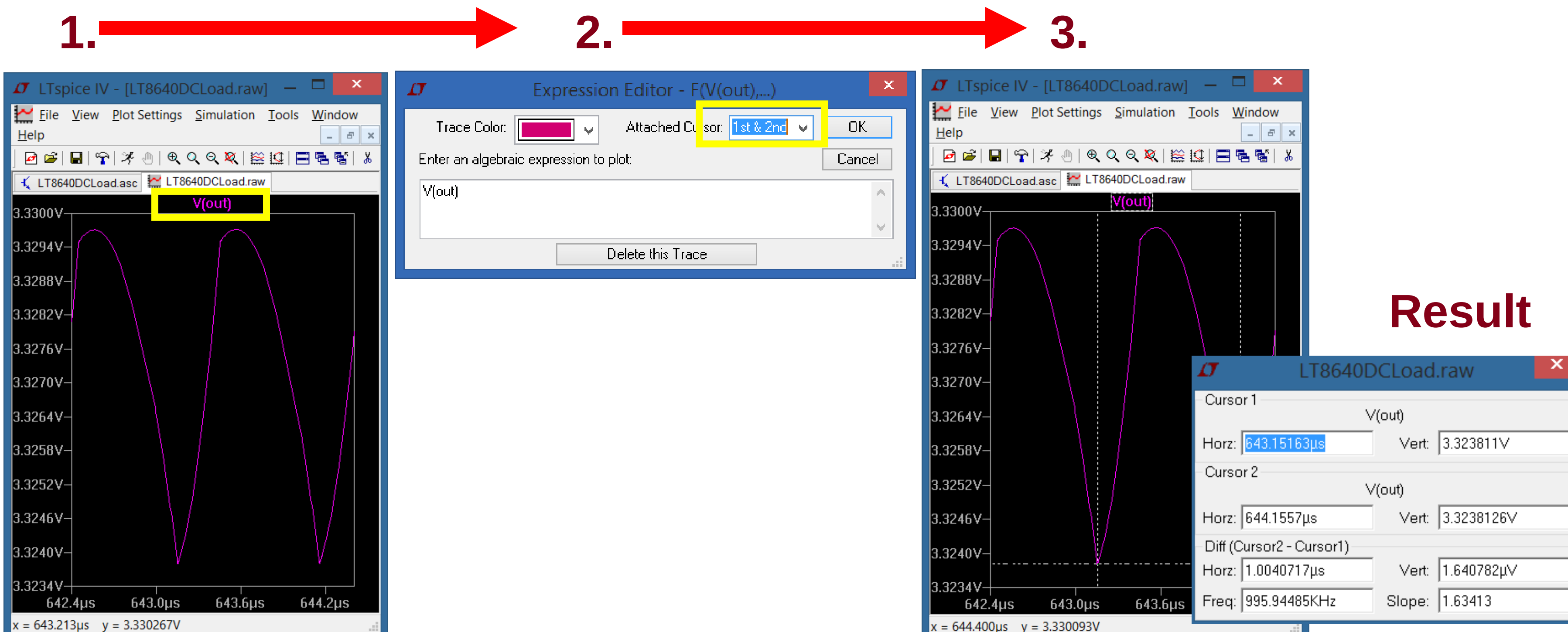
Zooming In and Out on a Waveform

- 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



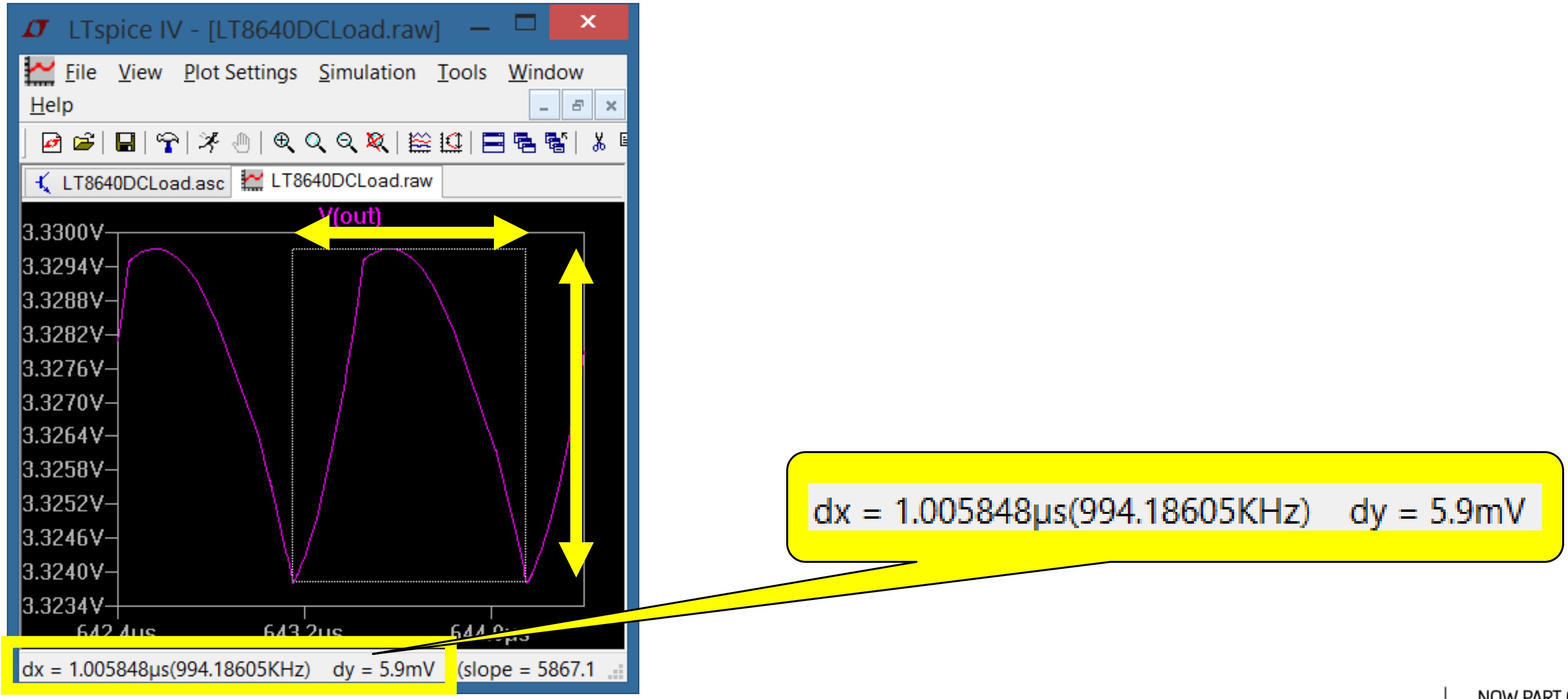
Measuring V, I and Time in the Waveform (Measurement Using Cursors)

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



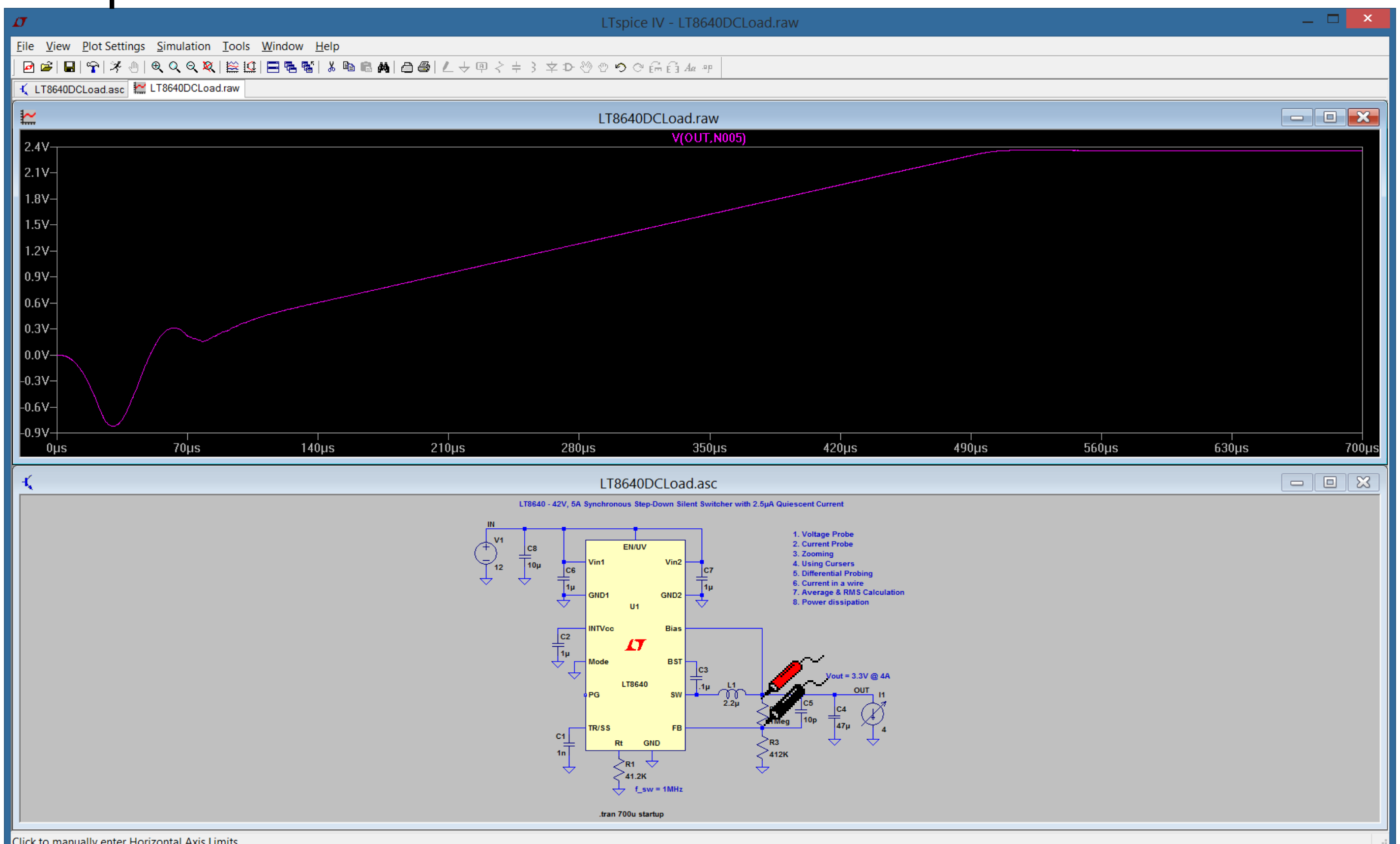
Measuring V, I and Time in the Waveform (Measurement Using Zoom Window)

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"



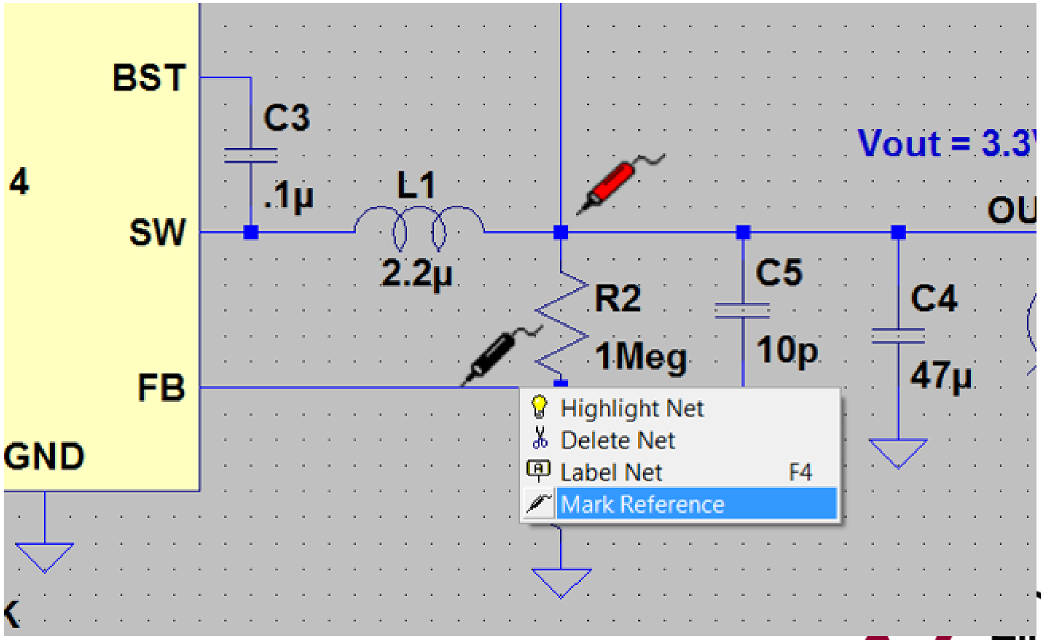
Viewing Differential Voltage Waveforms

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



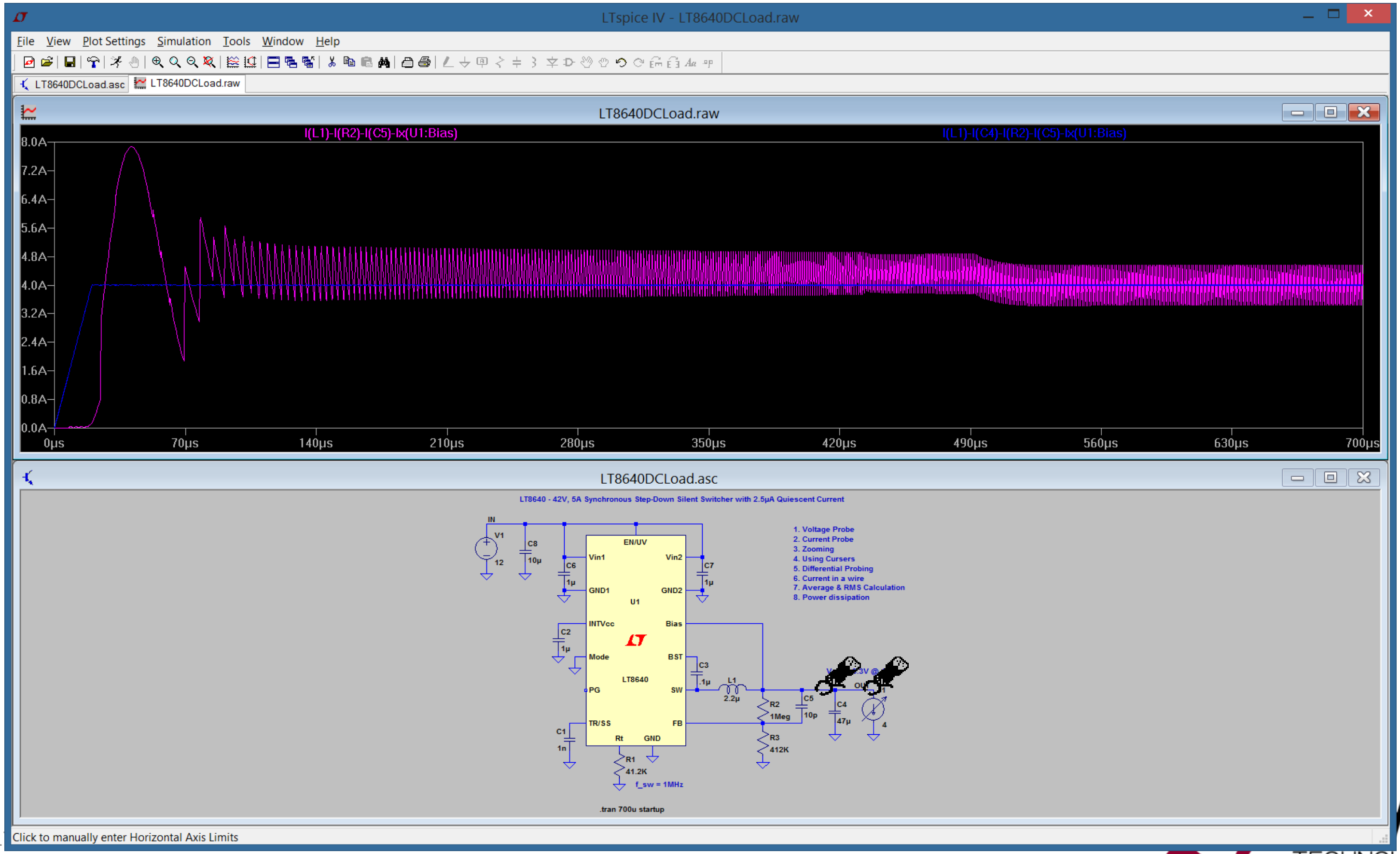
Viewing Differential Voltage Waveforms

- 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
- Hit the ESC key to remove the reference mark



