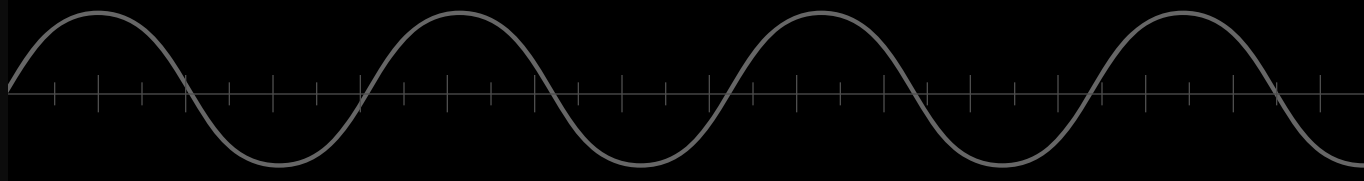


LTspice IV Basic Lab Class

Presented by Thomas Mosteller ADI FAE



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 - μ Power 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>

Module Solutions > μ Module Regulators > μ Module Buck Regulators

VIEW

ACKAGING

ORDER INFO

IMULATE

EMO BOARDS

IRCUITS

IDEOS

OTIFICATIONS

ECH SUPPORT

LTM4620 - Dual 13A or Single 26A DC/DC μ Module Regulator

Features

- Complete Standalone Dual Output Power Supply
- Dual 13A or Single 26A Output
- Wide Input Voltage Range: 4.5V to 16V
- Output Voltage Range: 0.6V to 2.5V
- $\pm 1.5\%$ Maximum Total DC Output Error
- Multiphase Current Sharing with Multiple LTM4620s Up to 100A
- Differential Remote Sense Amplifier
- Current Mode Control/Fast Transient Response
- Adjustable Switching Frequency
- Overcurrent Foldback Protection
- Frequency Synchronization
- Internal Temperature Sensing Diode Output
- Output Overvoltage Protection
- Low Profile (15mm x 15mm x 4.41mm) LGA Package

Typical Application

26A, 1.2V Output DC/DC μ Module[®] Regulator

1.2V Efficiency vs I_{OUT}

ORDER NOW

BUY Request Samples

DOCUMENTATION

Datasheet

Reliability Data

LT Journal

Press Release

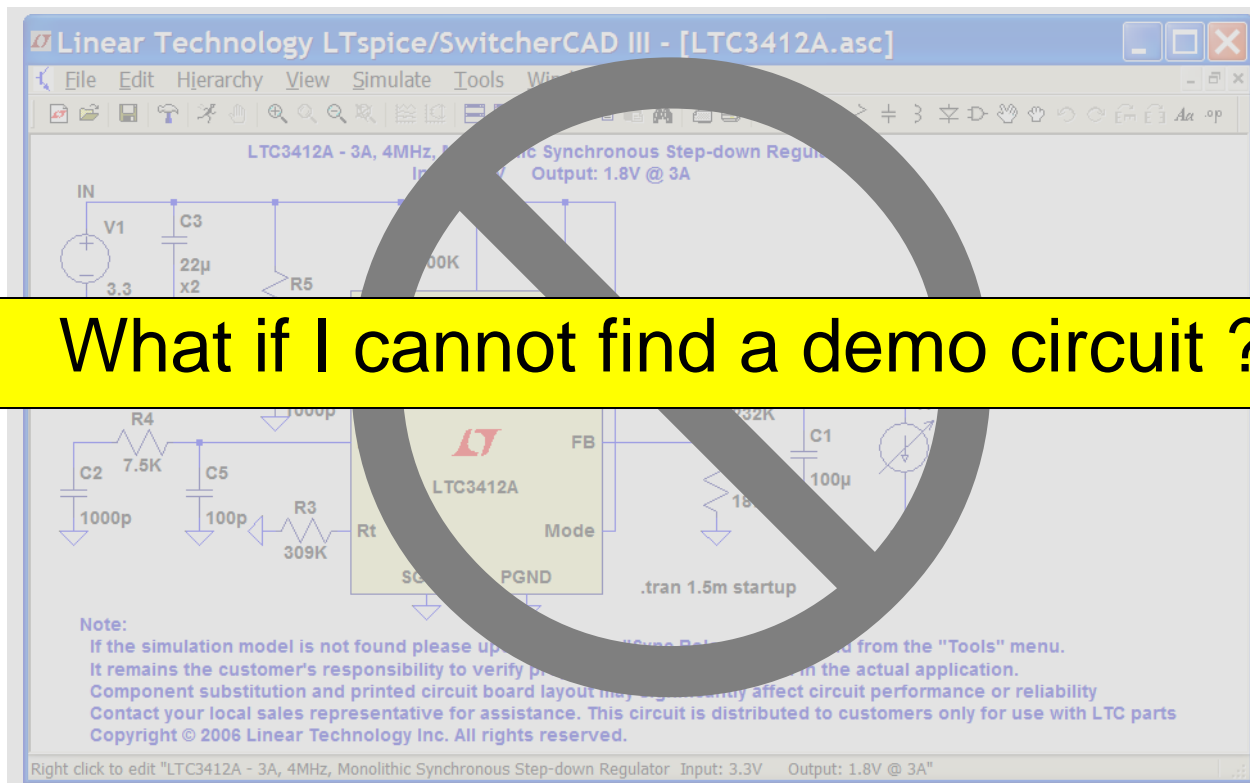
LTspice

LTspice

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

What are Demo Circuits ?

Circuits designed and reviewed by factory apps group



What if I cannot find a demo circuit ?

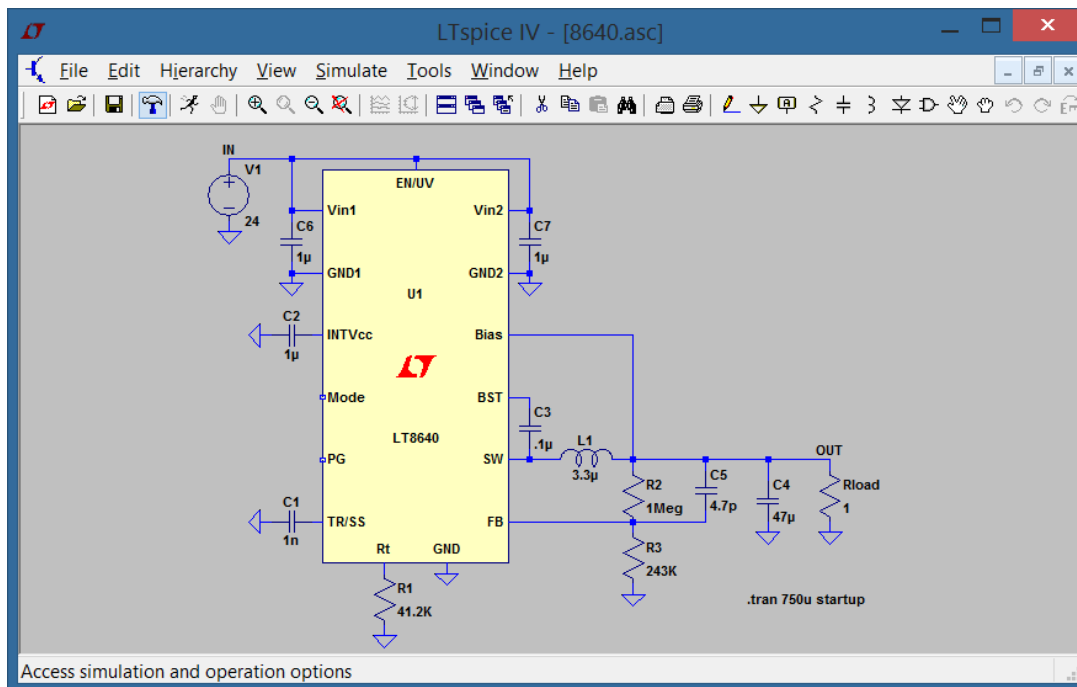
- ♦ It remains the customer's responsibility to verify proper and reliable operation in the actual application
- ♦ Printed circuit board layout may significantly affect circuit performance and reliability

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

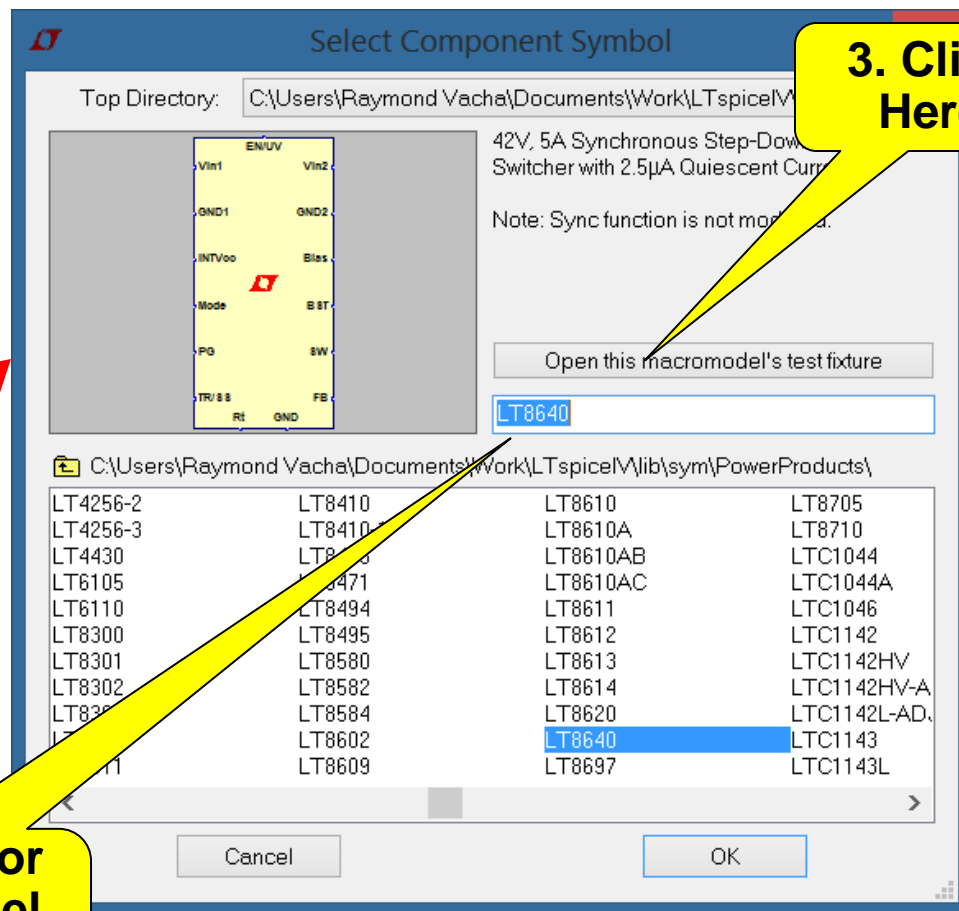
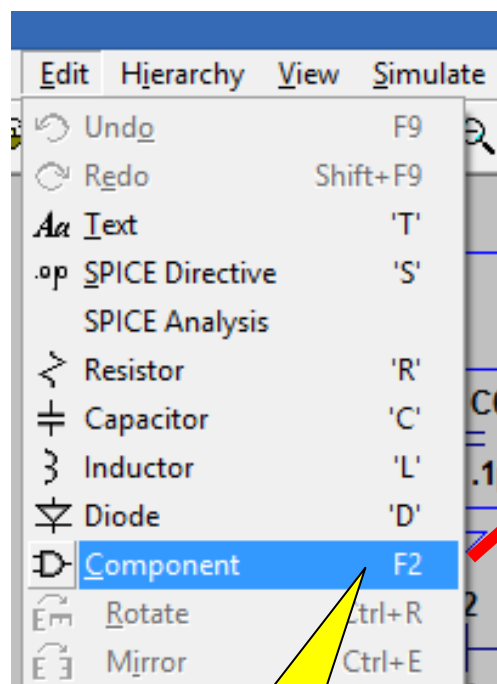
Pre-drafted Test Fixture

- ❖ 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

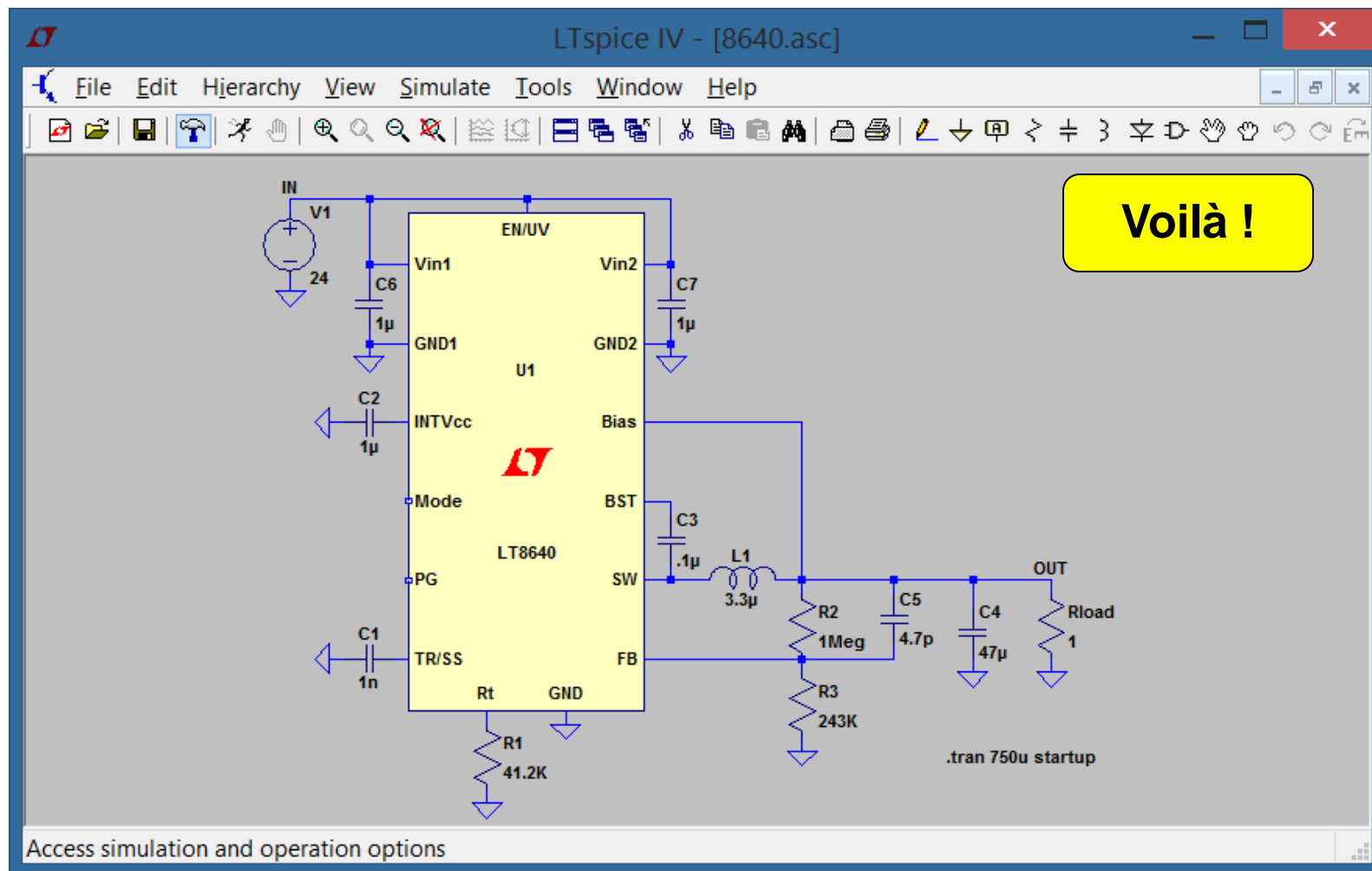


- ♦ It remains the customer's responsibility to verify proper and reliable operation in the actual application
- ♦ Printed circuit board layout may significantly affect circuit performance and reliability

Opening a Test Fixture



Opening a Test Fixture

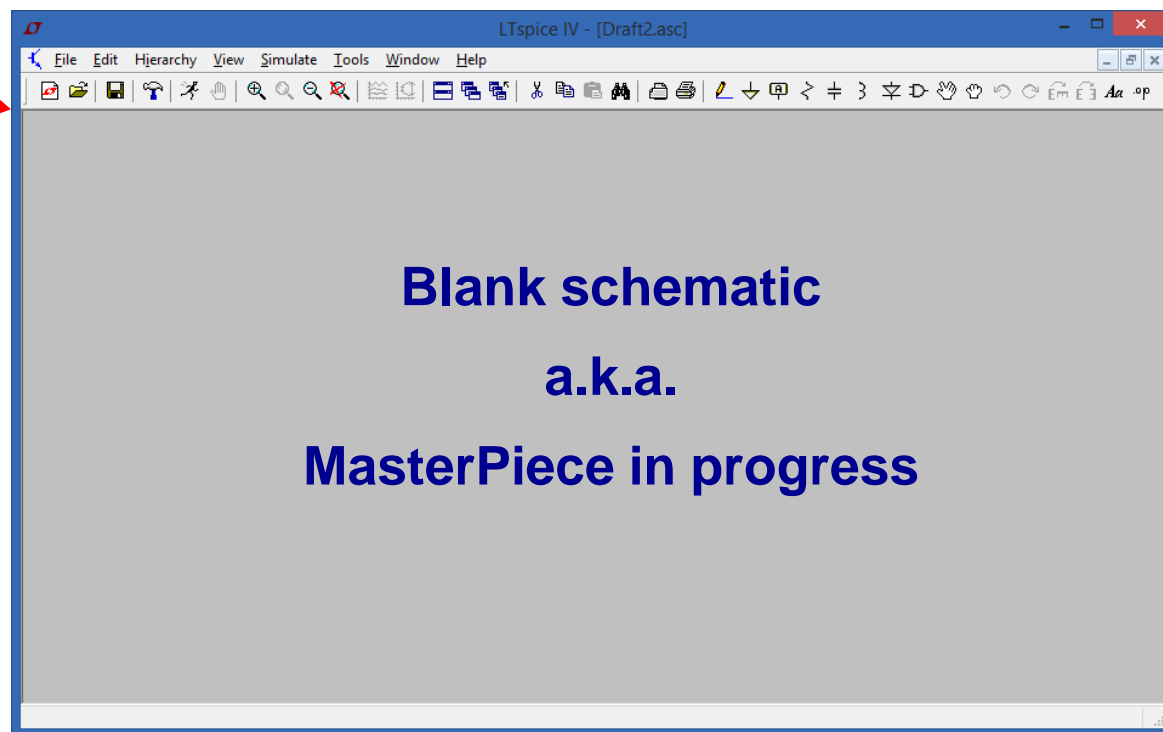
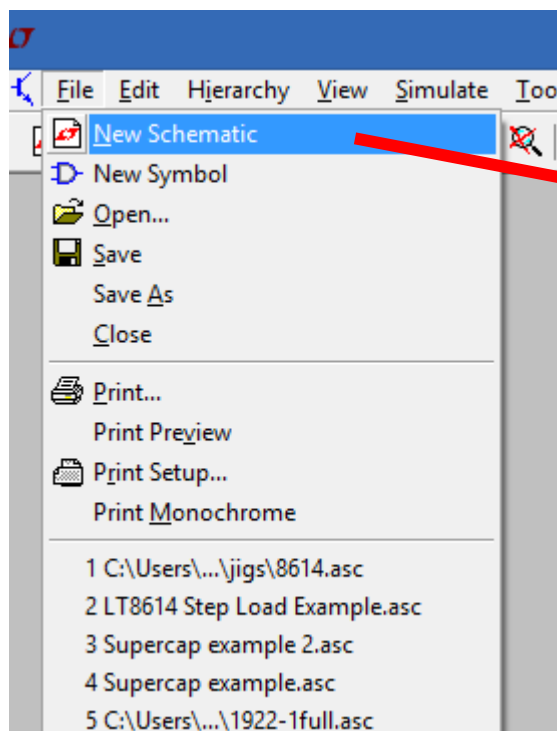


How Do I Get Started Using LTspice?

- ❖ 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

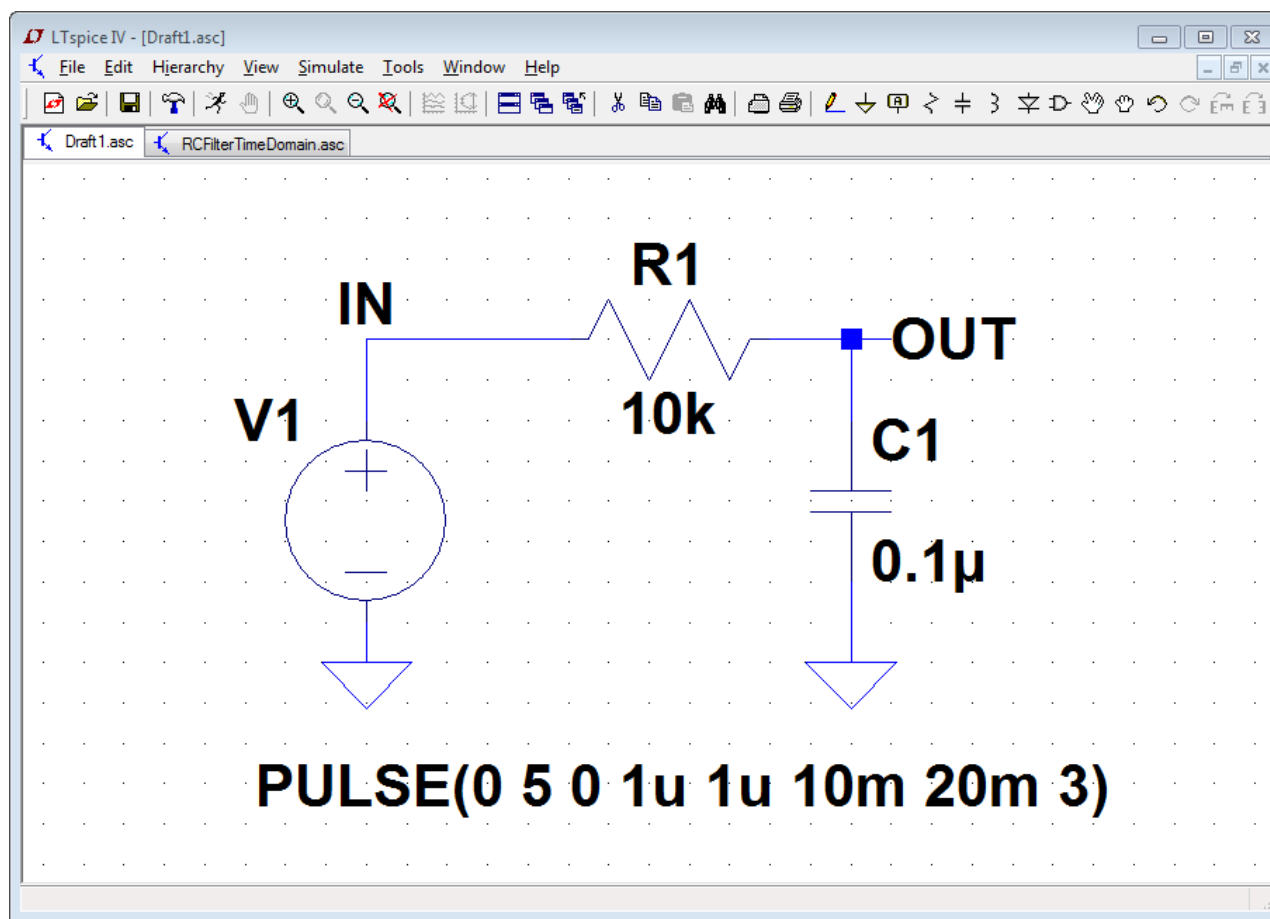
- ❖ To open up a blank schematic screen select “*File*” Menu and “*New Schematic*”



Using the Schematic Editor in LTspice

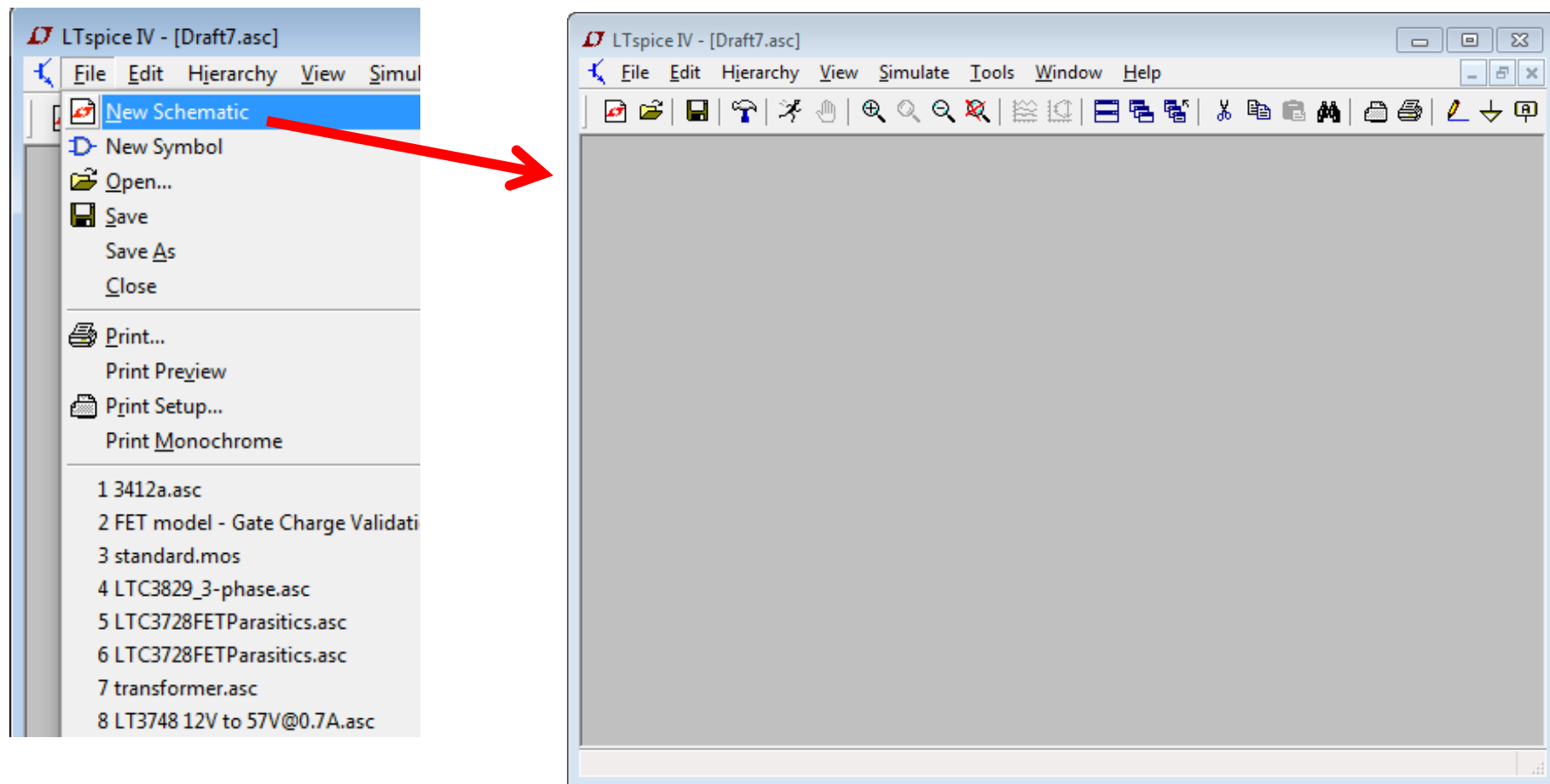
How to Wire up a Simple RC Circuit

❖ The completed exercise:



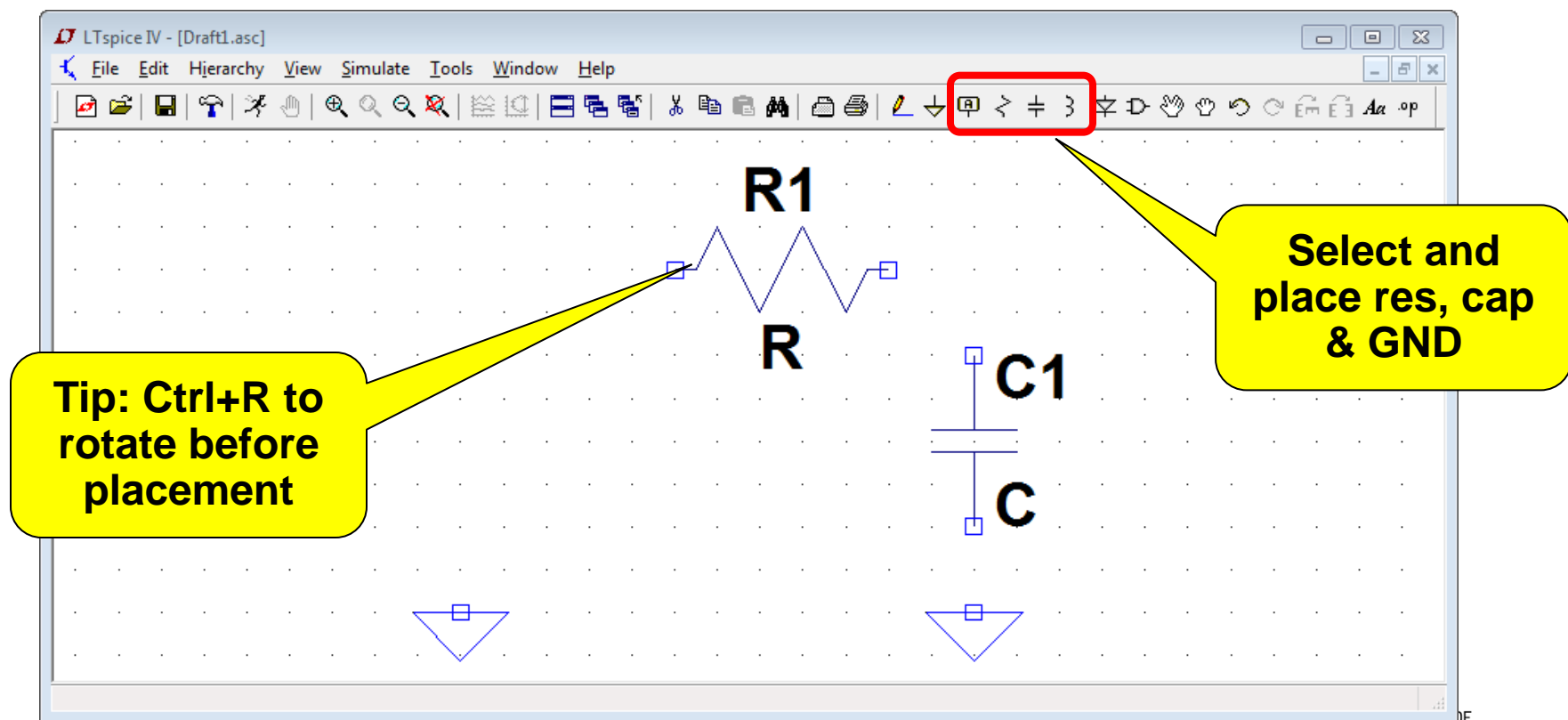
How to Wire up a Simple RC Circuit (cont.)

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



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

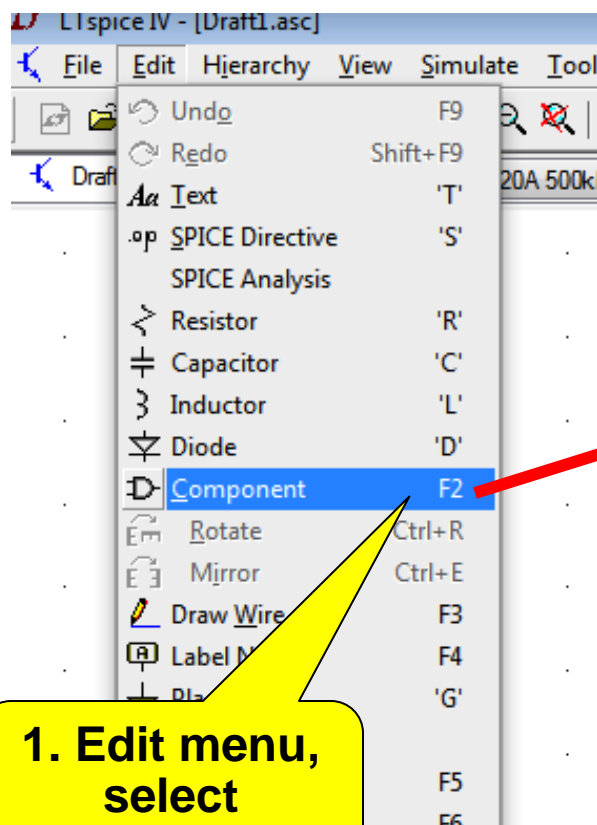
- ❖ Step 2: Add the passives and grounds
 - ❖ Using the toolbar, select Resistor, Capacitor and Ground. Place these symbols on the schematic as shown below. Use Ctrl+R to rotate before placement.



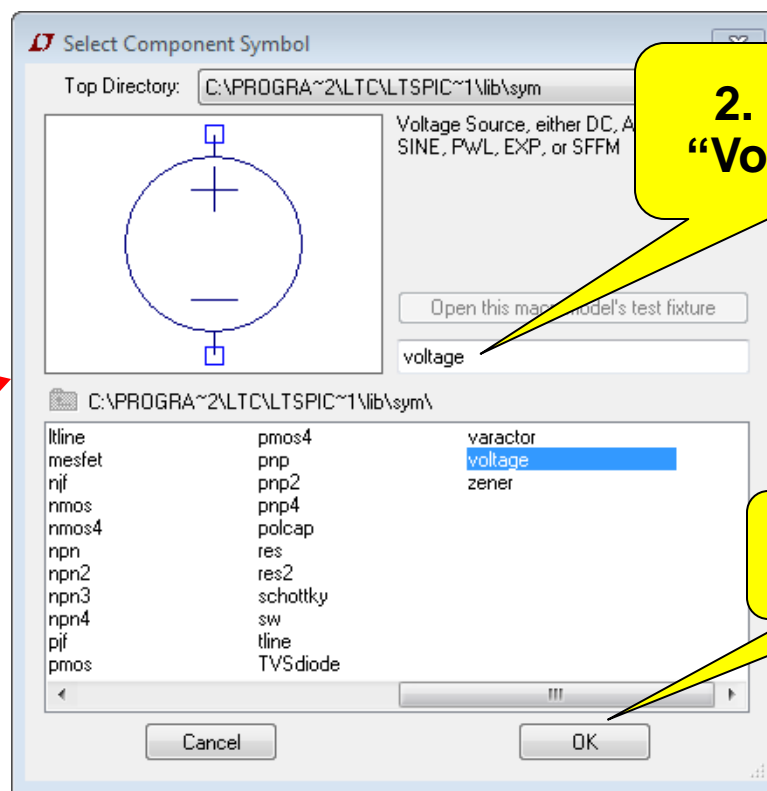
How to Wire up a Simple RC Circuit (cont.)

❖ 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”



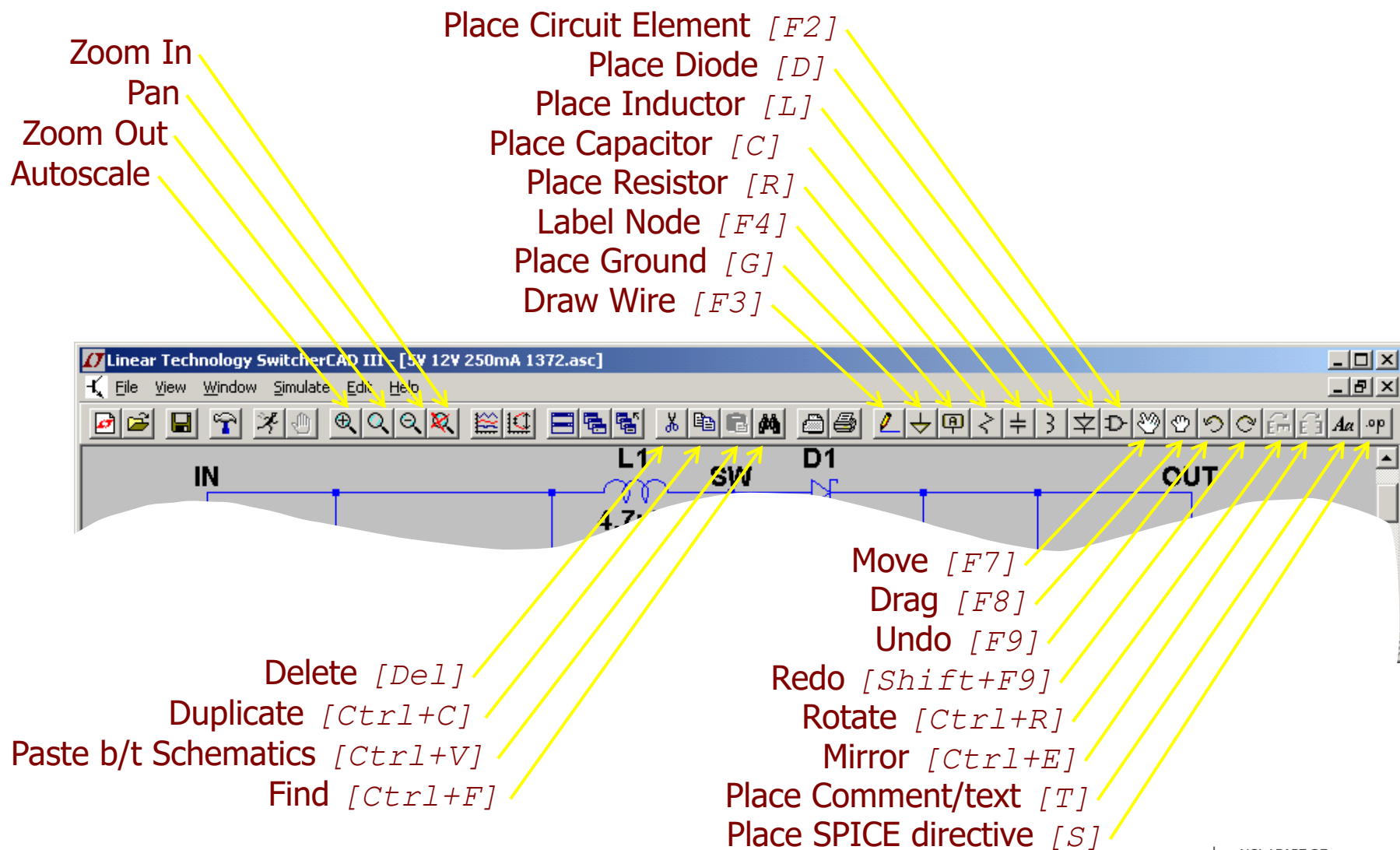
1. Edit menu, select “Component”