Viewing Wire Current Waveforms

- Plot the current through any wire by **Alt-Left-Clicking** on the wire
  - An ammeter will appear to indicate that the wire current will be displayed

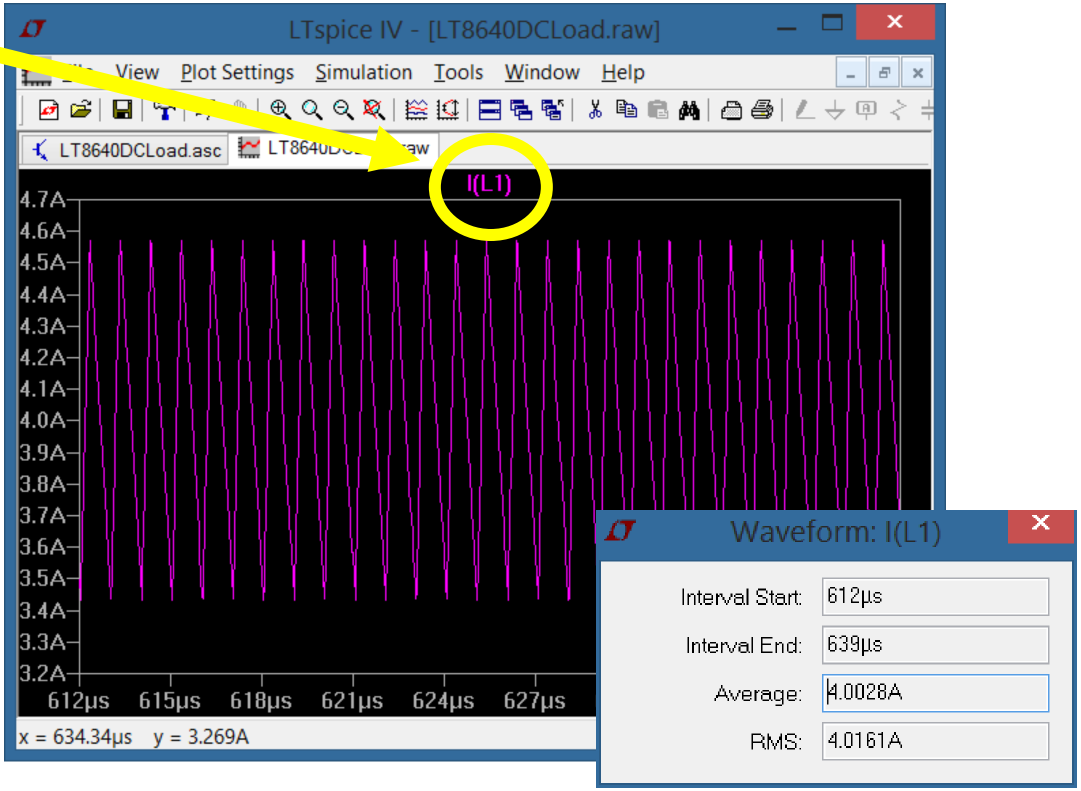


Average & RMS Calculations

- Average & RMS Current, Voltage, or Power Dissipation
  - Calculated only for the visible area of the plot window
- Click on inductor L1 to display the inductor current waveform
  - **Ctrl-Left-Click** the I(L1) trace label in the waveform view

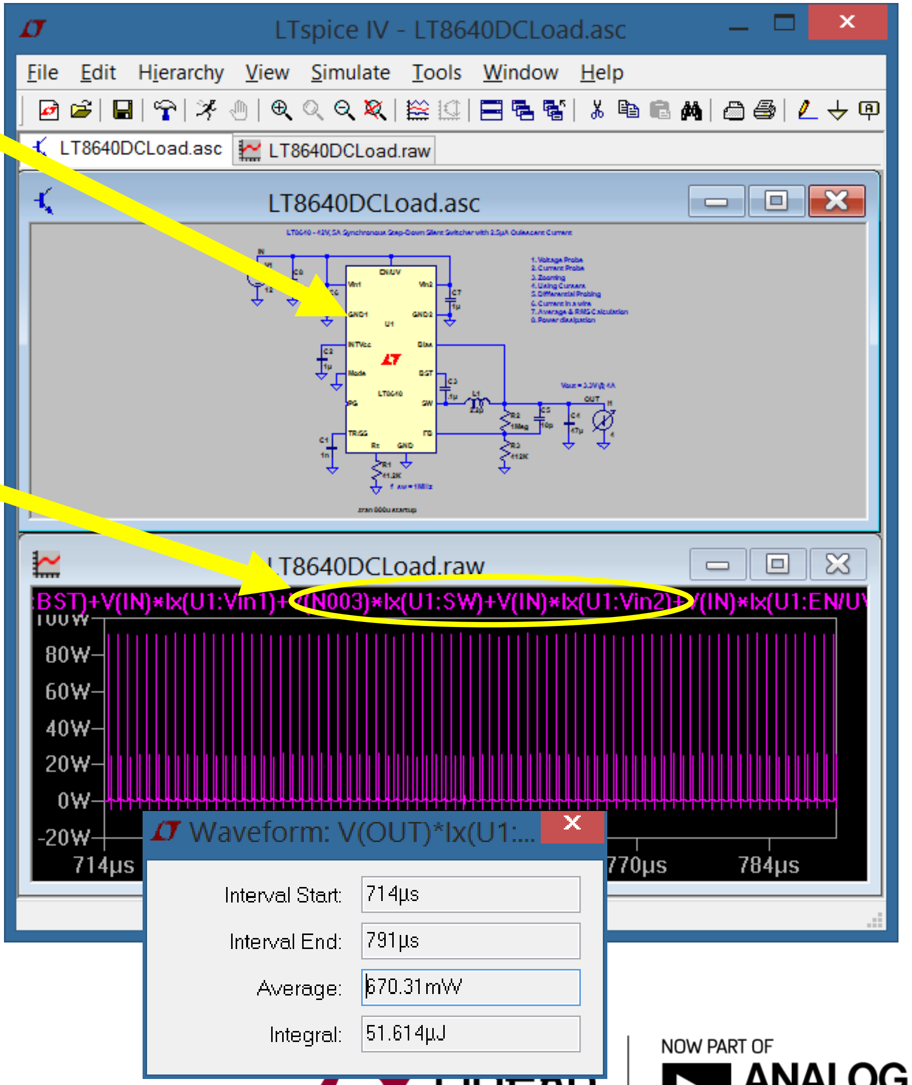
Example:

Measure average and RMS current for inductor in LT8640 circuit. Zoom in as shown for this waveform.



Instantaneous & Average Power Dissipation

- Instantaneous Power Dissipation
  - **Alt-Left-Click** on the symbol of the LT8640
  - 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

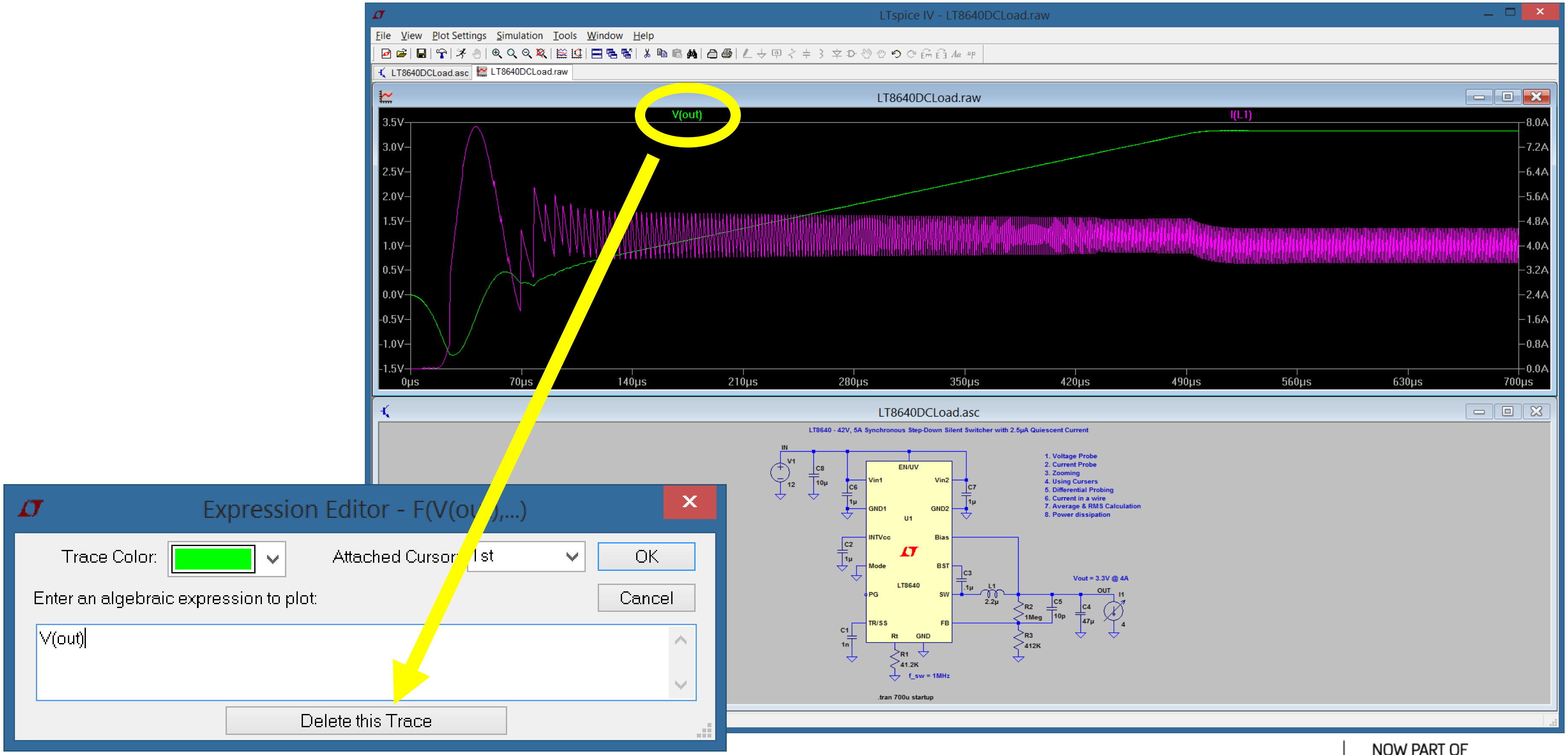


Example:

Measure the power dissipation in the LT8640 IC

Deleting Waveforms

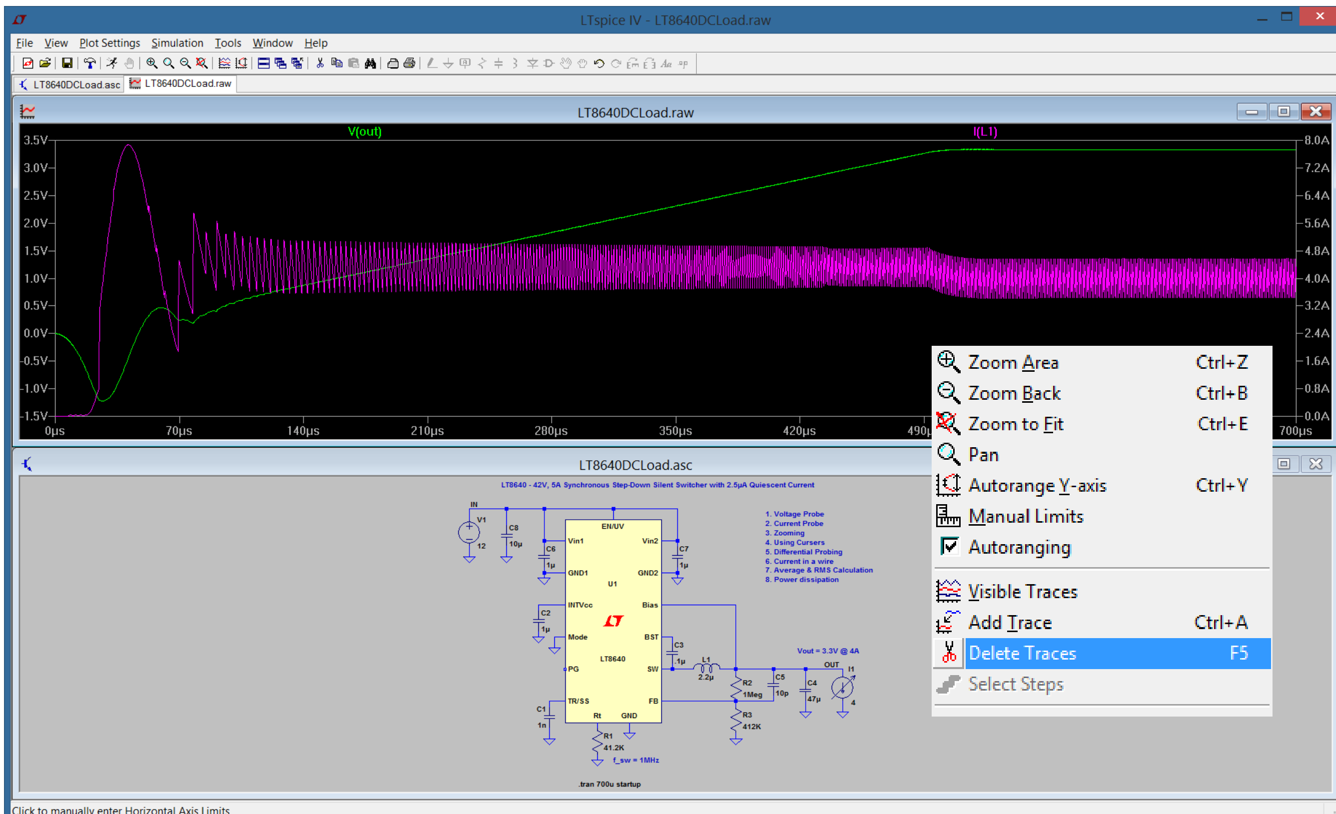
- Method #1: **Right-Click** on a trace label to be deleted
  - Select “Delete this Trace”
  - Deletes only the selected trace