2. Type “Voltage”

3. Click “OK”

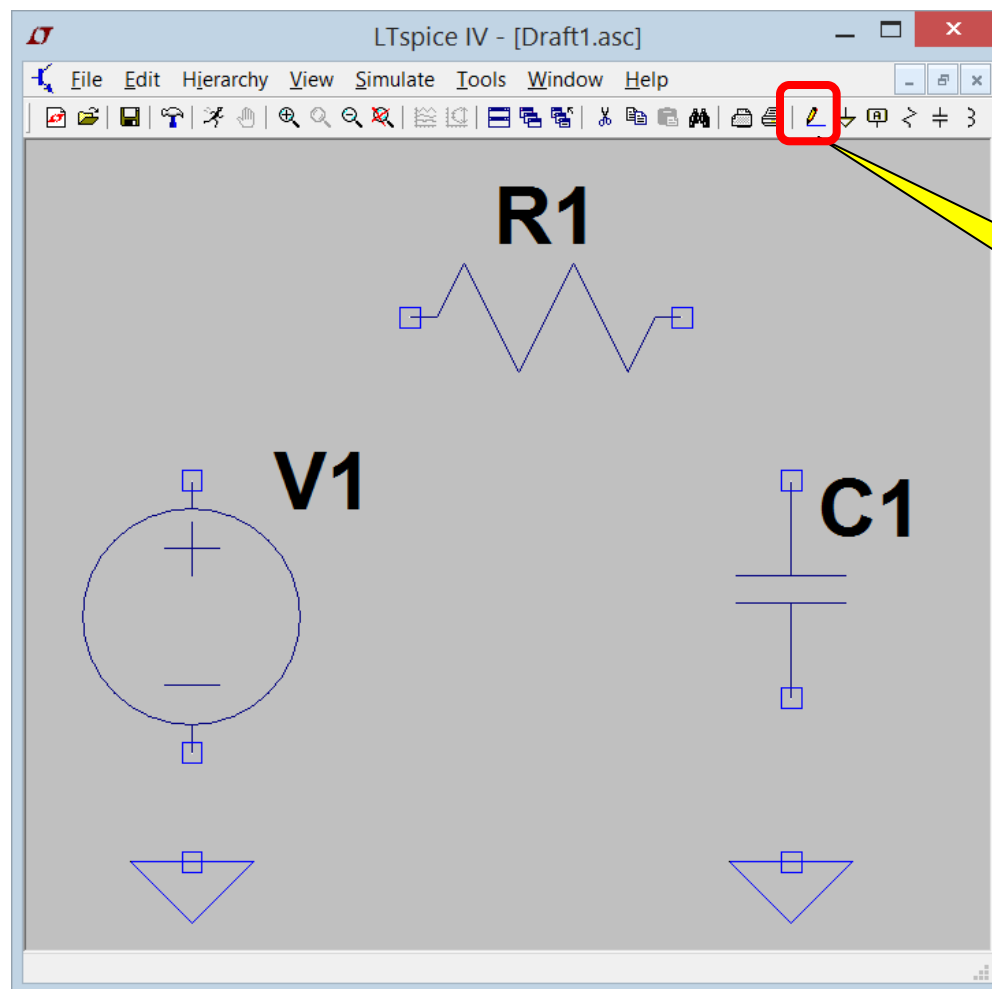
Toolbar and Keyboard Shortcuts



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

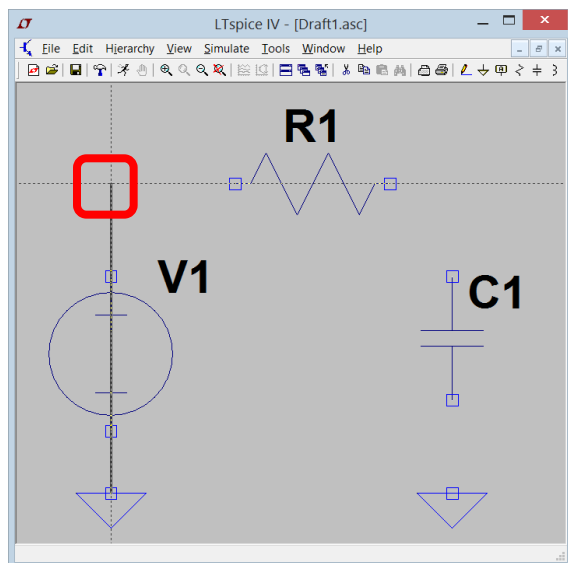
❖ Step 4: Wire up the circuit

- ❖ Using the toolbar, select Wire



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

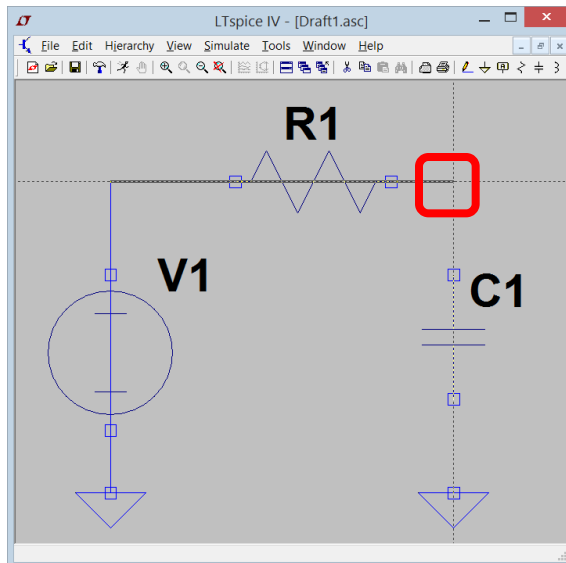
❖ Step 4: Wire up the circuit (cont.)



Left-Click ground

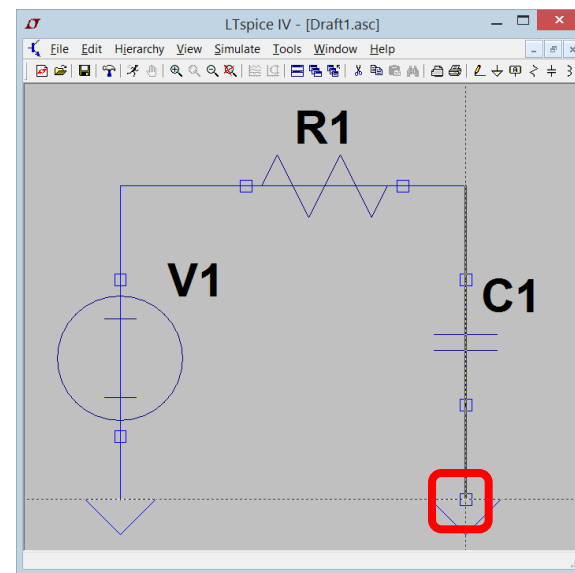
“Pull” wire up through the source

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

The image shows the LTspice IV interface with a circuit diagram and a 'Net Name' dialog box. The circuit diagram includes a voltage source V1, a resistor R1, a capacitor C1, and nodes labeled IN, OUT, and V. A yellow callout box labeled '1. Select "Label Net"' points to the 'Label Net' icon in the toolbar. A red arrow points from this icon to the 'Net Name' dialog box. A yellow callout box labeled '2. Enter net name' points to the text input field in the dialog box, which contains 'ABC IN'. A red arrow points from the dialog box to the circuit diagram. A yellow callout box labeled '3. Place on wire' points to a wire in the circuit diagram.

1. Select "Label Net"

2. Enter net name

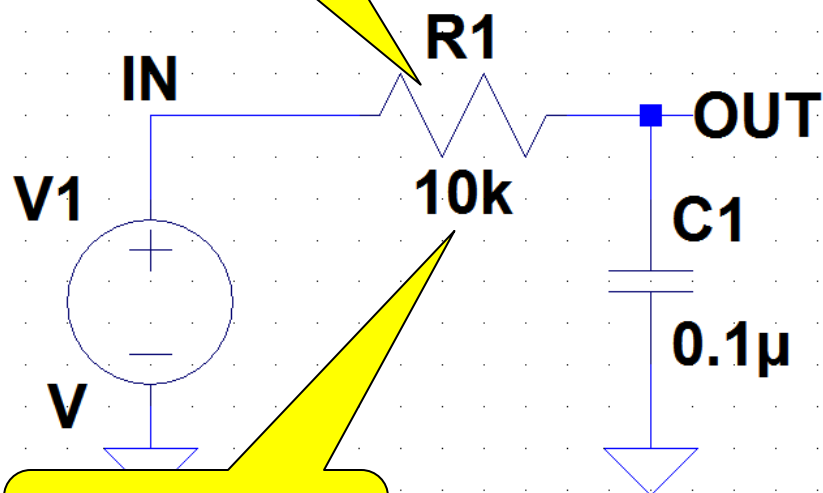
3. Place on wire

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

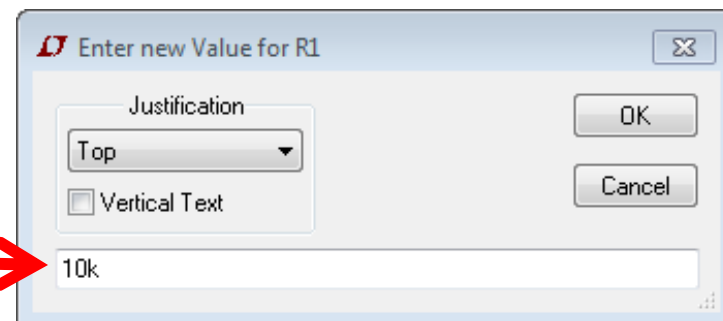
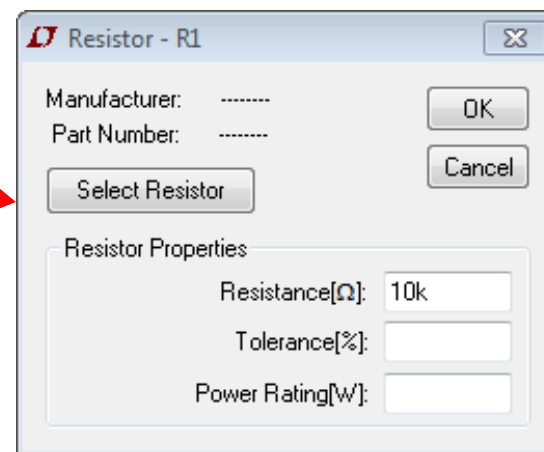
❖ Step 6a: Component values

- ❖ **Right-Click** on each component symbol to change its value as shown below

Right-click on
symbol



Or Right-click
on value



Using Labels to Specify Units for Component Attributes

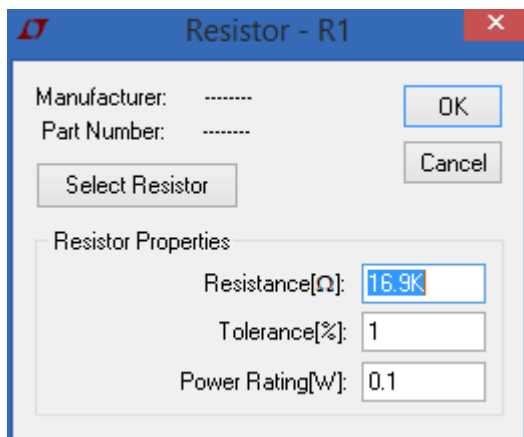
- ❖ $K = k = \text{kilo} = 10^3$
- ❖ $MEG = \text{meg} = 10^6$
- ❖ $G = g = \text{giga} = 10^9$
- ❖ $T = t = \text{tera} = 10^{12}$
- ❖ $M = m = \text{milli} = 10^{-3}$
- ❖ $U = u = \text{micro} = 10^{-6}$
- ❖ $N = n = \text{nano} = 10^{-9}$
- ❖ $P = p = \text{pico} = 10^{-12}$
- ❖ $F = f = \text{femto}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

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



Resistor - R1

Manufacturer: OK

Part Number: Cancel

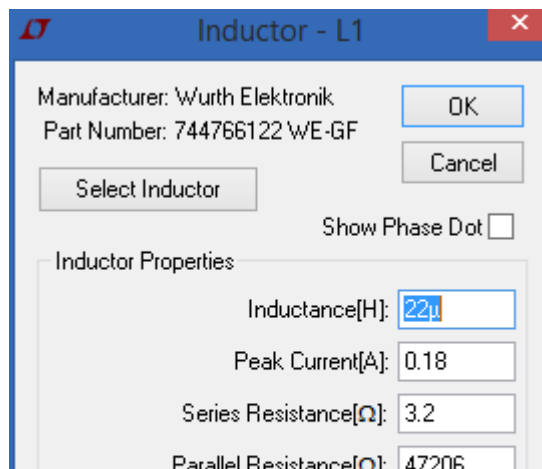
Select Resistor

Resistor Properties

Resistance[Ω]: 16.9K

Tolerance[%]: 1

Power Rating[W]: 0.1



Inductor - L1

Manufacturer: Würth Elektronik OK

Part Number: 744766122 WE-GF Cancel

Select Inductor

Show Phase Dot ☐

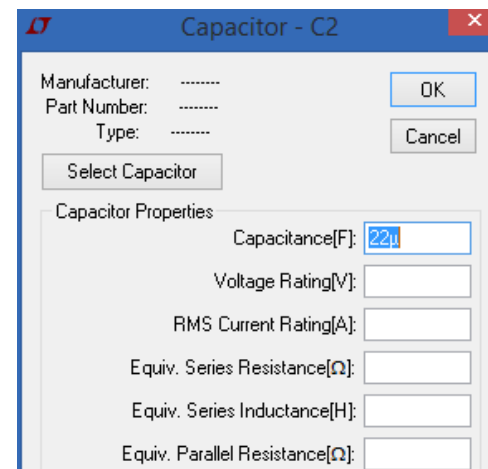
Inductor Properties

Inductance[H]: 22μ

Peak Current[A]: 0.18

Series Resistance[Ω]: 3.2

Parallel Resistance[Ω]: 47206



Capacitor - C2

Manufacturer: OK

Part Number: Cancel

Type:

Select Capacitor

Capacitor Properties

Capacitance[F]: 22μ

Voltage Rating[V]:

RMS Current Rating[A]:

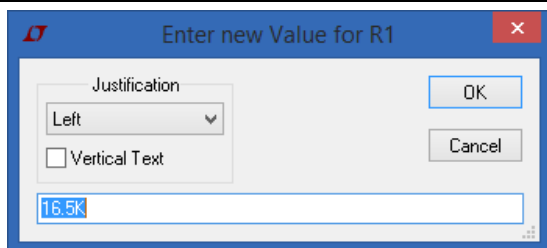
Equiv. Series Resistance[Ω]:

Equiv. Series Inductance[H]:

Equiv. Parallel Resistance[Ω]:

- ❖ 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



Enter new Value for R1

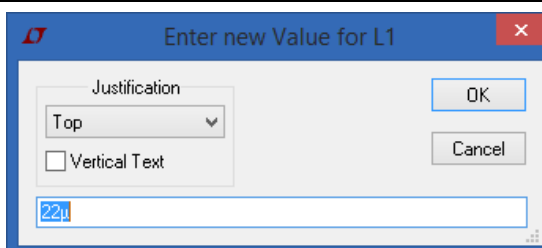
Justification: Left

☐ Vertical Text

OK

Cancel

16.5K



Enter new Value for L1

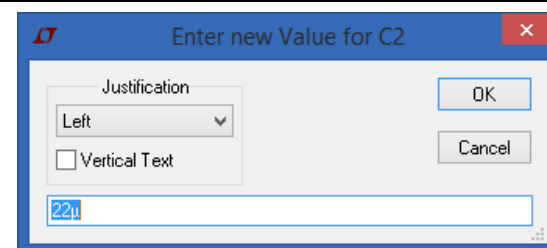
Justification: Top

☐ Vertical Text

OK

Cancel

22μ



Enter new Value for C2

Justification: Left

☐ Vertical Text

OK

Cancel

22μ

Component Database

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

Resistor - R1

Manufacturer: OK
Part Number: Cancel

Select Resistor

Resistor Properties

Resistance[Ω]: 16.9K
Tolerance[%]: 1
Power Rating[W]: 0.1

Select Standard Resistor

Quit and Edit Database OK
List All Resistors in Database Cancel

R[Ω]	Mfg.	Part No.	Power[W]	Tolerance[%]
16.90K			0.100	1.00
16.50K			0.100	1.00
17.40K			0.100	1.00
16.20K			0.100	1.00
17.80K			0.100	1.00
15.80K			0.100	1.00
18.20K			0.100	1.00
15.40K			0.100	1.00
18.70K			0.100	1.00