LINEAR TECHNOLOGY NOW PART OF ANALOG DEVICES

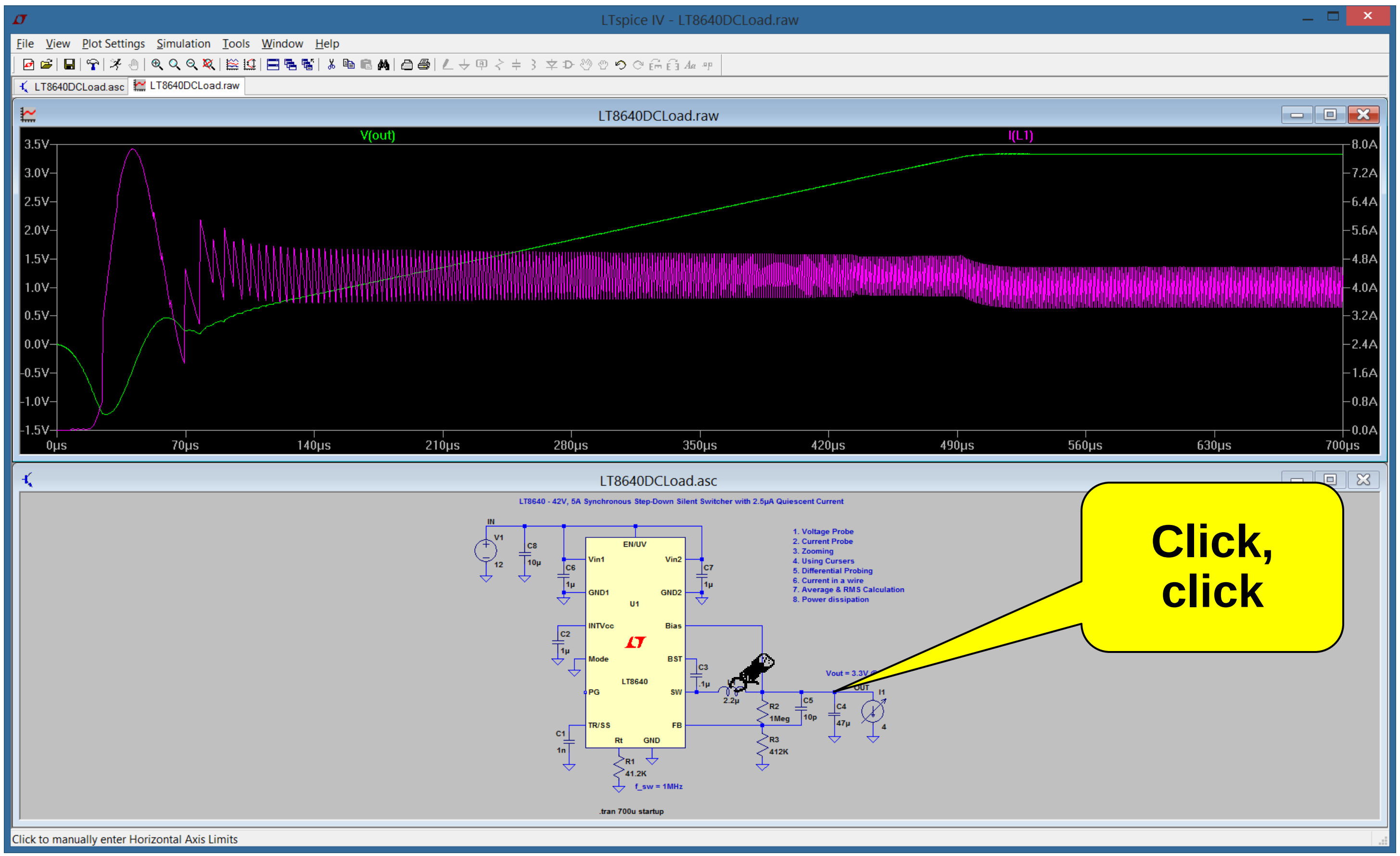
Deleting Waveforms

- Method #2a: **Right-Click** on the plot background and select “Delete Traces”
- Method #2b: If the plot window is active hotkey F5 is equivalent
  - Cursor turns into scissors
  - **Left-Click** on one or more trace labels to delete. ESC(ape) to quit



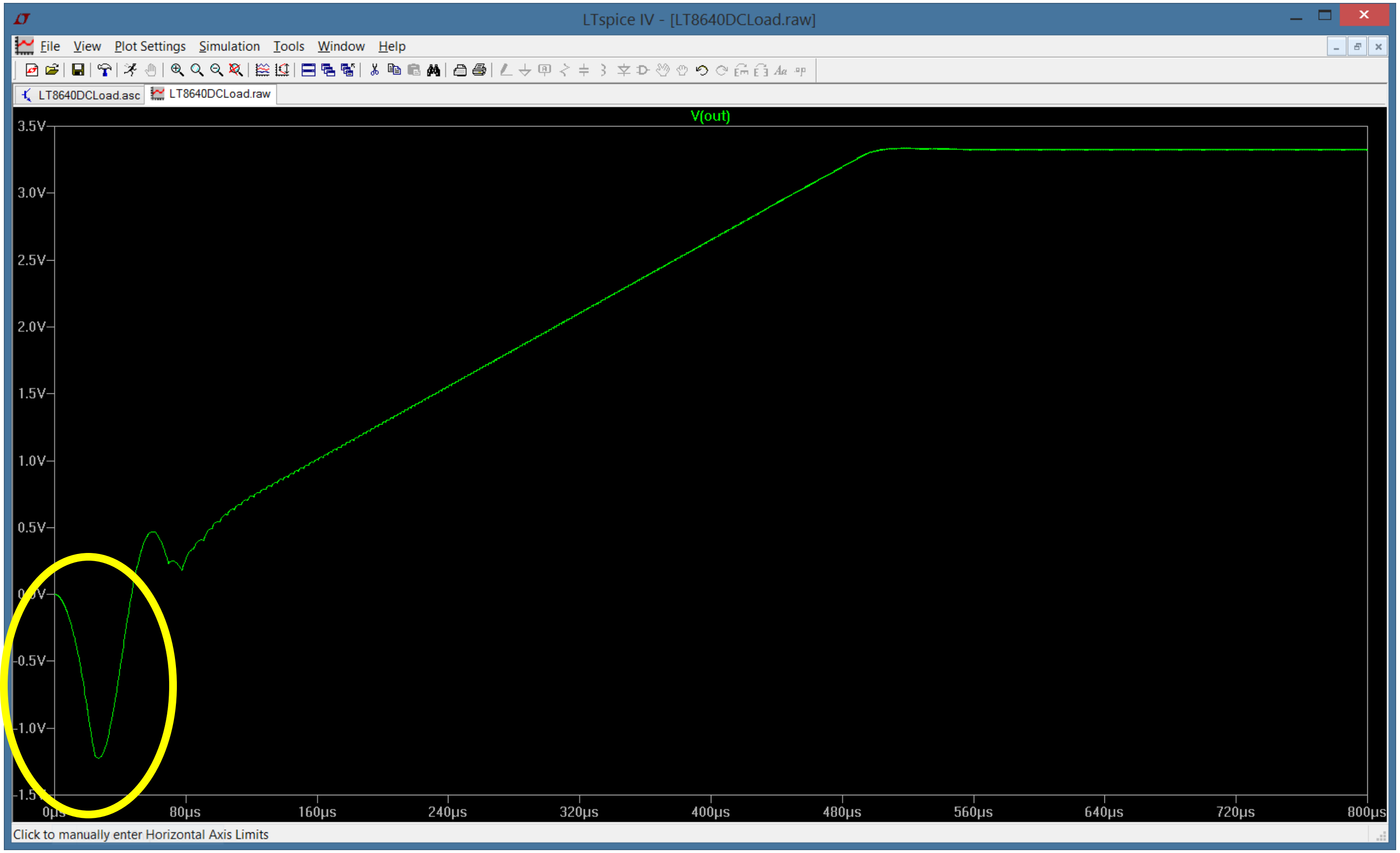
Deleting Waveforms

- Method #3: Plot the same waveform twice in succession
  - Deletes all but that waveform



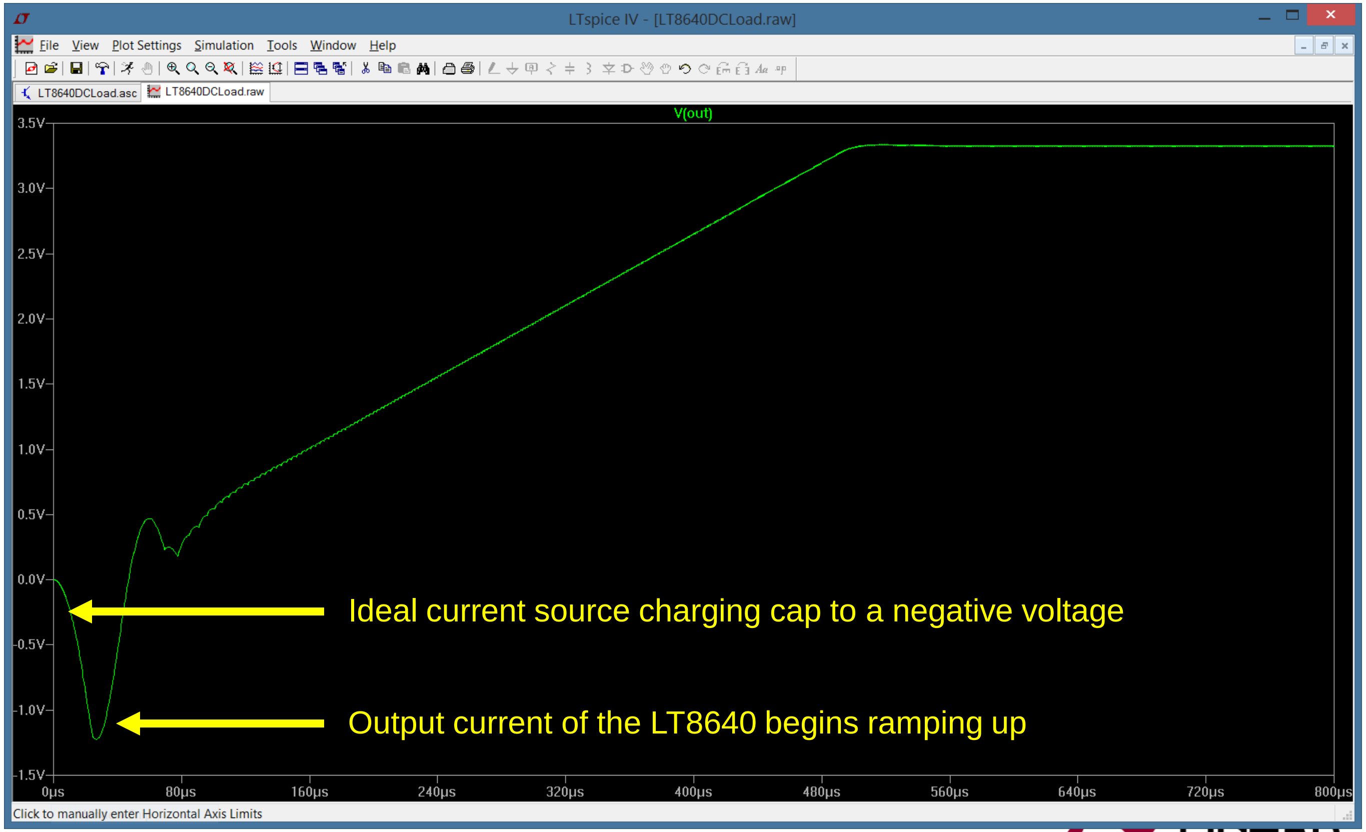
Current Sources

- The simulation shows the output momentarily going to -1.2V but this can't happen in the real world – what's going on?



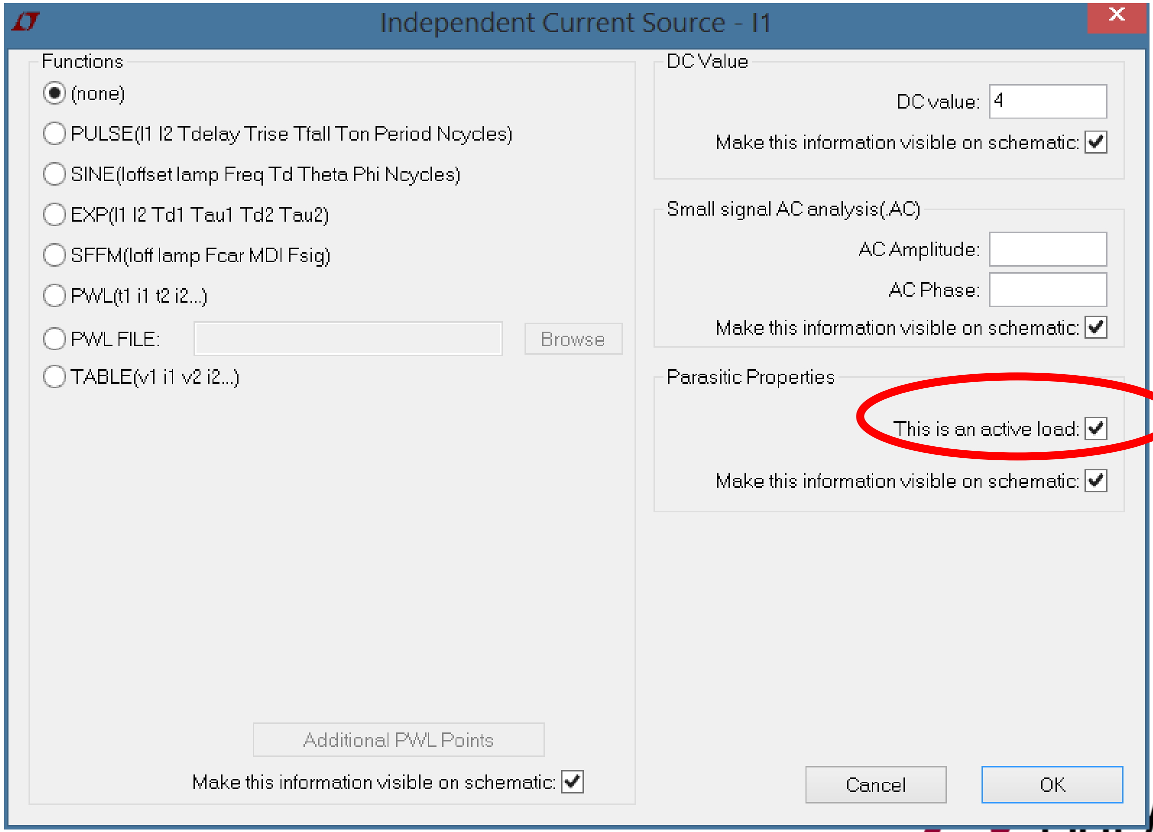
Current Sources (cont.)