Inductor - L1

Manufacturer: Wurth Elektronik OK
Part Number: 744766122 WE-GF Cancel

Select Inductor

Inductor Properties

Inductance[H]: 22μ
Peak Current[A]: 0.18

Select Stock Inductor

Quit and Edit Database OK
List All Inductors in Database Cancel

L[μH]	Mfg.	Part No.	Ipk[A]	Rser[Ω]
22.0	Wurth Elektronik	744766122 WE-GF	0.180	3.200
22.0	Murata	LQH2MCN220K0	0.185	2.100
22.0	Sumida	CDRH2D18BLDN	0.300	0.320
22.0	Murata	LQH3NPN220NG	0.340	1.100
22.0	Murata	LQH3NPN220MG	0.340	1.100
22.0	Taiyo Yuden	NR3010T-220M	0.350	1.030
22.0	Taiyo Yuden	NR4010T-220M	0.360	0.870
22.0	Taiyo Yuden	NR3012T-220M	0.375	0.630

Capacitor - C2

Manufacturer: OK
Part Number: Cancel
Type: Cancel

Select Capacitor

Capacitor Properties

Capacitance[F]: 22μ
Voltage Rating[V]:
RMS Current Rating[A]:
equiv. Series Resistance[Ω]:

Select Stock Capacitor

Quit and Edit Database OK
List All Capacitors in Database Cancel

C[μF]	Mfg.	type	Part No.	Voltage[V]	Rser[Ω]
22.0	Sanyo POSCA	Al electrolytic	2R5TPU22M	2.5	0.250
22.0	AVX	Tantalum	TAJA226M004	4.0	3.500
22.0	AVX	Tantalum	TAJB226M006	6.3	2.500
22.0	TDK	X5R	C3225X5R0J22	6.3	0.001
22.0	Nichicon	Al electrolytic	UPR0J220MAH	6.3	2.400
22.0	Sanyo	Al electrolytic	6CE22GA	6.3	2.300

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.

The diagram shows an RC circuit schematic with a voltage source V1, a resistor R1 (10k), and a capacitor C1 (0.1μ). The input is labeled IN and the output is labeled OUT. A red arrow points from the voltage source V1 to the 'Voltage Source - V1' dialog box. A yellow callout bubble points to the 'Advanced' button in this dialog box, with the text 'Click "Advanced"'. Another yellow callout bubble points to the voltage source V1 with the text 'Right-click source'. The 'Voltage Source - V1' dialog box shows the 'Advanced' tab selected, with the 'PULSE' function chosen. The 'Independent Voltage Source - V1' dialog box shows the 'PULSE' function parameters: V1=0, V2=5, Tdelay=0, Trise=1u, Tfall=1u, Ton=10m, Tperiod=20m, Ncycles=3. The 'DC Value' section shows 'DC value: 0'. The 'Small signal AC analysis' section shows 'AC Amplitude: 1' and 'AC Phase: 0'. The 'Parasitic Properties' section shows 'Series Resistance: 0' and 'Parallel Capacitance: 0'. The 'Make this information visible on schematic' checkbox is checked in all three sections.

Voltage Source - V1

DC value[V]:

Series Resistance[Ω]:

OK Cancel Advanced

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

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: ☒

Initial[V]:

Von[V]:

Tdelay[s]:

Trise[s]:

Tfall[s]:

Ton[s]:

Tperiod[s]:

Ncycles:

Additional PWL Points

Make this information visible on schematic: ☒

Cancel OK

PULSE(0 5 0 1u 10m 20m 3)

Click "Advanced"

Right-click source

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.

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:

Vinitial[V]:

Von[V]:

Tdelay[s]:

Trise[s]:

Tfall[s]:

Ton[s]:

Tperiod[s]:

Ncycles:

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: ☒

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



❖ Hotlink Nomenclature:



Class exercise



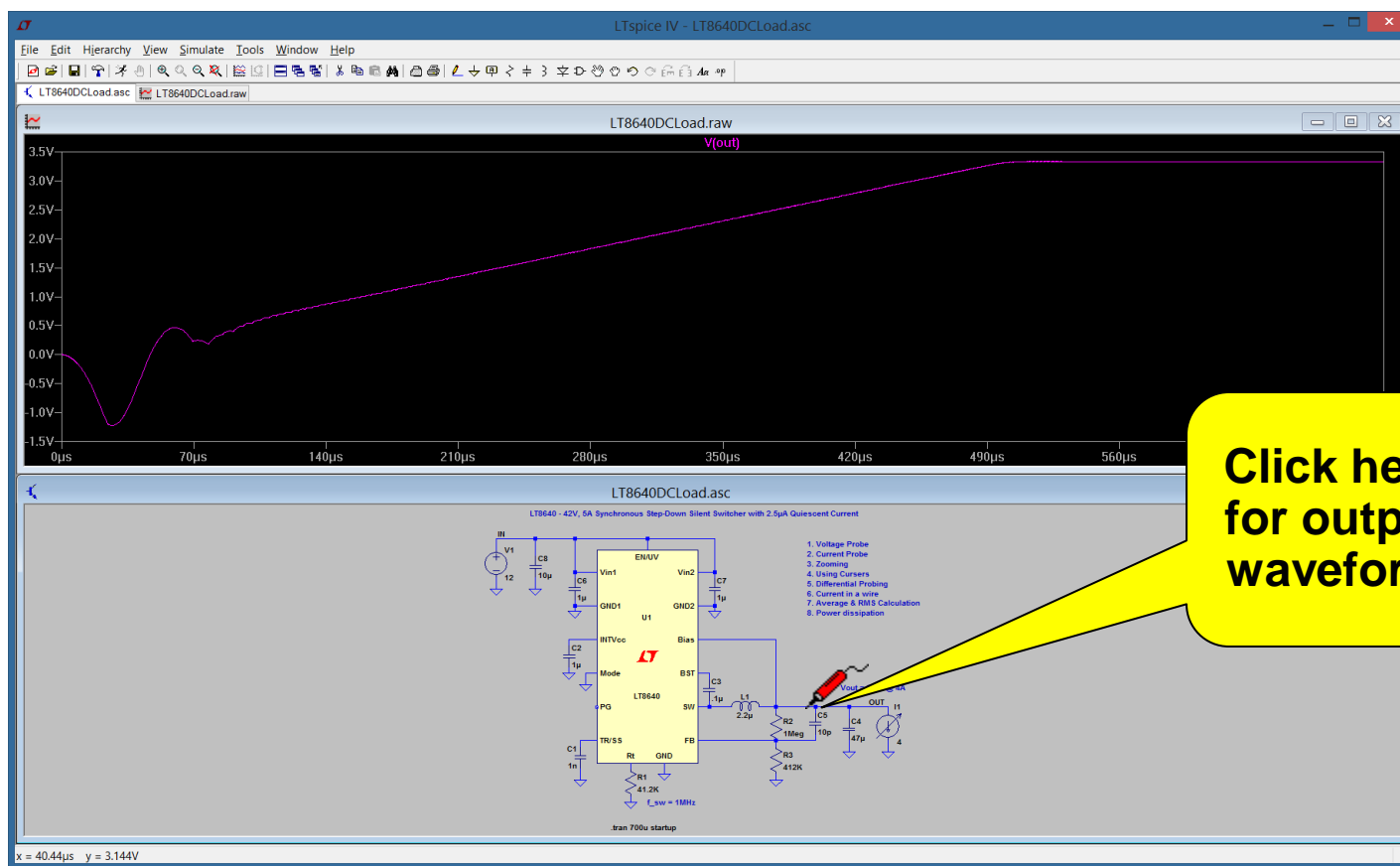
Solution to exercise



Circuits to explore at your leisure

Viewing Voltage Waveforms

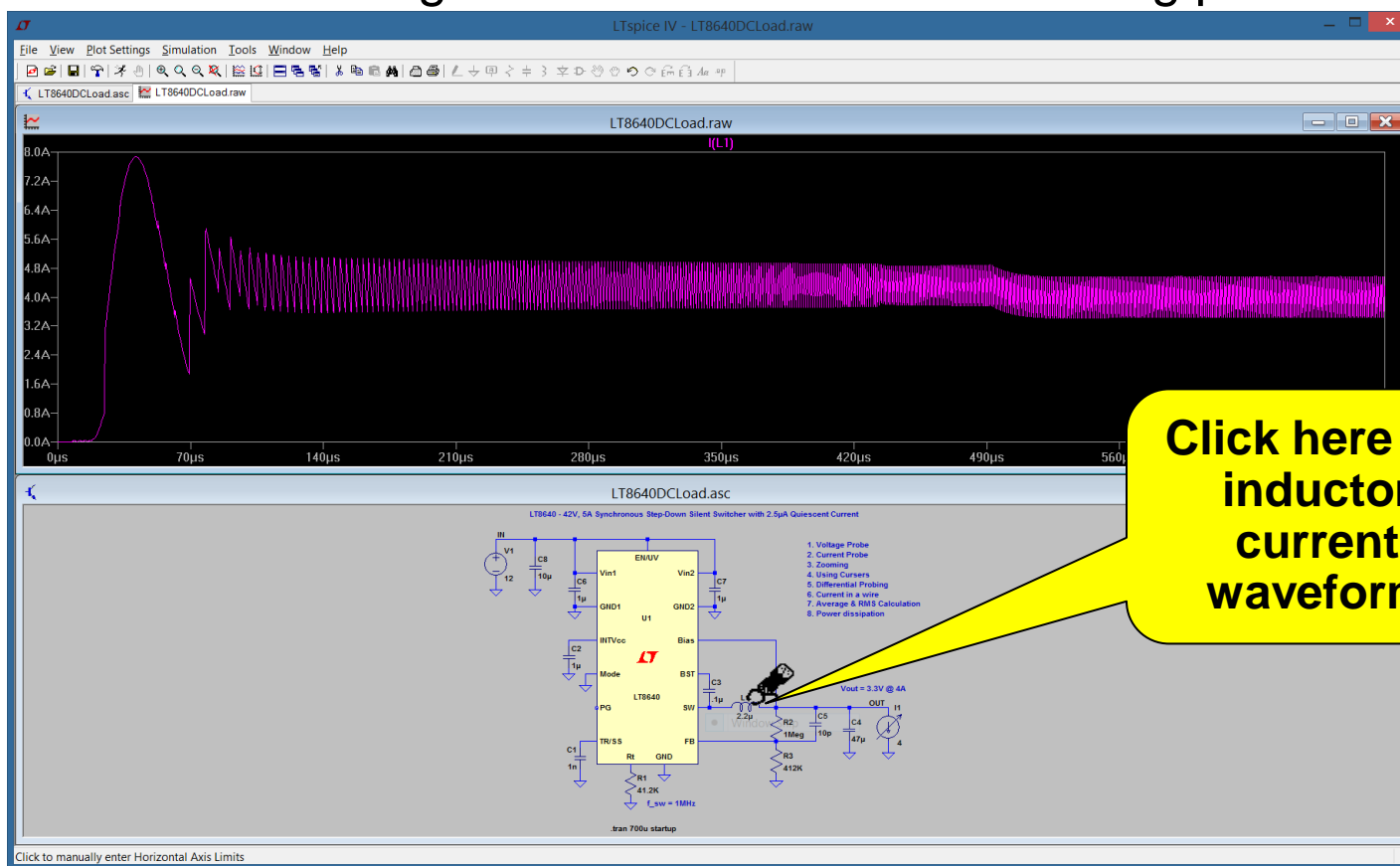
- ❖ Plot the voltage on any wire by **Left-Clicking** it
 - ❖ Tip: All Demo Circuits have INs and OUTs clearly labeled to help you quickly select them