- Ideal current sources source current even with zero or negative differential across them
  - This results in the output capacitor initially charging to a negative value



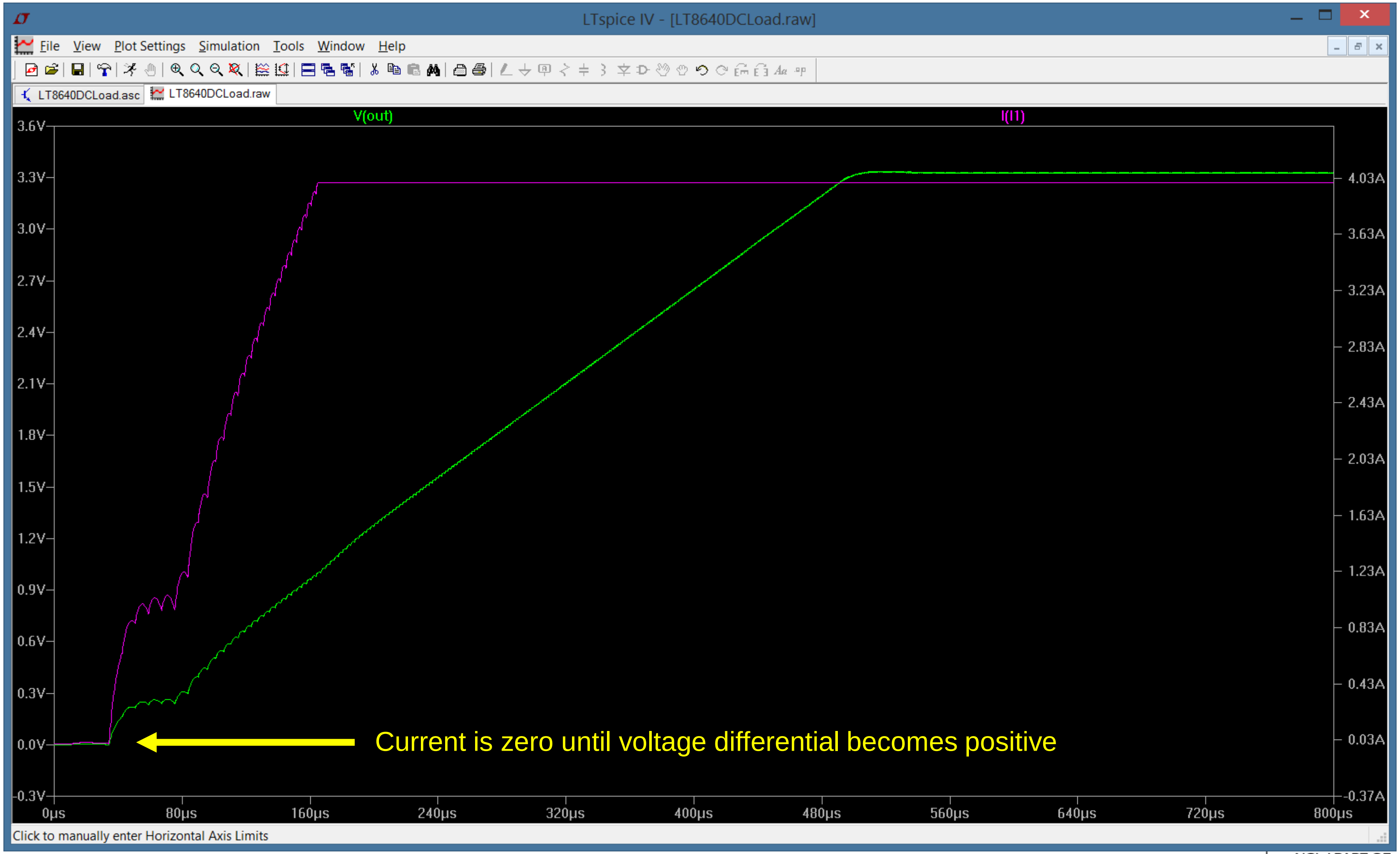
Current Sources (cont.)

- Active load
  - Modifies the ideal current source's behavior such that current is zero for zero or negative voltage differentials
  - Roughly emulates the behavior of an integrated circuit load



Current Sources (cont.)

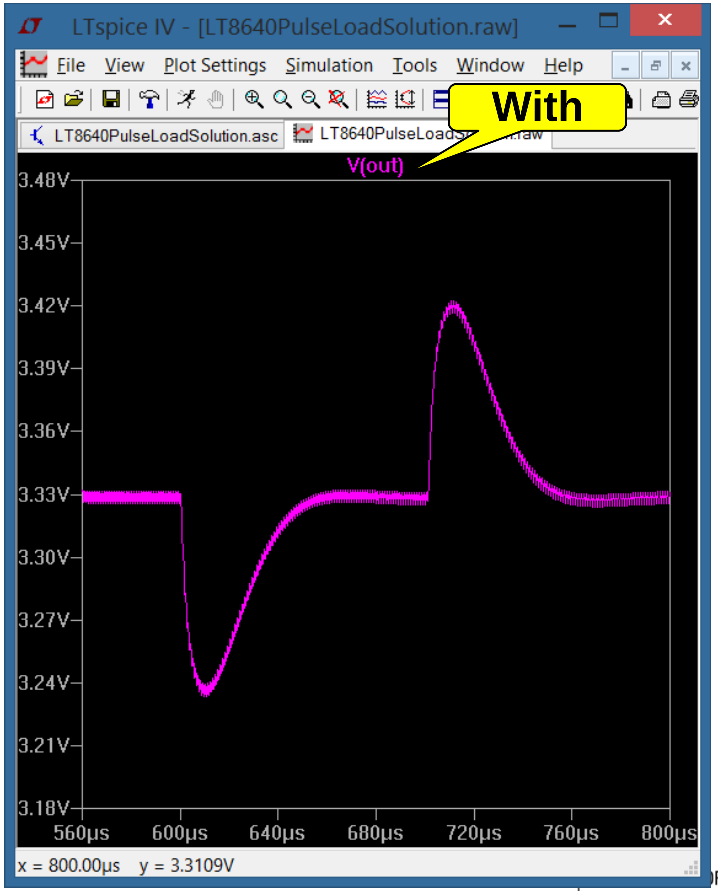
- Active load (cont.)



Net Labeling

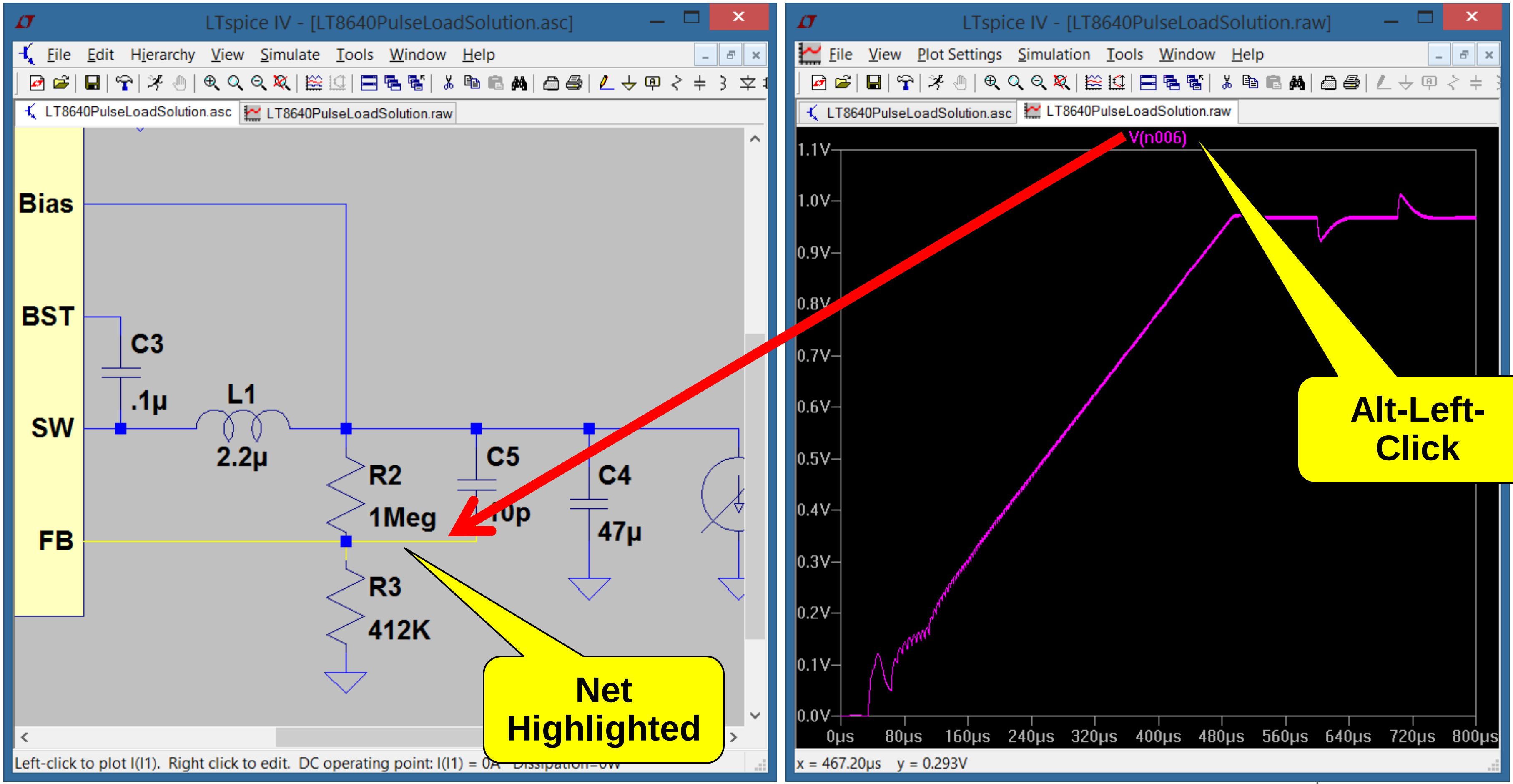
Advantages of Labeling

- 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”



Labeling - Trick

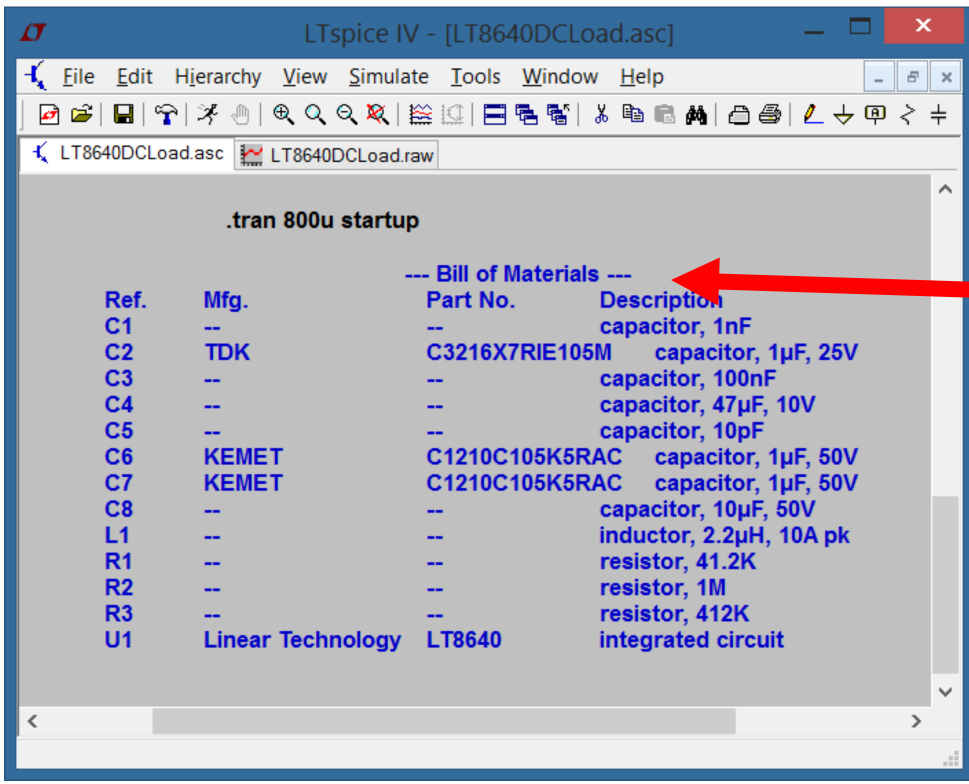
- Highlight net from waveform viewer
  - **Alt-Left-Click** on the label in the waveform viewer (i.e. V(n006)) and it will now highlight that particular net on the schematic