LT8640DCLoad.asc

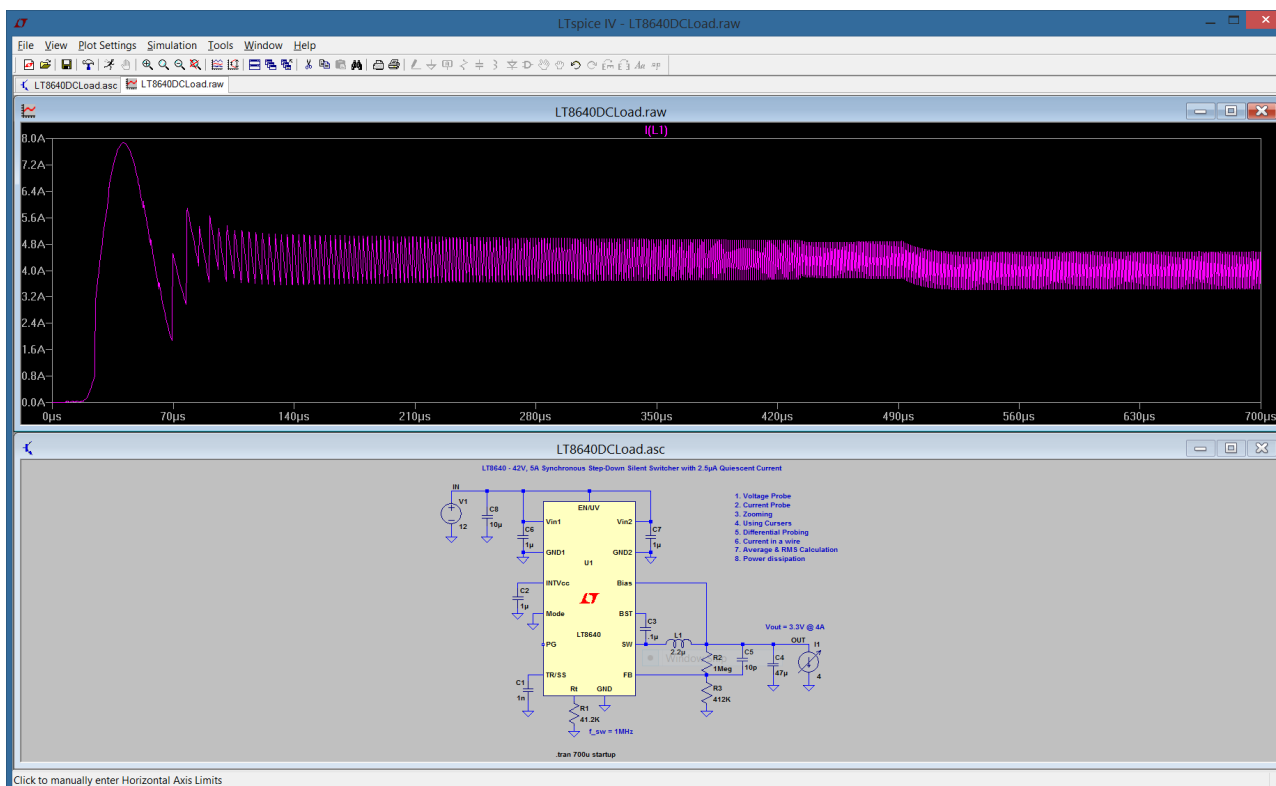
Viewing Current Waveforms

- ❖ 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

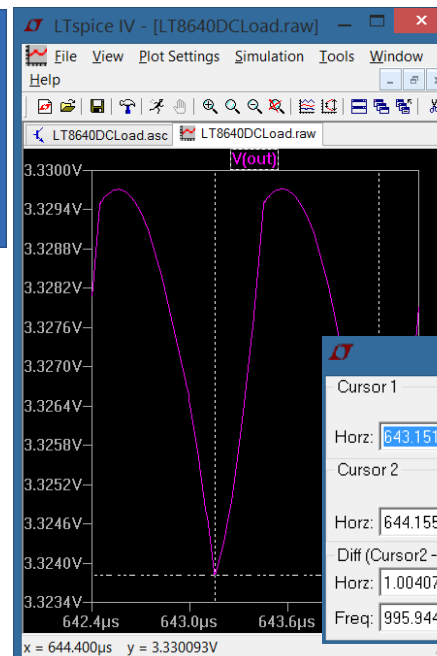
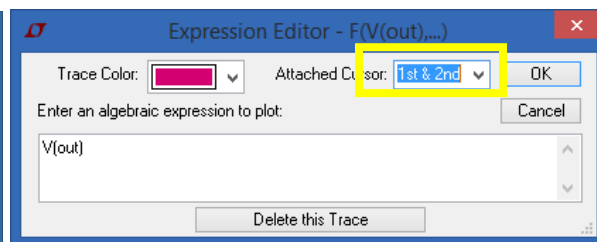
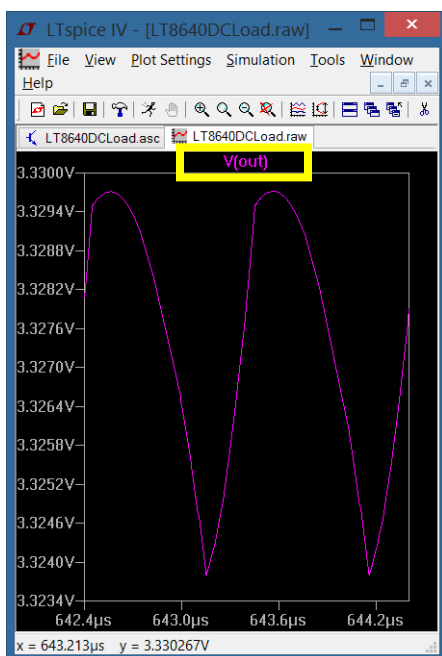
- ❖ In the waveform window, use the mouse to zoom in and out. Click and drag a box about the region you wish to see drawn larger
- ❖ Using the toolbar, click on “Zoom full extents”, to zoom back out



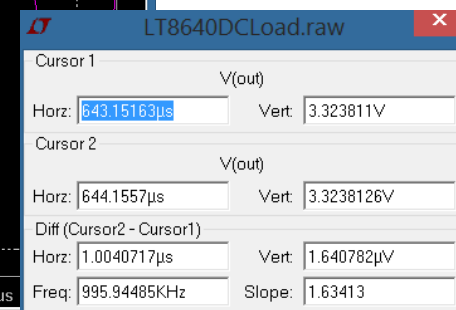
Measuring V, I and Time in the Waveform (Measurement Using Cursors)

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

1.  2.  3.

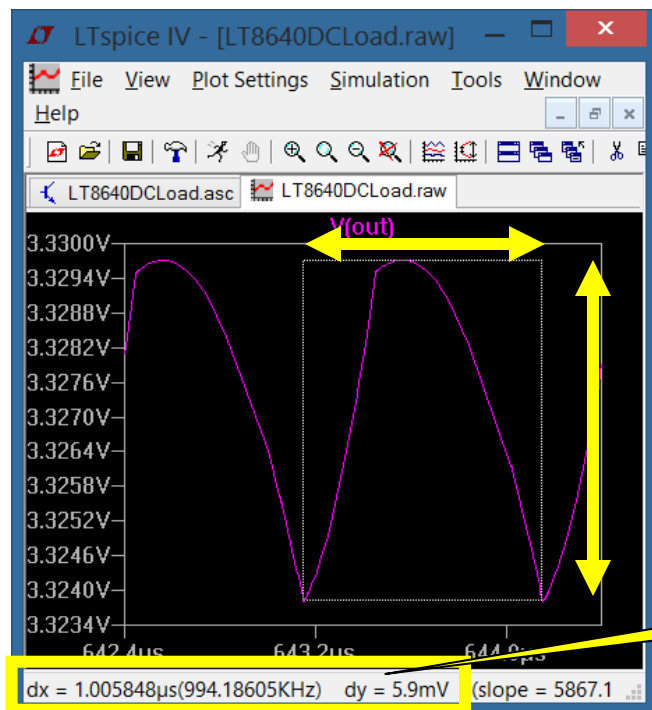


Result



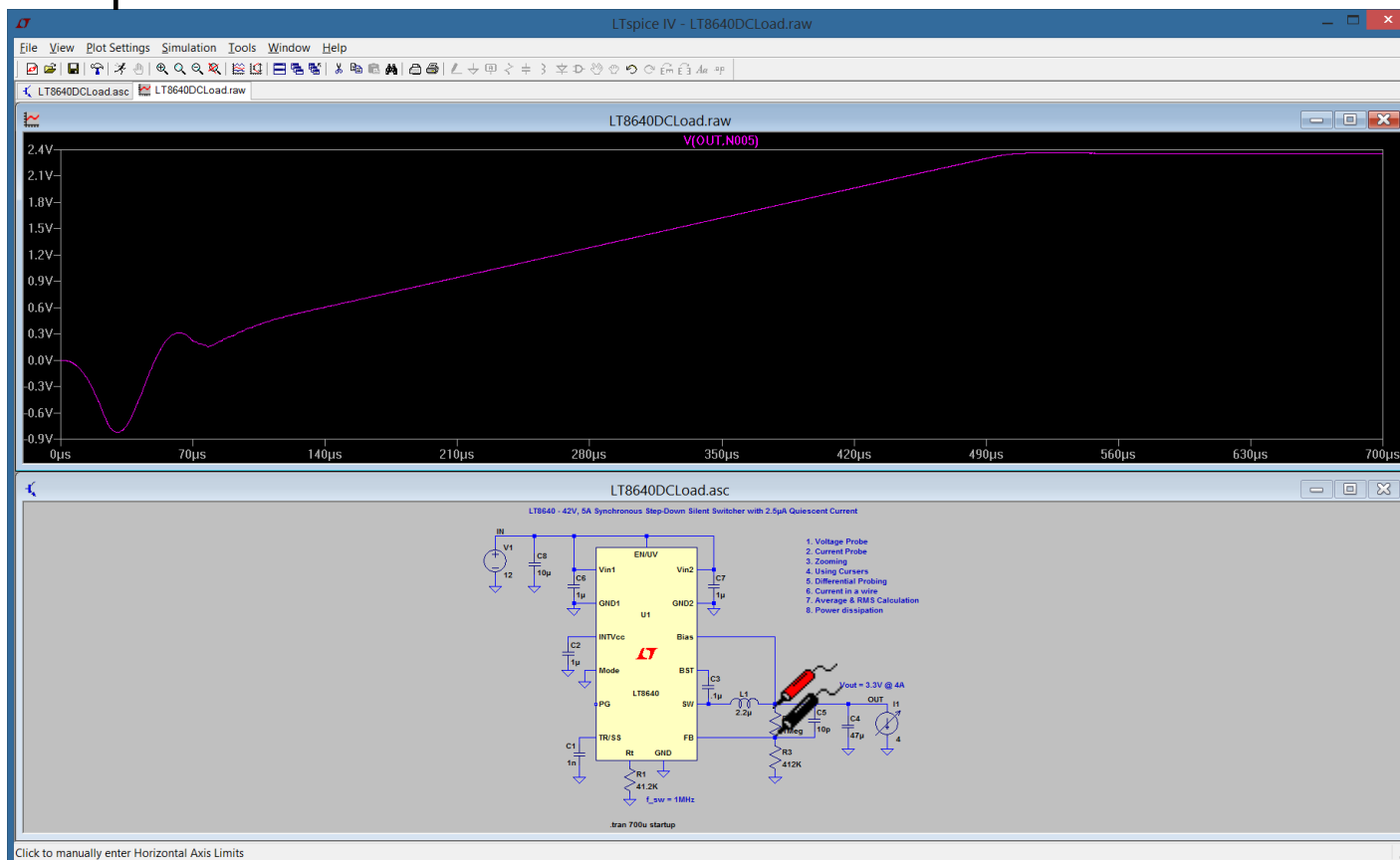
Measuring V, I and Time in the Waveform (Measurement Using Zoom Window)

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”