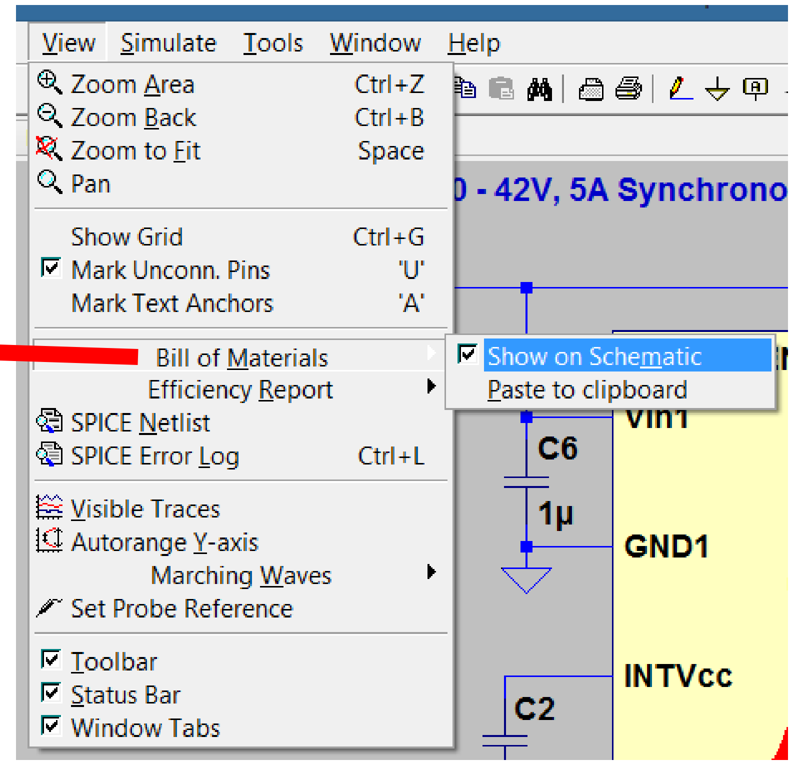


Generating a BOM and Efficiency Report

BOM

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



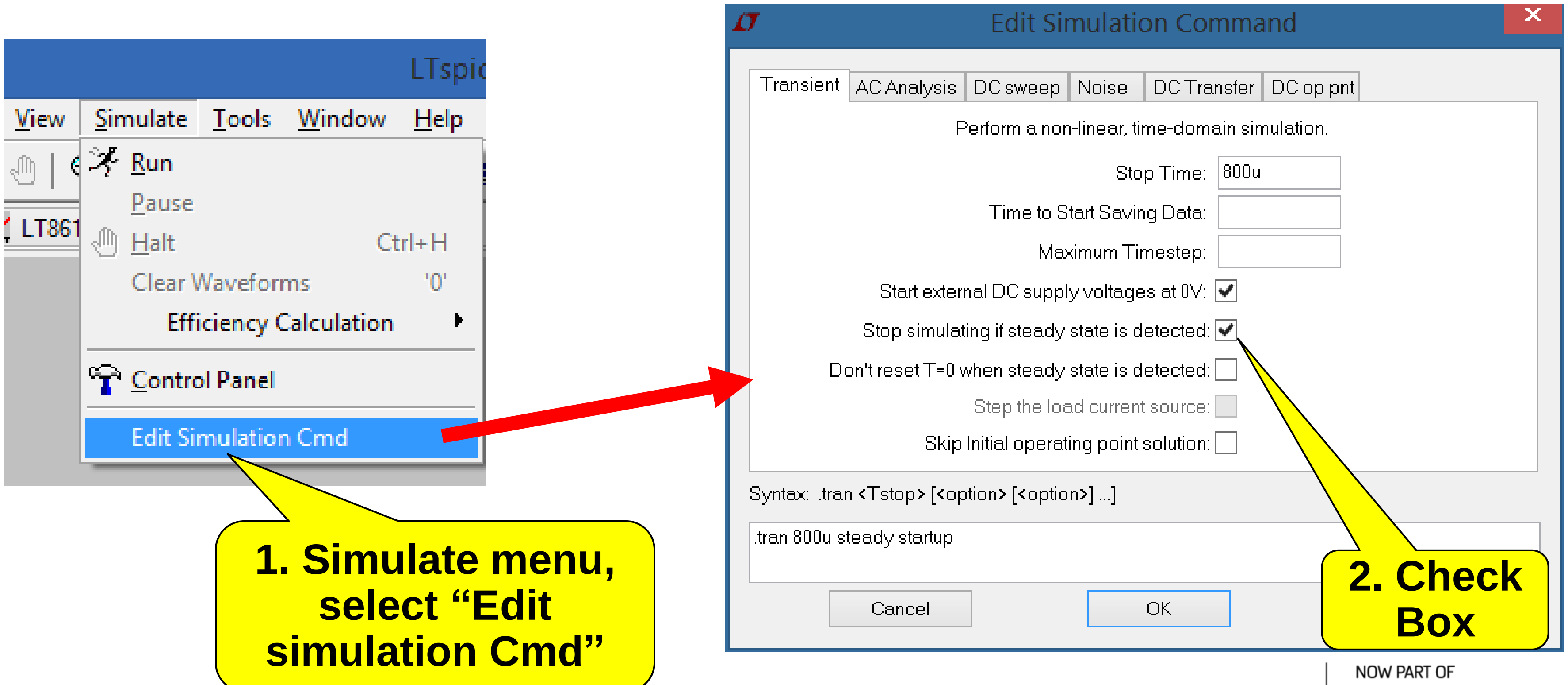


Steps to Computing Efficiency

Note: Efficiency will only be calculated in the steady state condition

1. **Right-Click** the .tran statement on the schematic to bring up the Edit Simulation Command dialog box

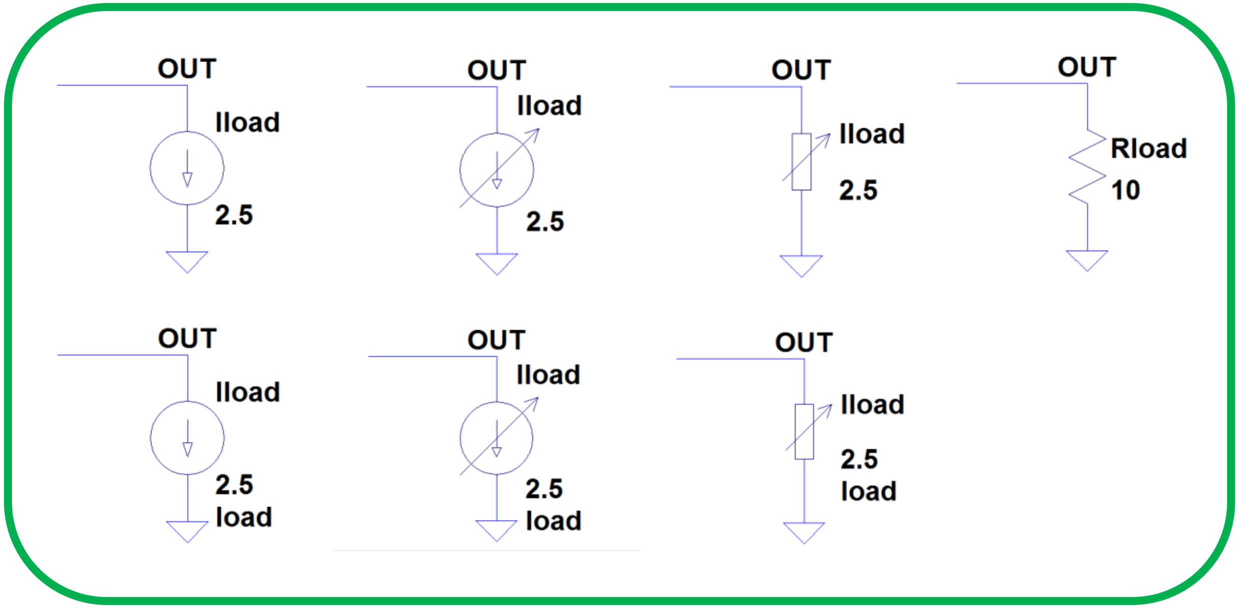
2. Check the box “Stop simulating if steady state is detected” …



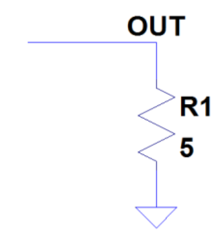
Steps to Computing Efficiency

3. Load must be a current source labeled Iload* or resistor labeled Rload**

The Good



The Bad



This will be treated just like any other resistor – efficiency will read **ZERO**

*If there is more than one current source

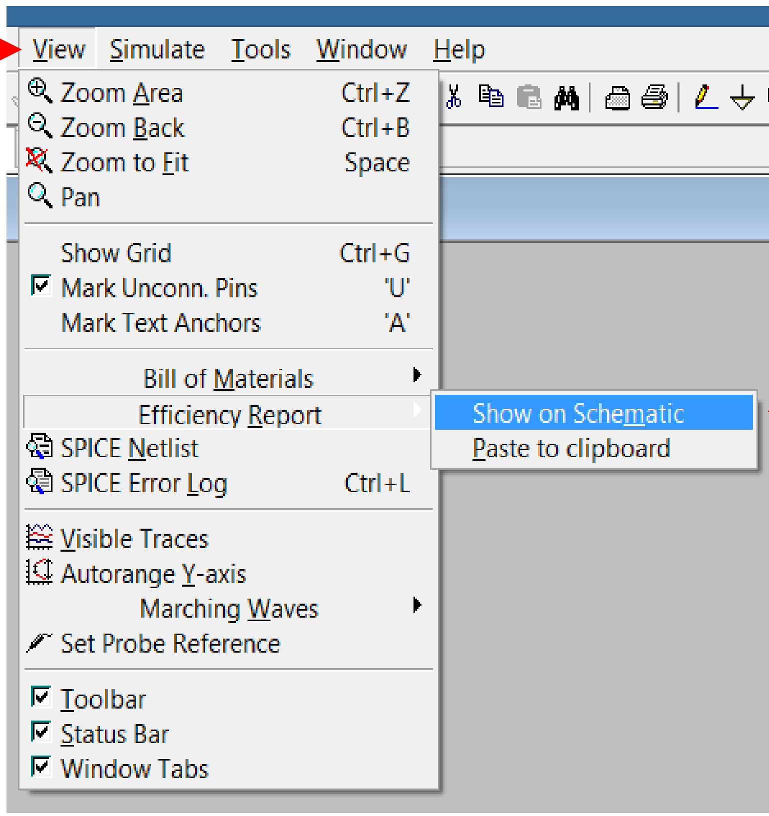
**If there is more than one resistor

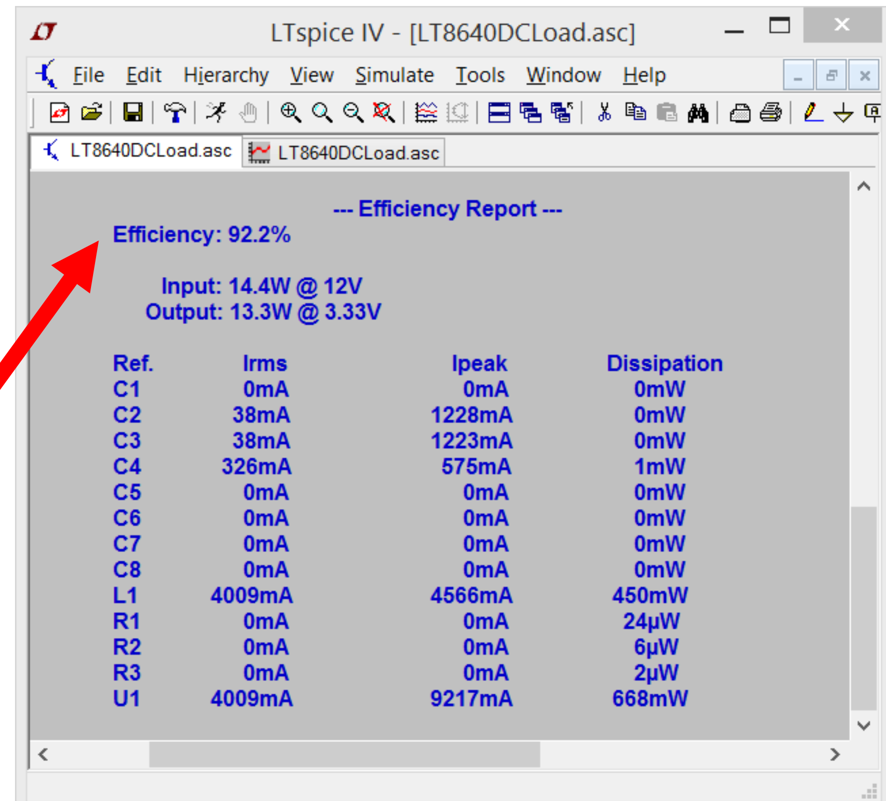
4. Run the simulation …

Steps to Computing Efficiency

5. Upon completion select the View dropdown menu, Efficiency Report, then Show on Schematic

6. Efficiency report will be pasted under the schematic





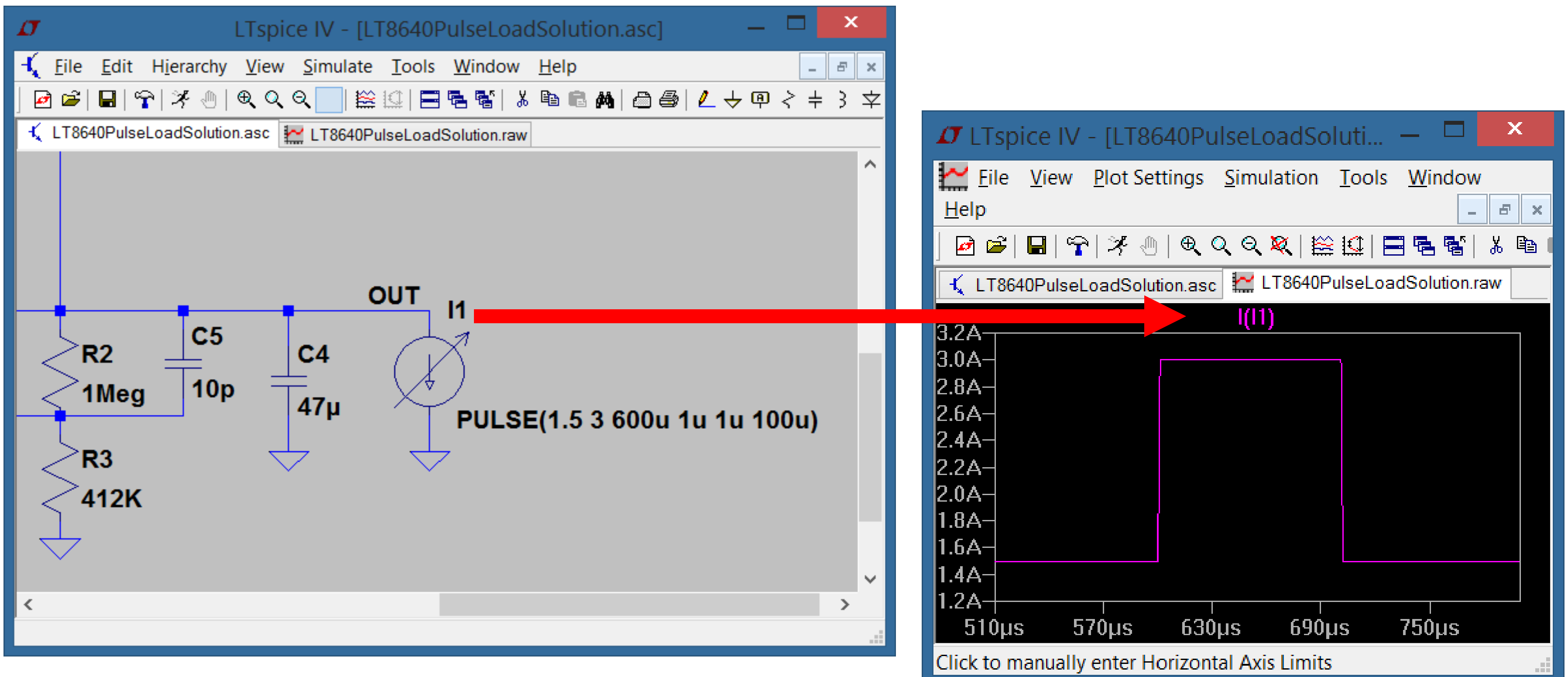
SMPS Efficiency Tips

- LTspice will not always be able to determine steady state, but this is rare!
  - Workaround: **Alt-Left-Click** & Calculator
- Probe the various nodes and verify the circuit is stabilized
  - If not edit the .tran statement and increase the Stop Time parameter. Re-run simulation
- For multiple output and/or multiple input supplies, efficiency must be determined partially by hand from the efficiency report
  - Alternatively use behavioral models
- **Right-Clicking** any component will report power dissipation if steady state has been detected or Mark Start/End has been used
- If circuit has stabilized for a long time and LTspice still hasn't detected the steady-state
  - Use Mark Start/End (Simulate -> Efficiency Calculation)
  - Only steady-state data is displayed before Mark End

Simulating a Transient Response

Current Load and Pulse Function

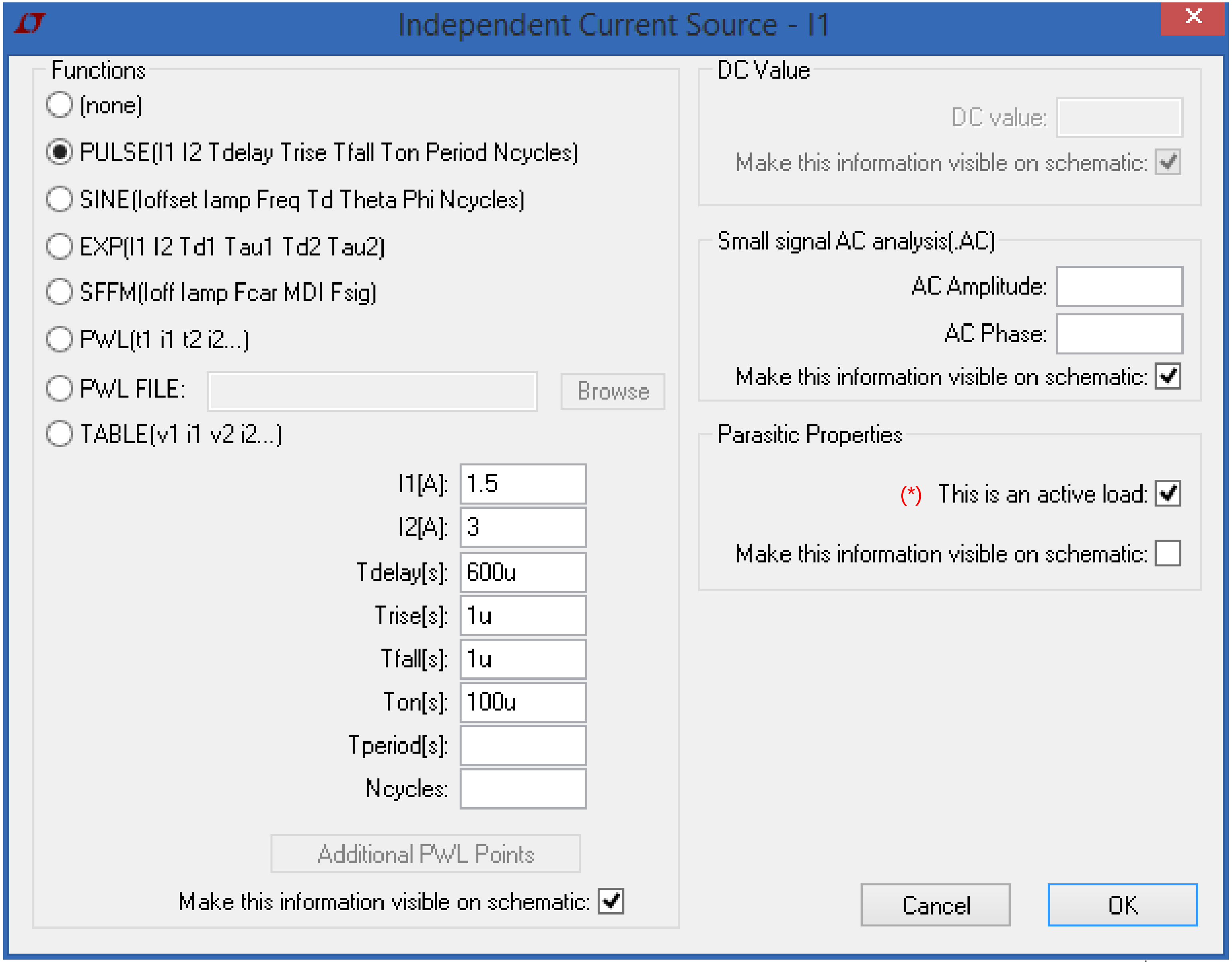
- You can simulate a load with a Resistor or Current load
- In particular the Pulse function in a current load is helpful in transient response analysis
  - Steps a current load from one value to another value



NOW PART OF
LINEAR TECHNOLOGY
ANALOG DEVICES

Edit the Current Load to a Pulse Function

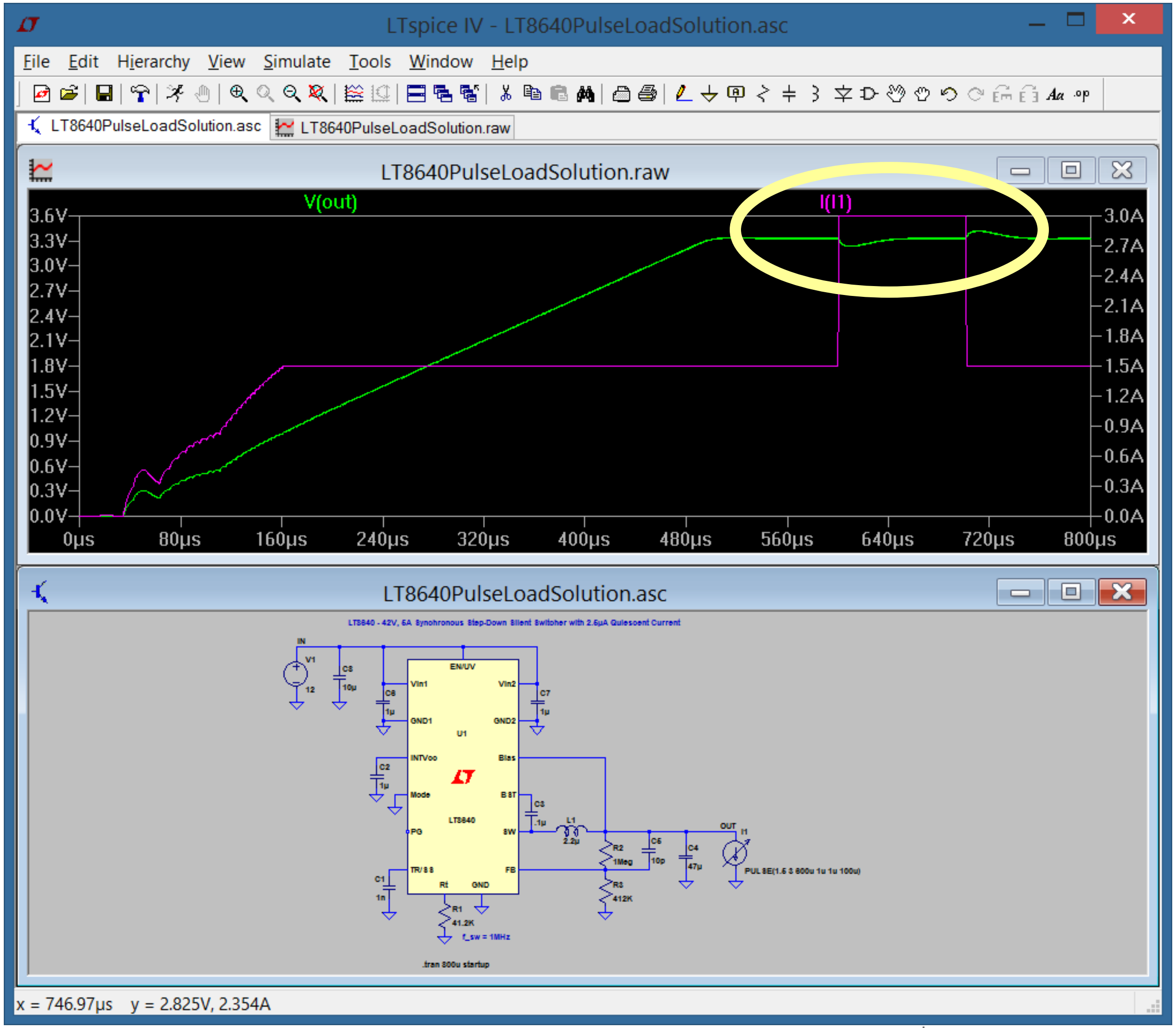
- Edit the .tran directive to disable steady state detection
- **Right-Click** on the current load
  - Select “Pulse”
  - Modify the attributes (see next slide). Click “OK”



(*) Forces current to be zero when voltage is zero

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



LT8640PulseLoadSolution.asc

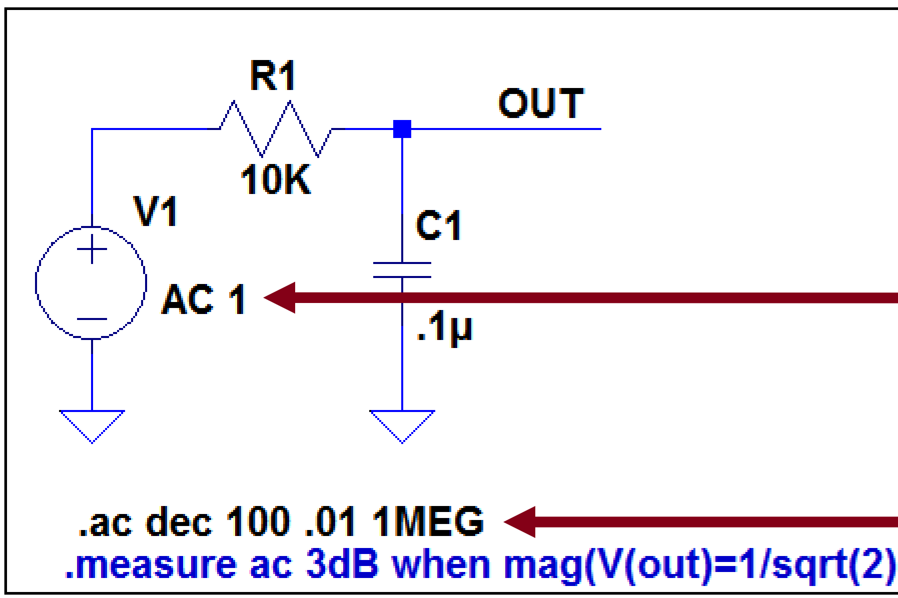
AC Analysis

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

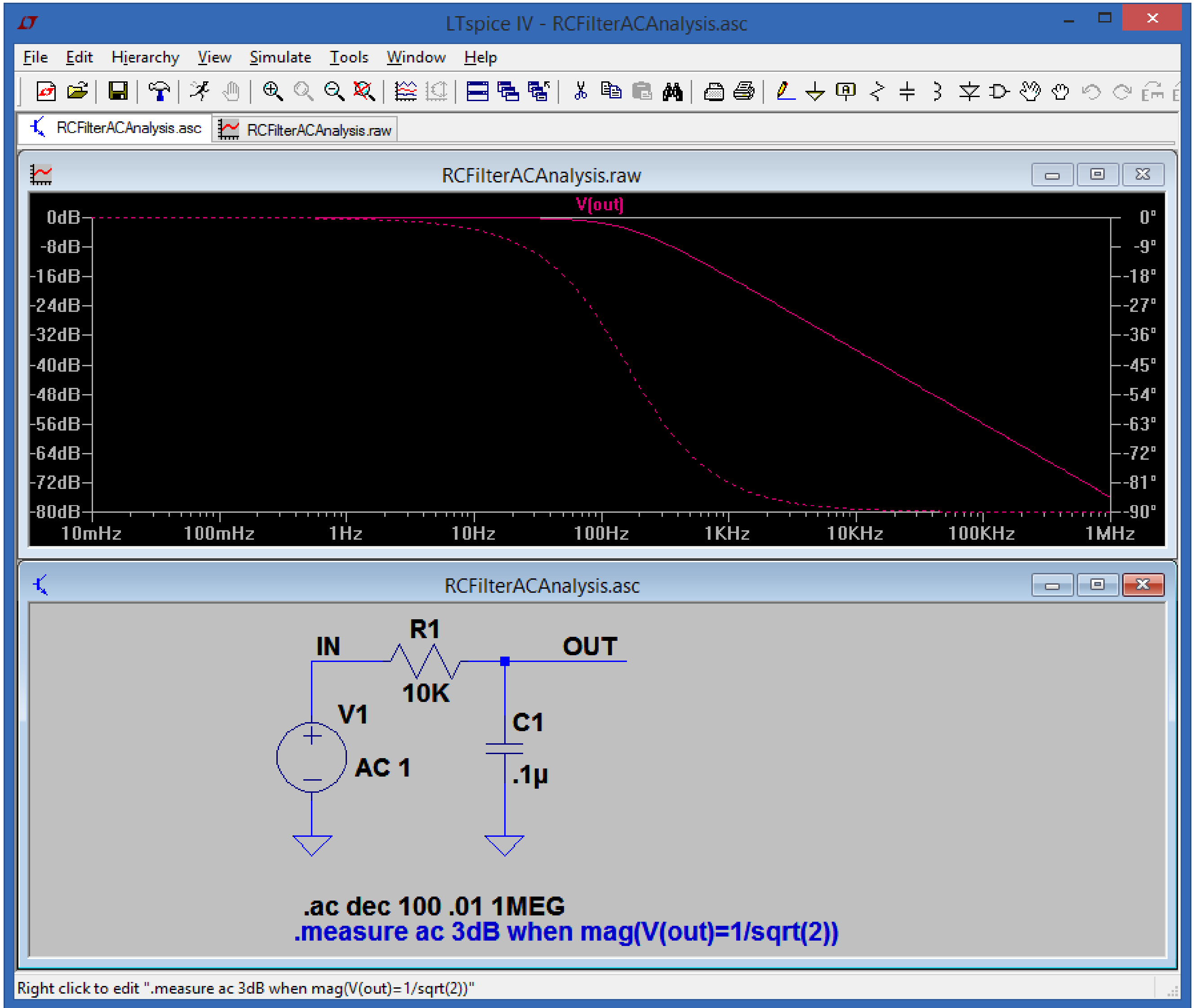


-3dB point:
1/(2*pi*R*C) = 159Hz

AC amplitude must be set to 1 for accurate results

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

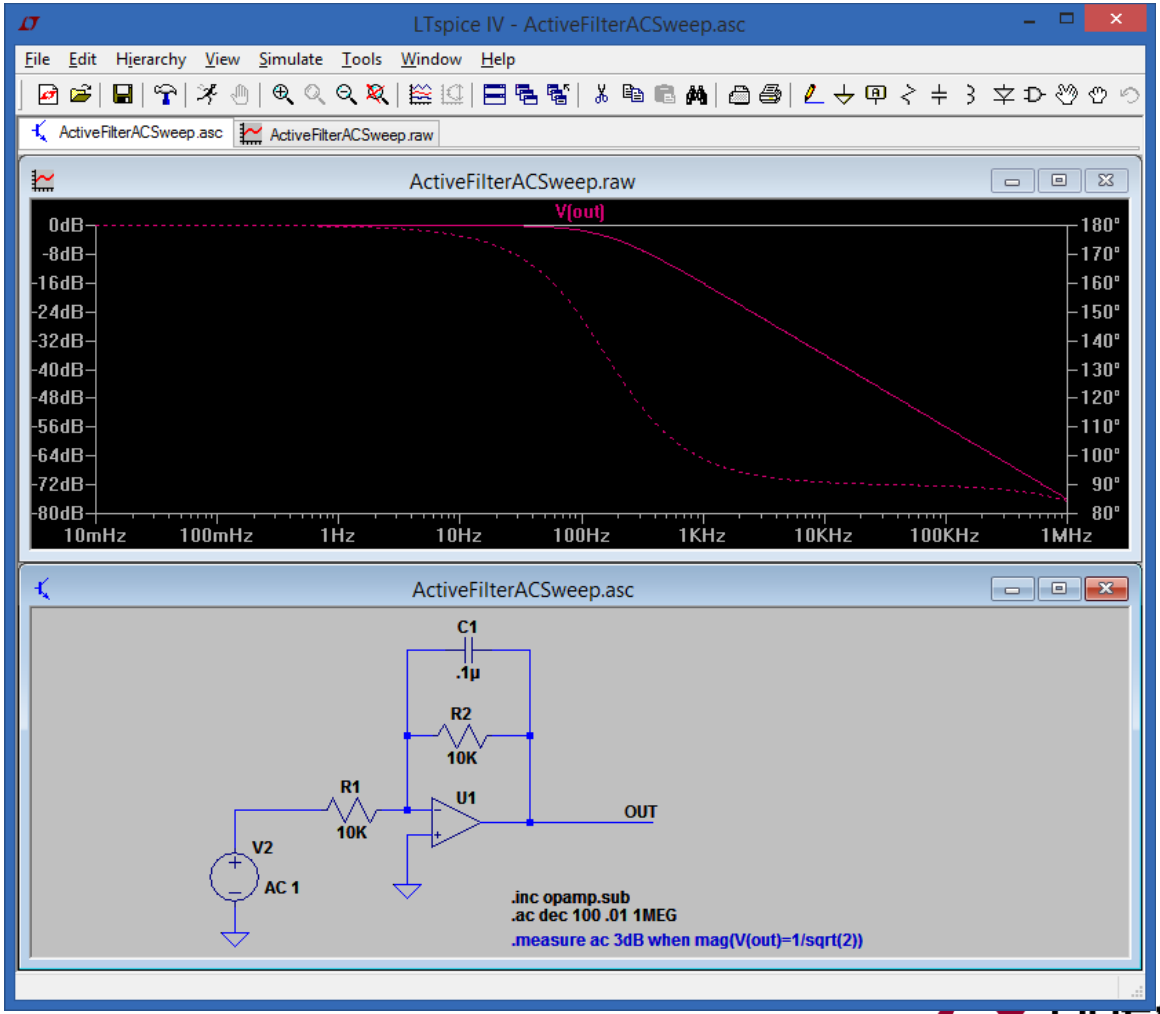
Simulating AC Analysis – RC Result



RCFilterACAnalysis.asc

Simulating AC Analysis – Active Filter

- Single pole active filter using an opamp



ActiveFilterACSweep.asc

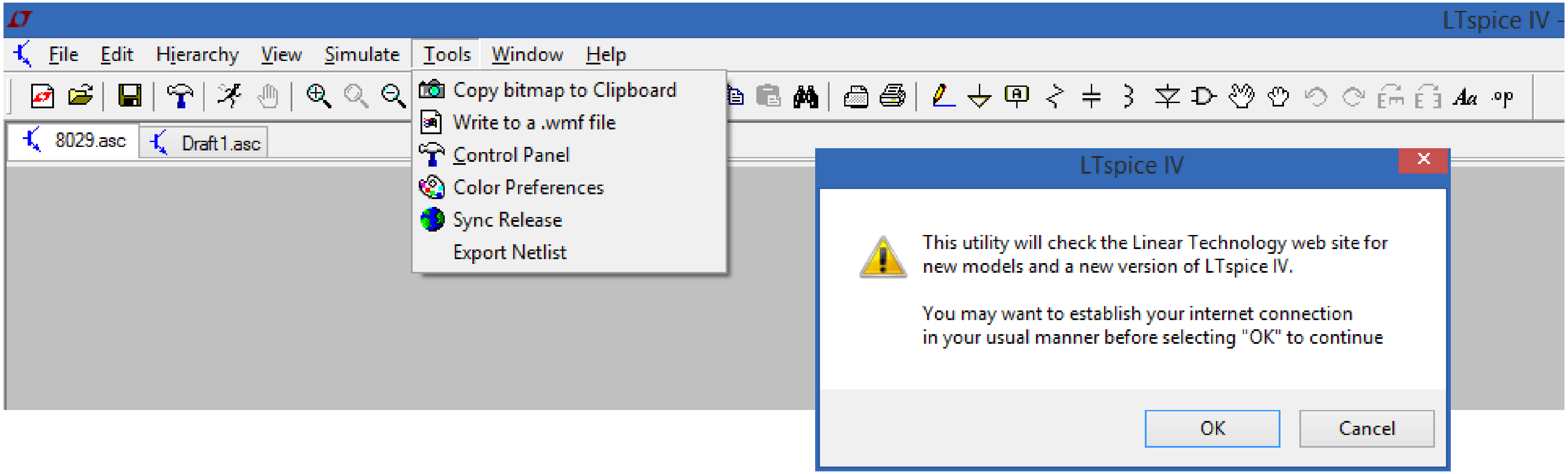
More Information and Support

Reminder to Periodically Sync Release

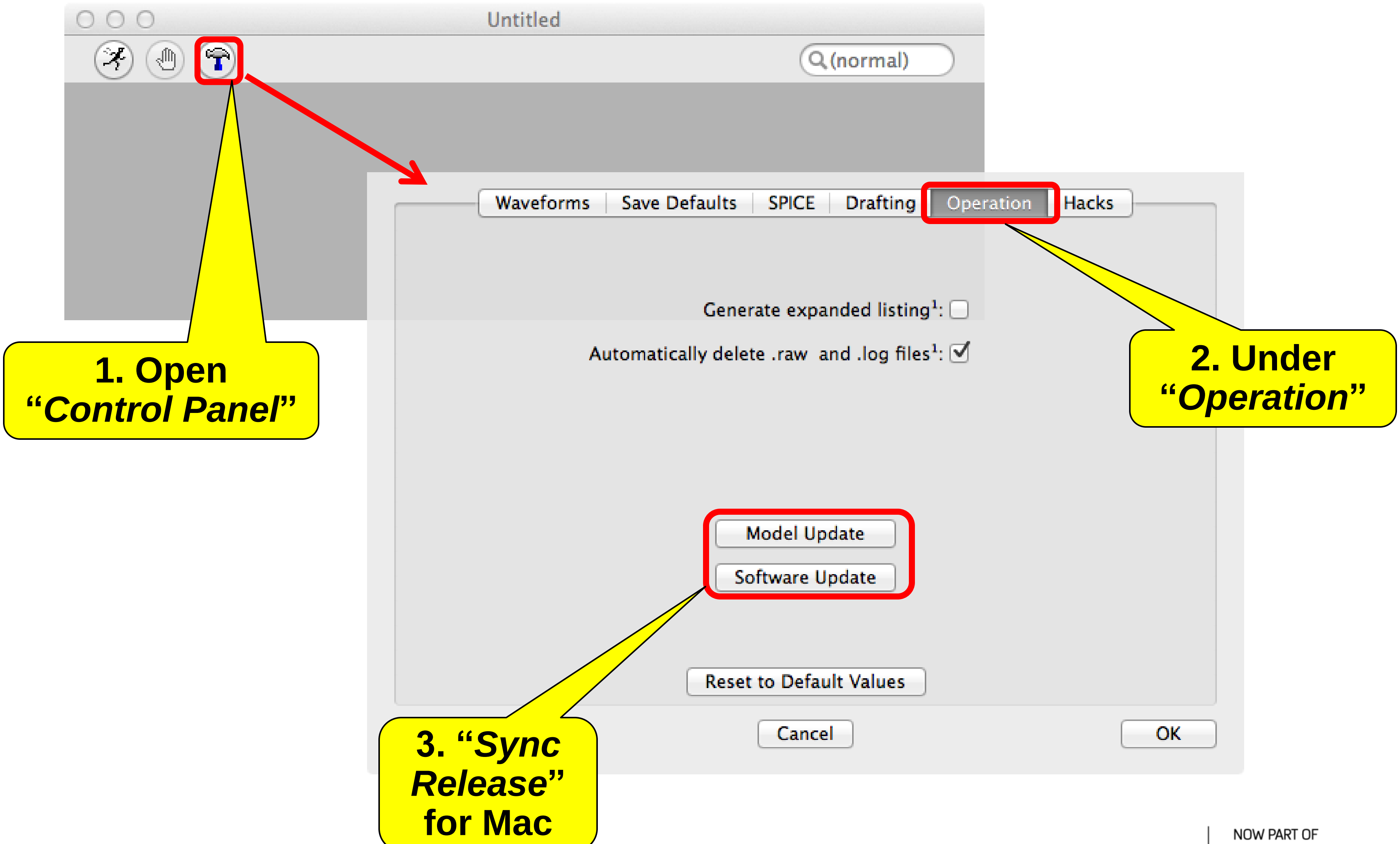
- It is important to sync your release of LTspice once a month to get the latest updates
  - Software updates and bug fixes
  - Models
  - Sample circuits and examples

Reminder to Periodically Sync Release (Windows)

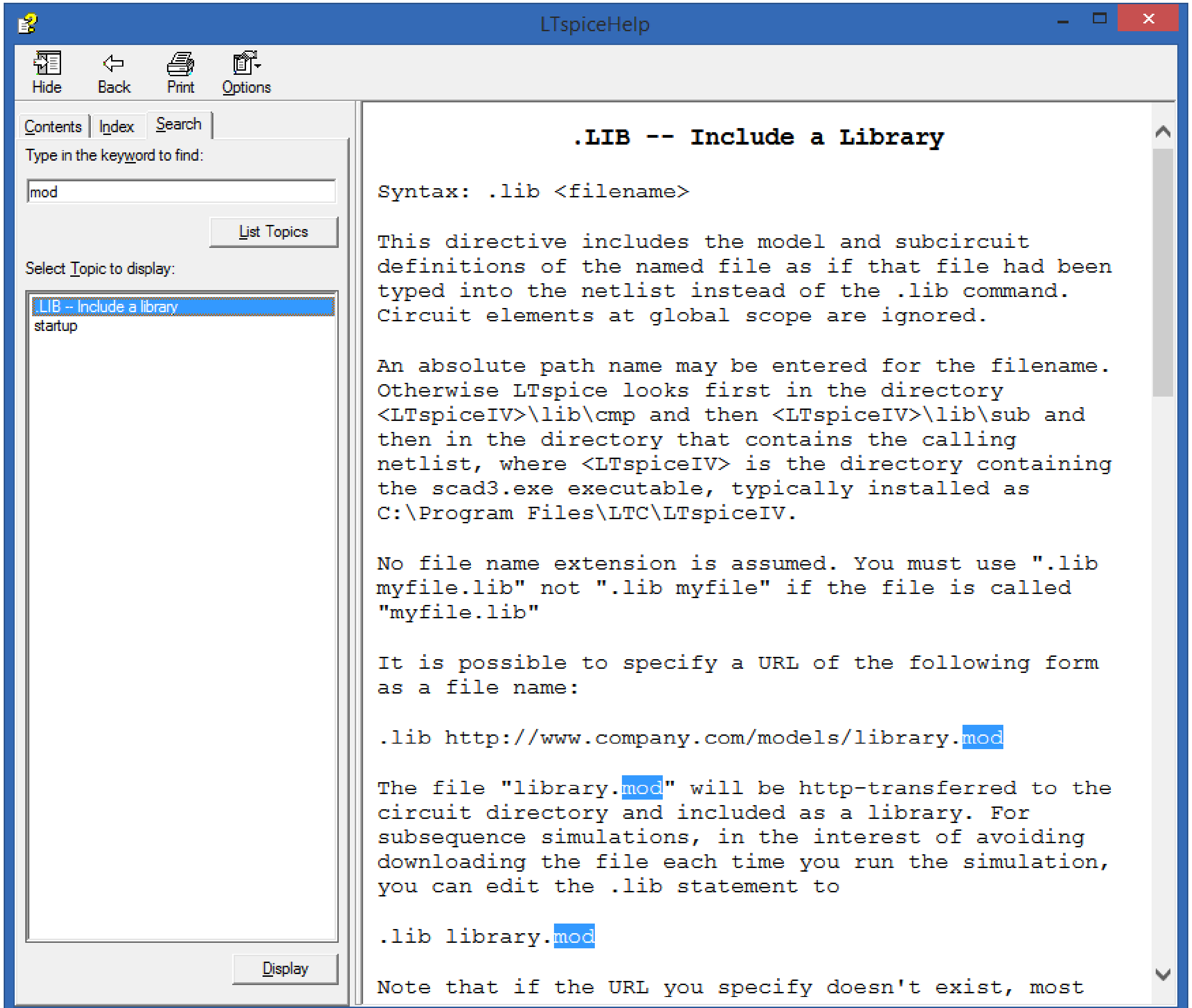
- Vista, Win7, and Win8 users (any UAC-*enabled* OS)
  - You must “Run as administrator” scad.exe or its shortcut even if you are logged in as an administrator



Reminder to Periodically Sync Release (Mac)



Built-in Help System

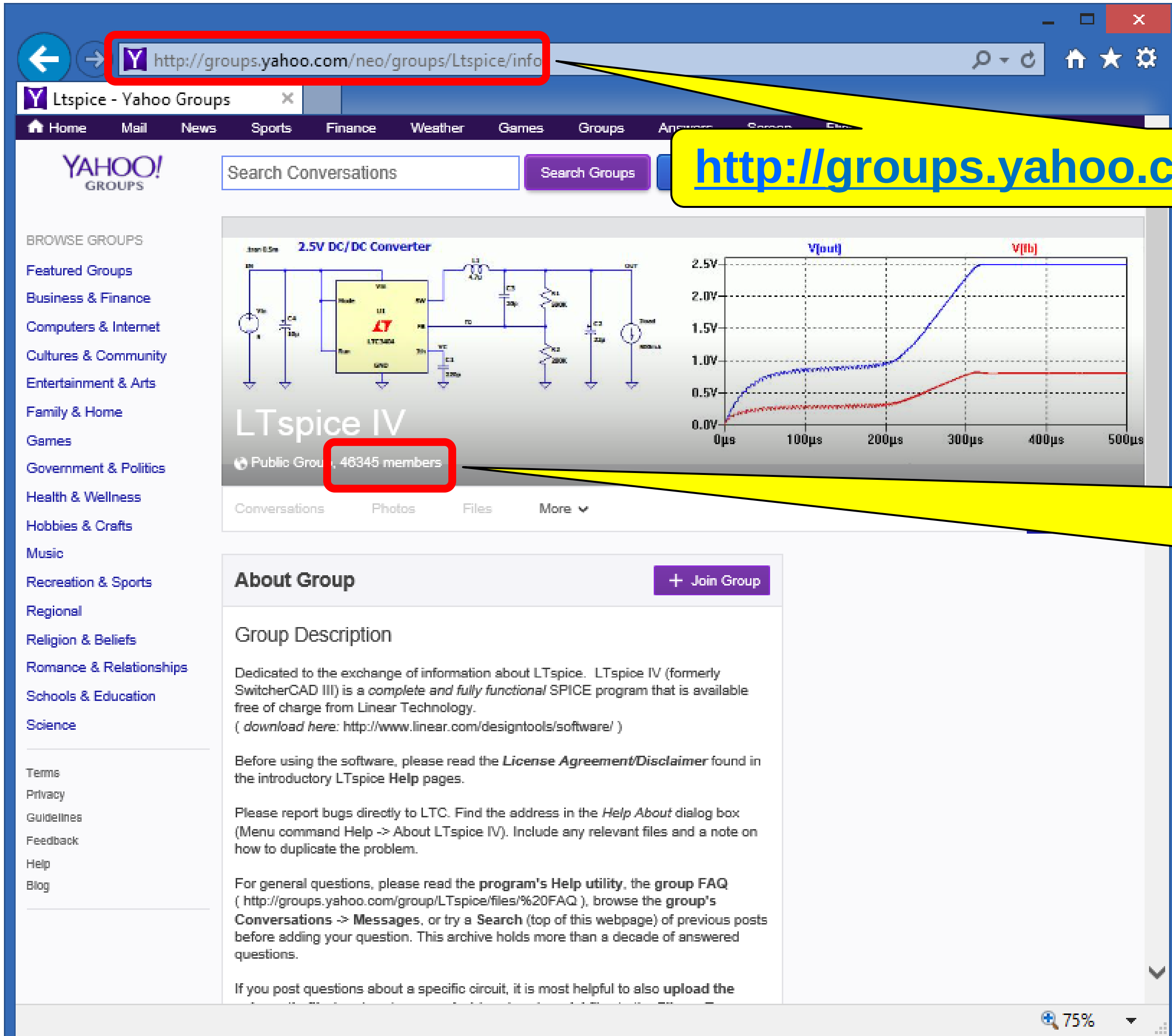


Appendices

Other Resources

- *LTspice forum*: Use simulation circuits posted on LTspice Yahoo! User's Group
  - Go to http://groups.yahoo.com/neo/groups/Ltspice/info
    - Also contains many very helpful discussion threads
- *Educational Files*: Check out LTspice capabilities using the education examples
  - Available on C:\ … \LTspiceIV\examples\Educational
- LTspice videos: Video tutorials by Linear's technical staff
  - http://video.linear.com/all--ltspice
- LTwiki: Undocumented features …
  - http://ltwiki.org/
- Wurth LTspice Book

Other Resources – Yahoo LTspice Forum



http://groups.yahoo.com/neo/groups/Ltspice/info

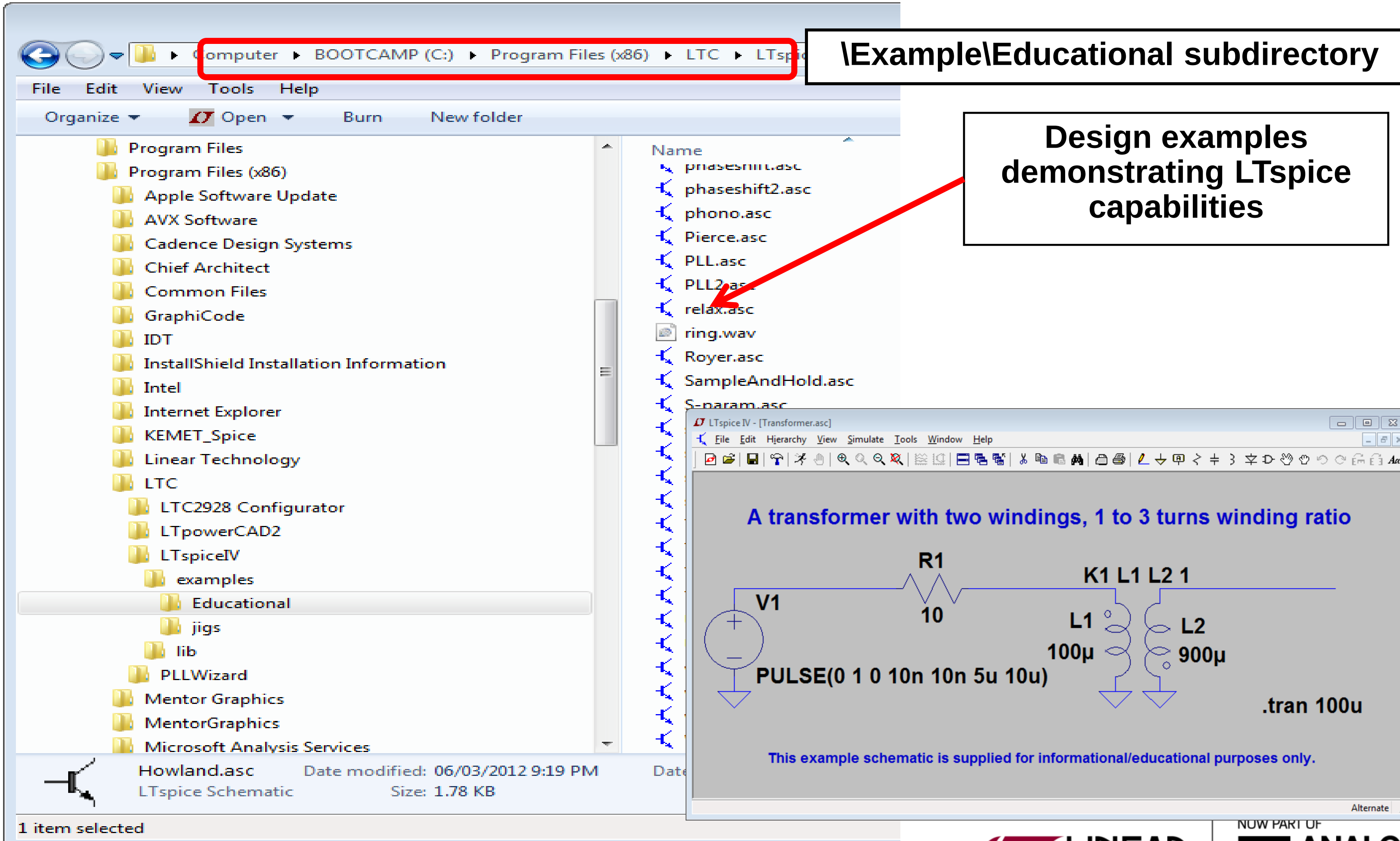
Join the group here.

As of April 2015, there are over 53,329 members!

Other Resources – Educational Files

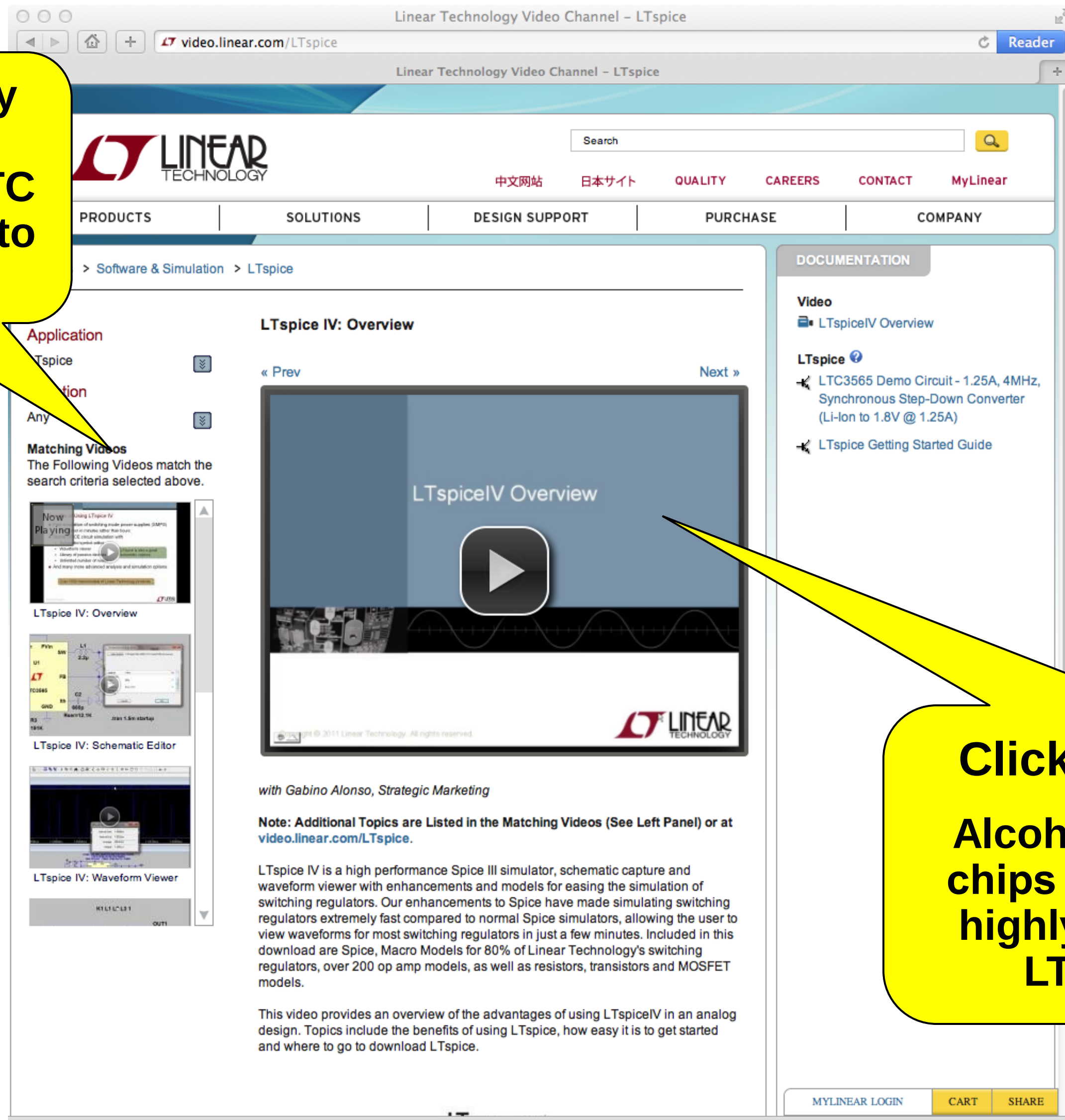
\Example\Educational subdirectory

Design examples demonstrating LTspice capabilities



Other Resources – LTspice Videos

A continuously growing collection of LTC videos for you to discover.

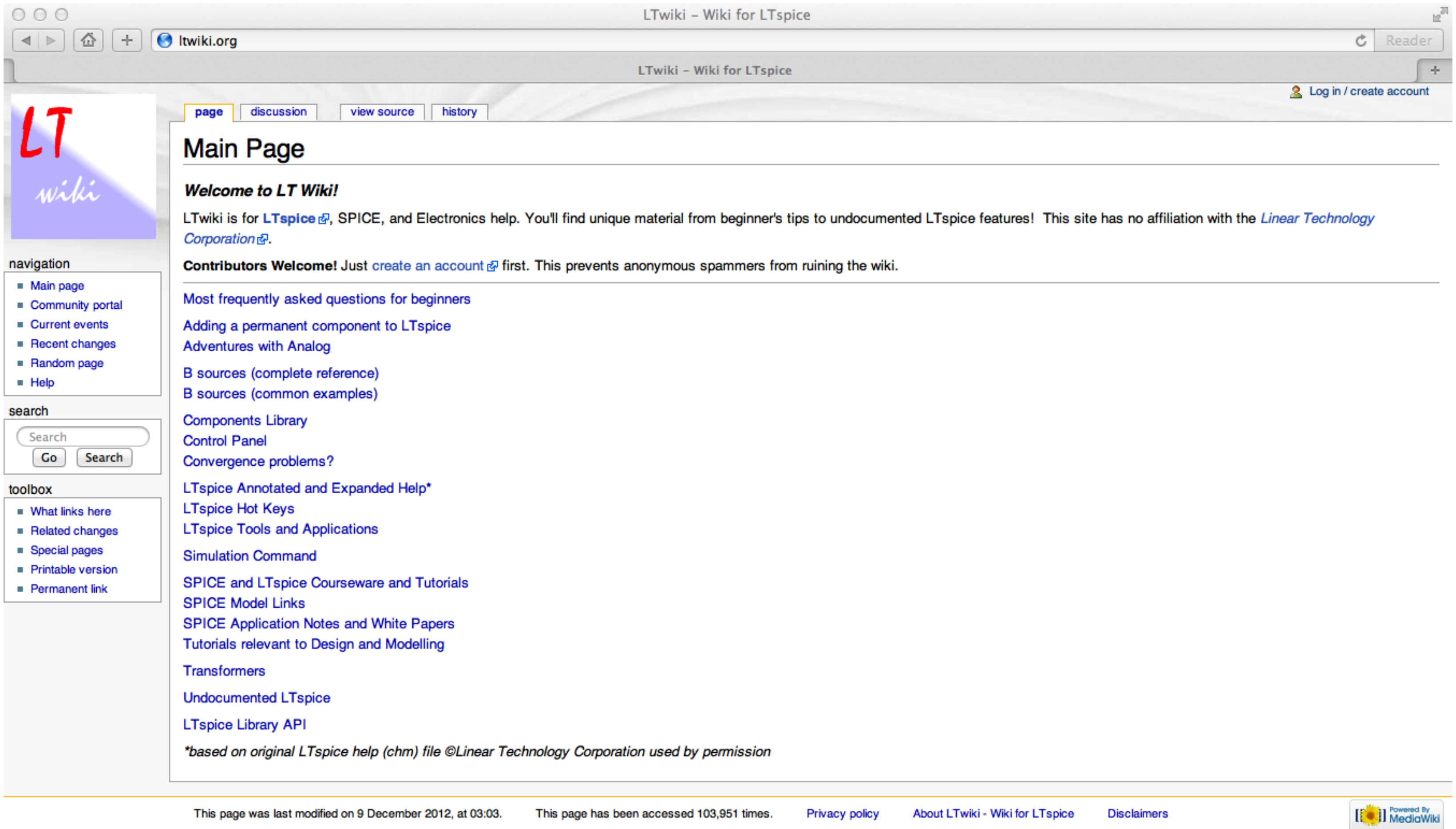


Click, Relax & Enjoy !

Alcoholic beverages and chips not included … But highly recommended by LTC sales teams !

Other Resources - LTwiki

LTwiki



Other Resources – LTspice Book

Appendix - Steps to Calculate Power Supply Efficiency

- Efficiency will only be calculated in the steady state condition
- **Right-Click** the .tran statement on the schematic to bring up the Edit Simulation Command dialog box
- Check the box "Stop simulating if steady state is detected"
- Load must be a current source or resistor labeled Rload
- Run the simulation
- Upon completion select the View dropdown menu, then Efficiency Report, then Show on Schematic
- Efficiency report will be pasted under the schematic

Appendix – Summary of Special Mouse and Keyboard Commands

- Schematic-Based Special Commands
  - **Alt-Left-Click** on a wire
    - This will display the waveform for the current flowing in the wire
  - **Alt-Left-Click** on a component
    - This will display the instantaneous power dissipation in the component
  - **Ctrl-Right-Click** on a component
    - Allows you to edit embedded component attributes
- Waveform-Based Special Commands
  - **Ctrl-Left-Click** on a waveform title
    - Displays the average and RMS values for the waveform
  - **Left-Click** on node and drag to another node
    - Displays differential voltage
  - **Alt-Left-Click** on the label in the waveform viewer (i.e. V(n006))
    - Particular net on the schematic is highlighted

Appendix – Summary of Additional Features

- Pause a simulation
  - "Simulate" pull down menu ---> Pause
  - There is no toolbar button for this function
- Zoom in/out using the schematic editor:
  - Just use the wheel on your mouse
- 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!

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