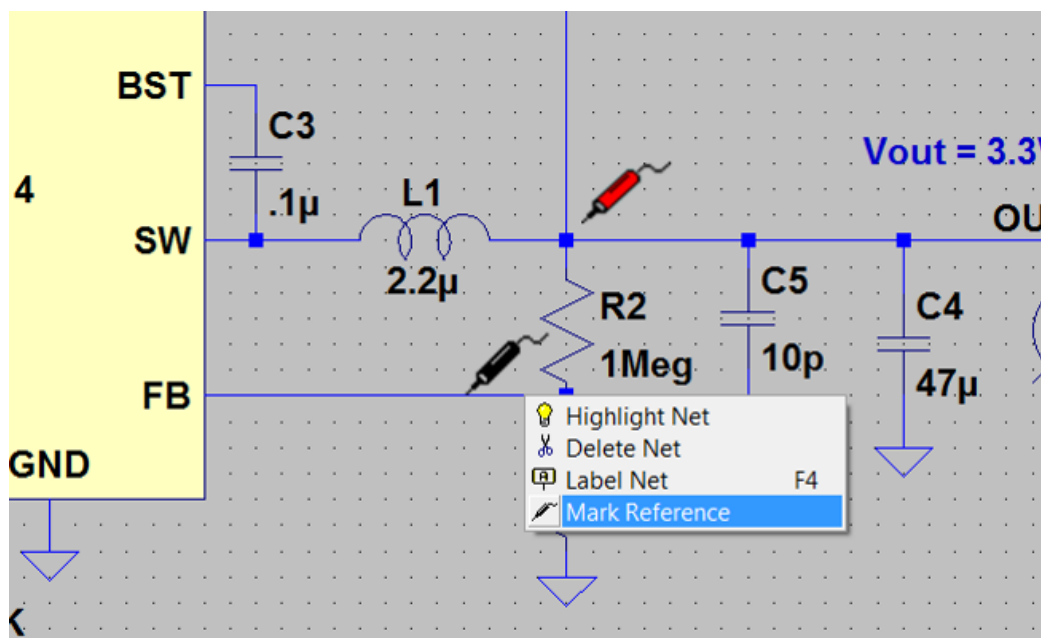
Viewing Differential Voltage Waveforms

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



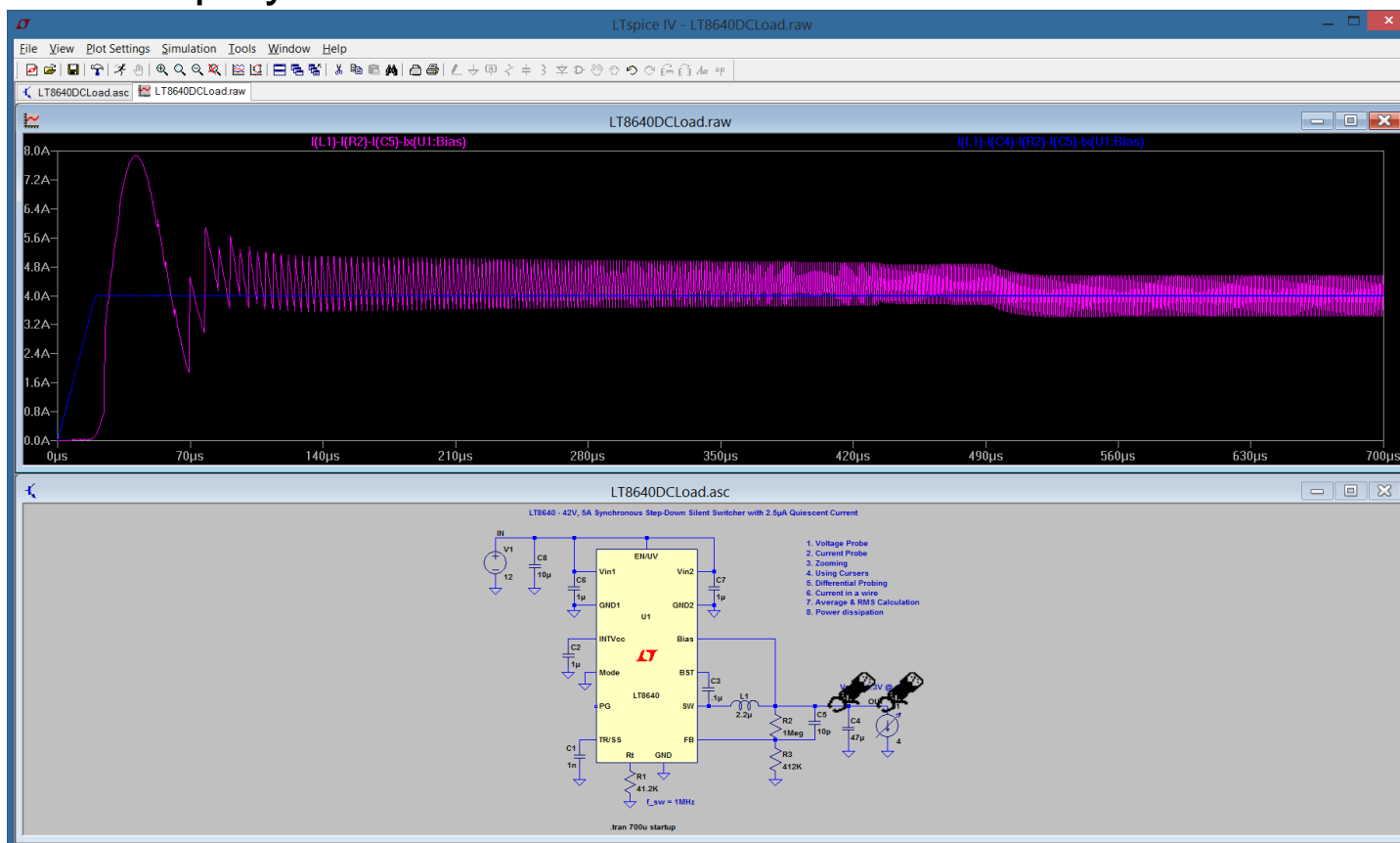
Viewing Differential Voltage Waveforms

- ❖ 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

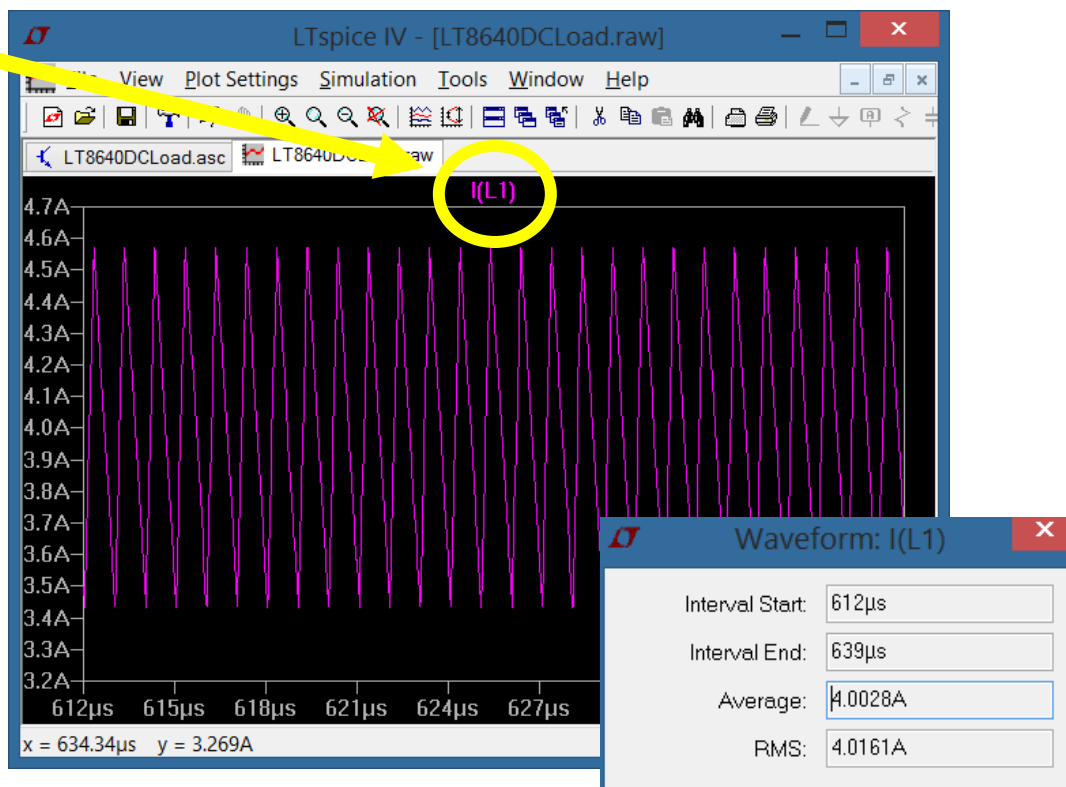


Average & RMS Calculations

- ❖ 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.

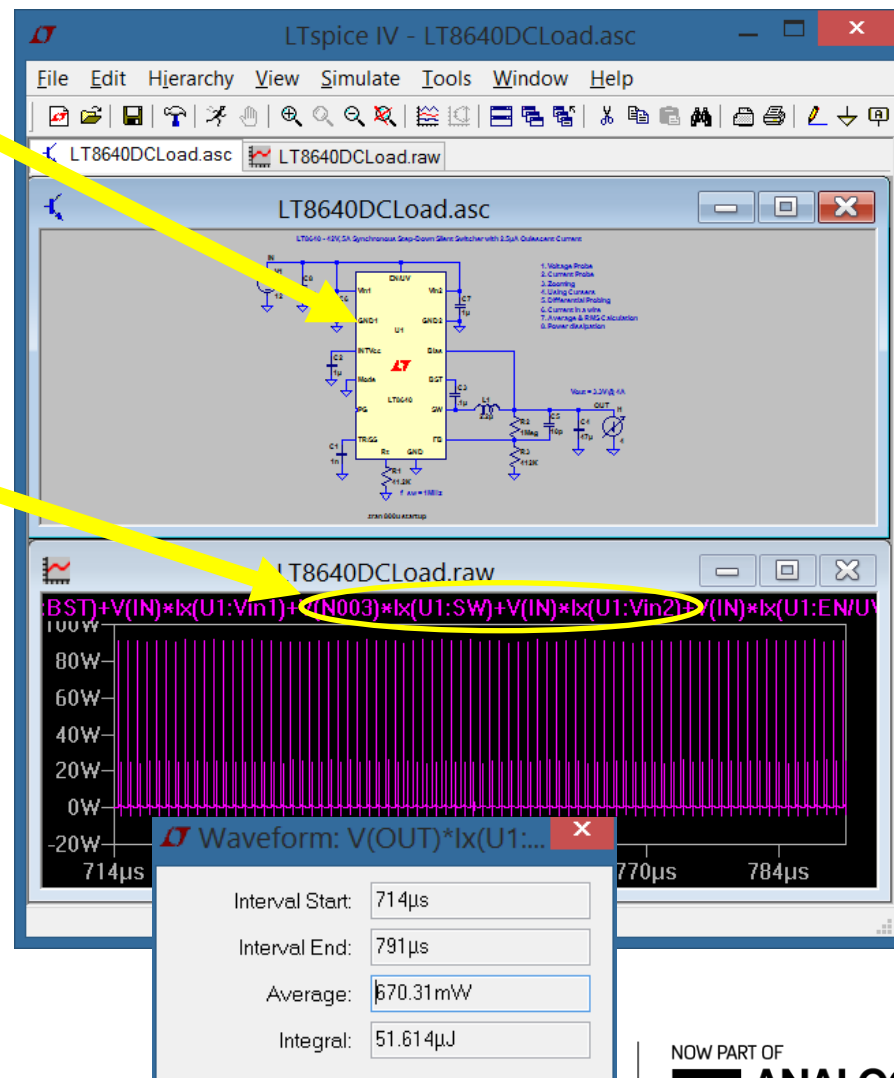


Instantaneous & Average Power Dissipation

- ❖ 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

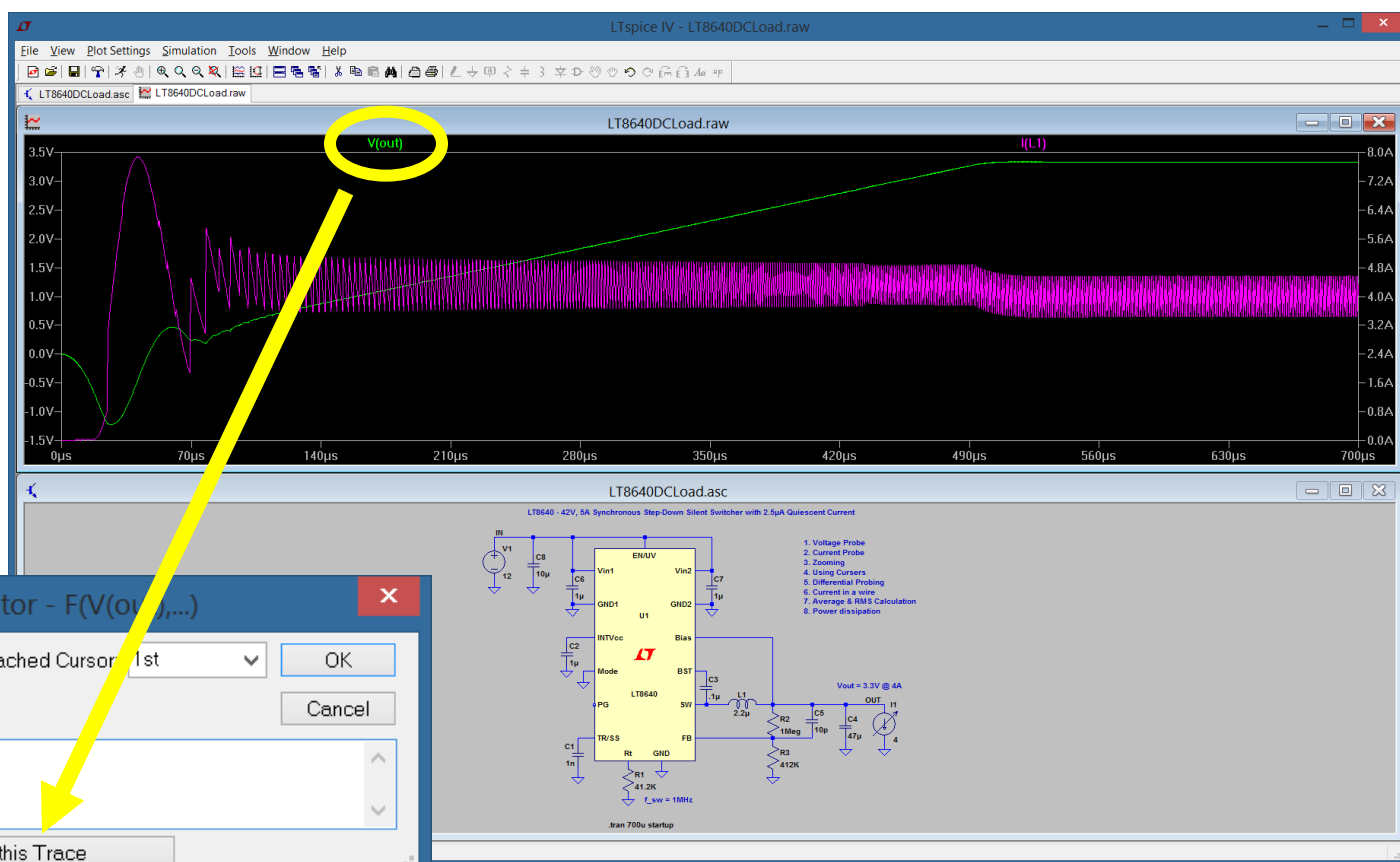
Example:

Measure the power dissipation in the LT8640 IC



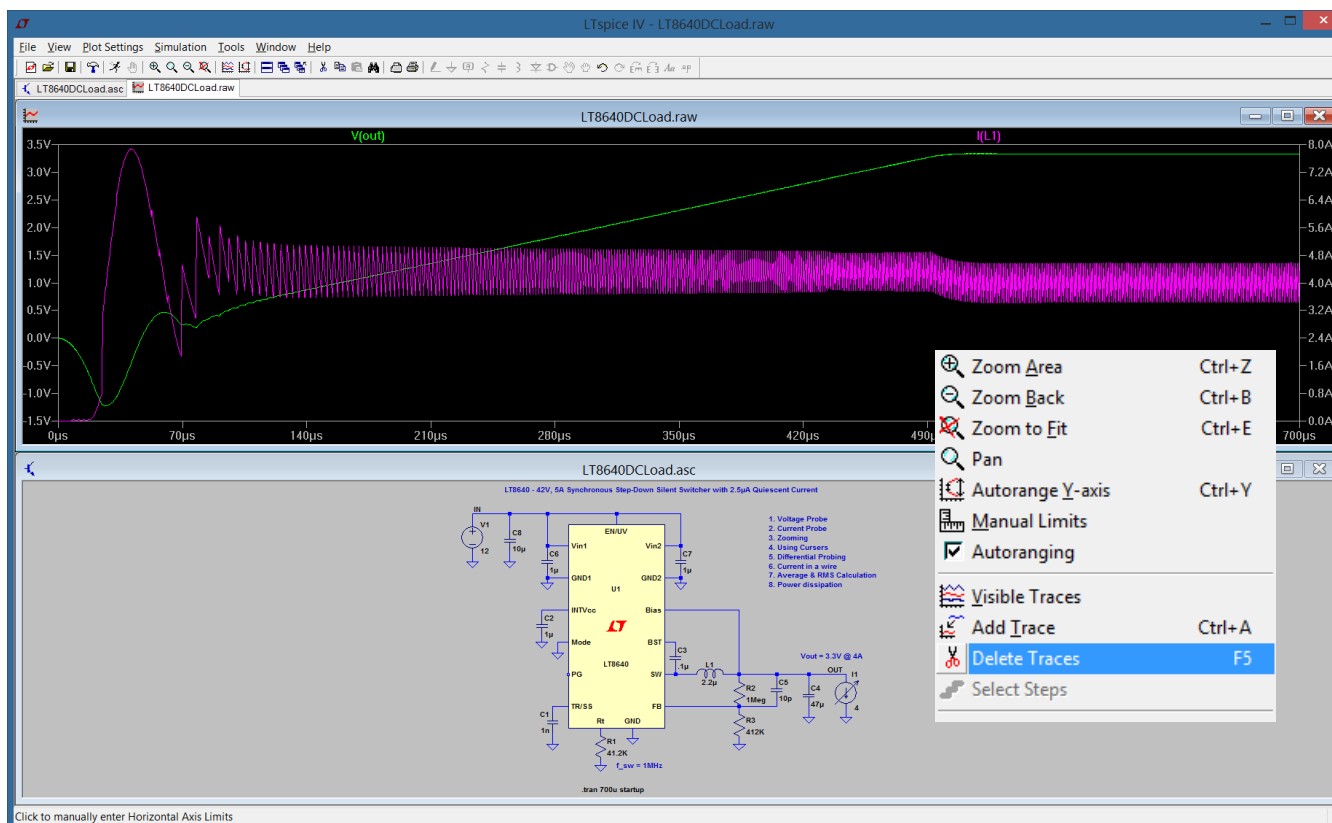
Deleting Waveforms

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



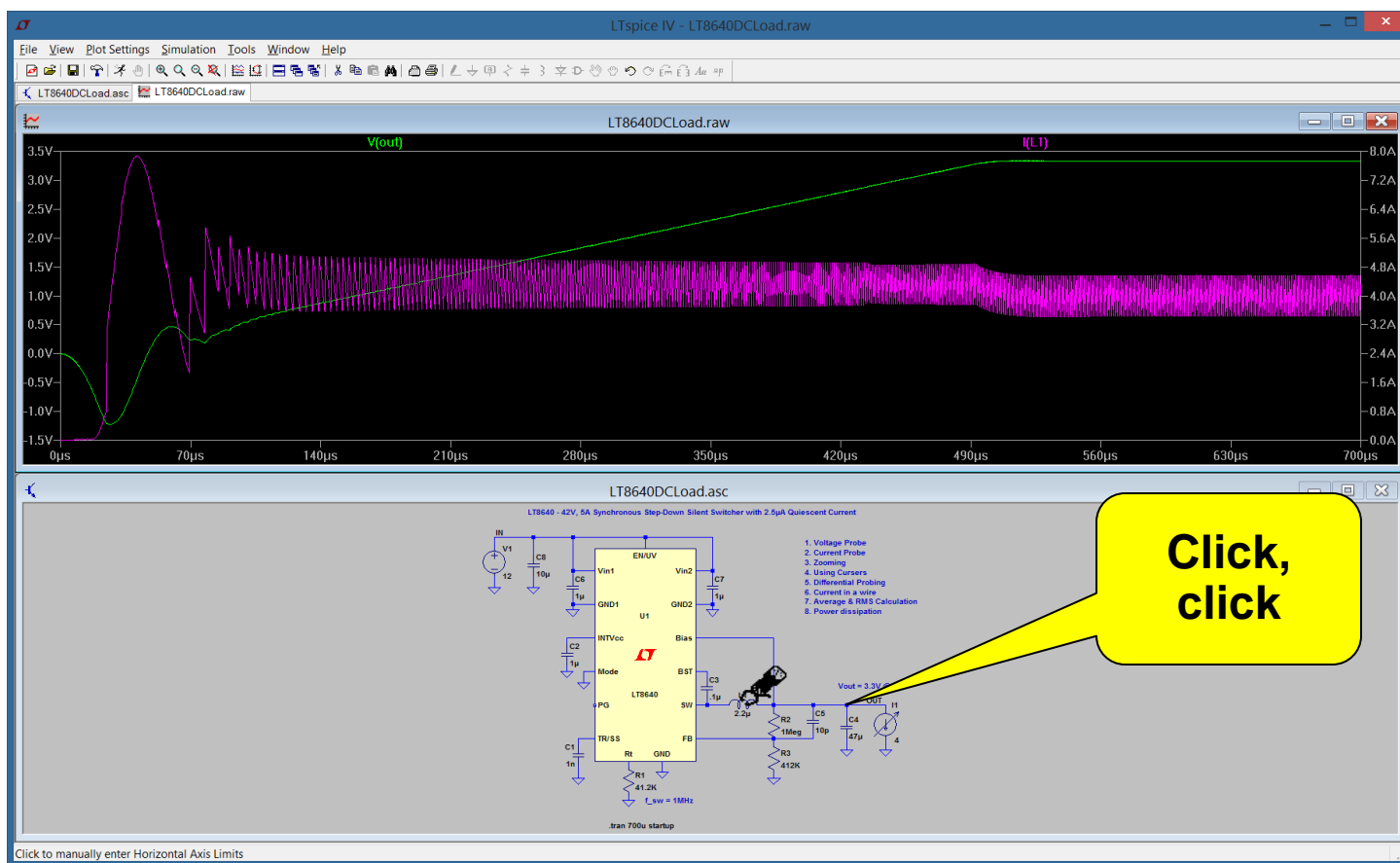
Deleting Waveforms

- ❖ 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



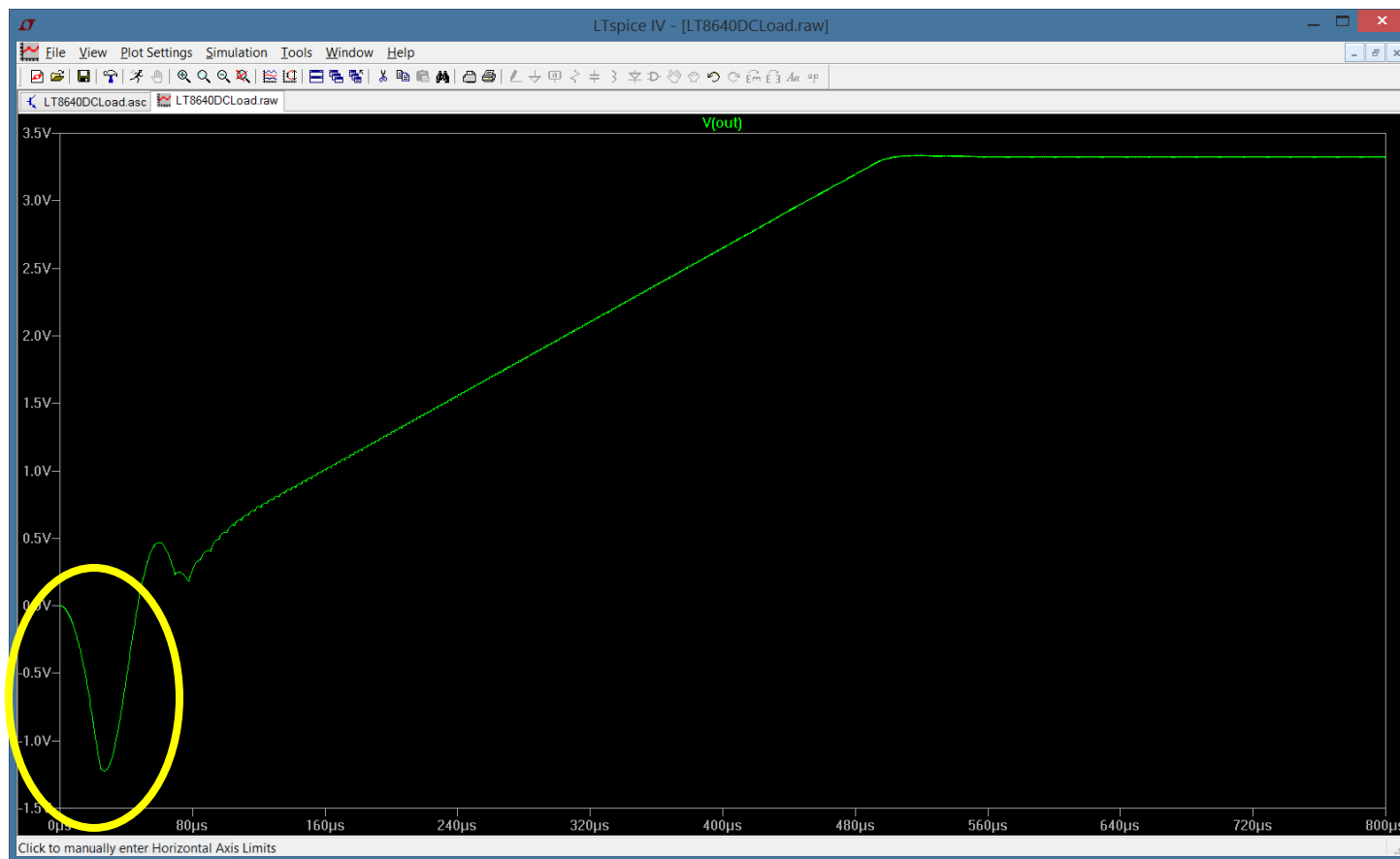
Deleting Waveforms

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



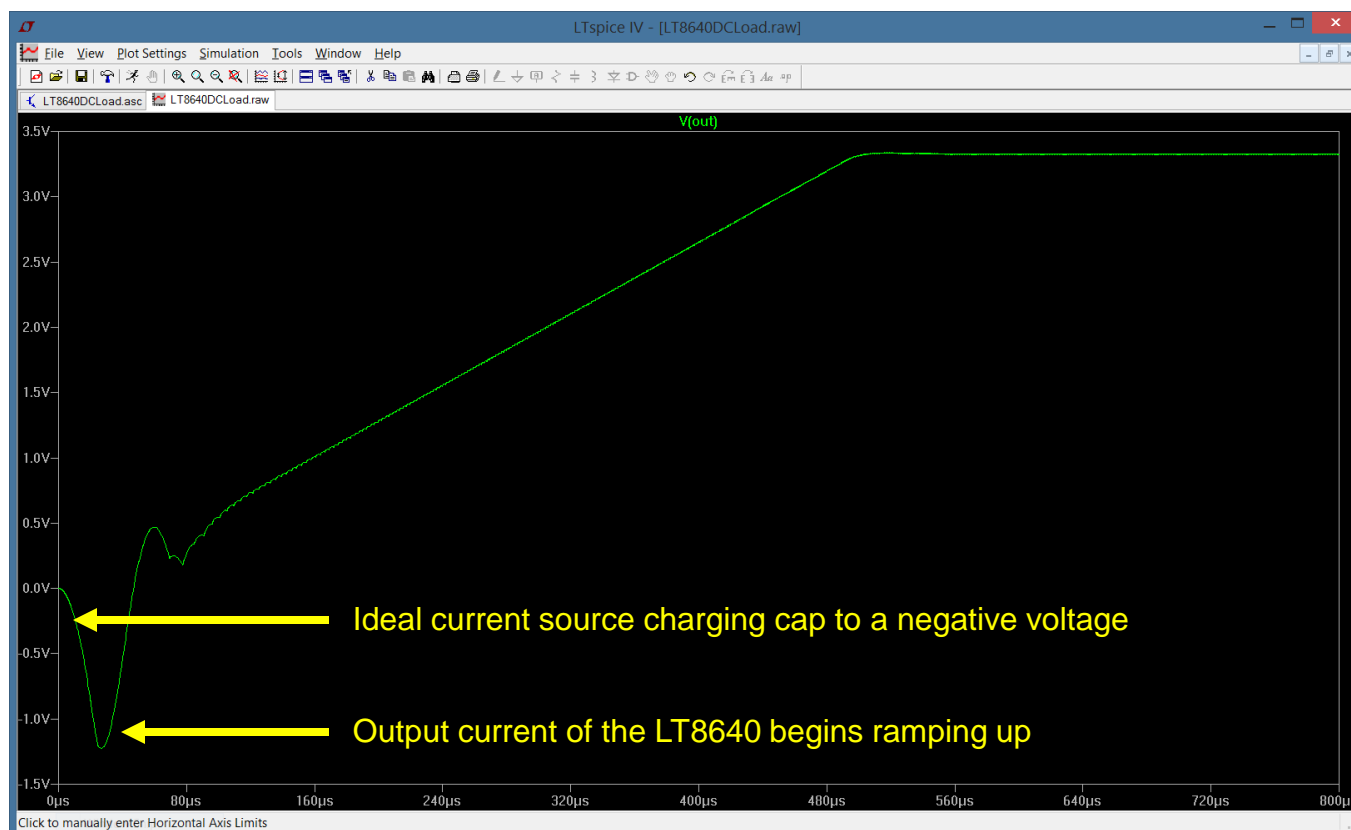
Current Sources

- ❖ 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.)

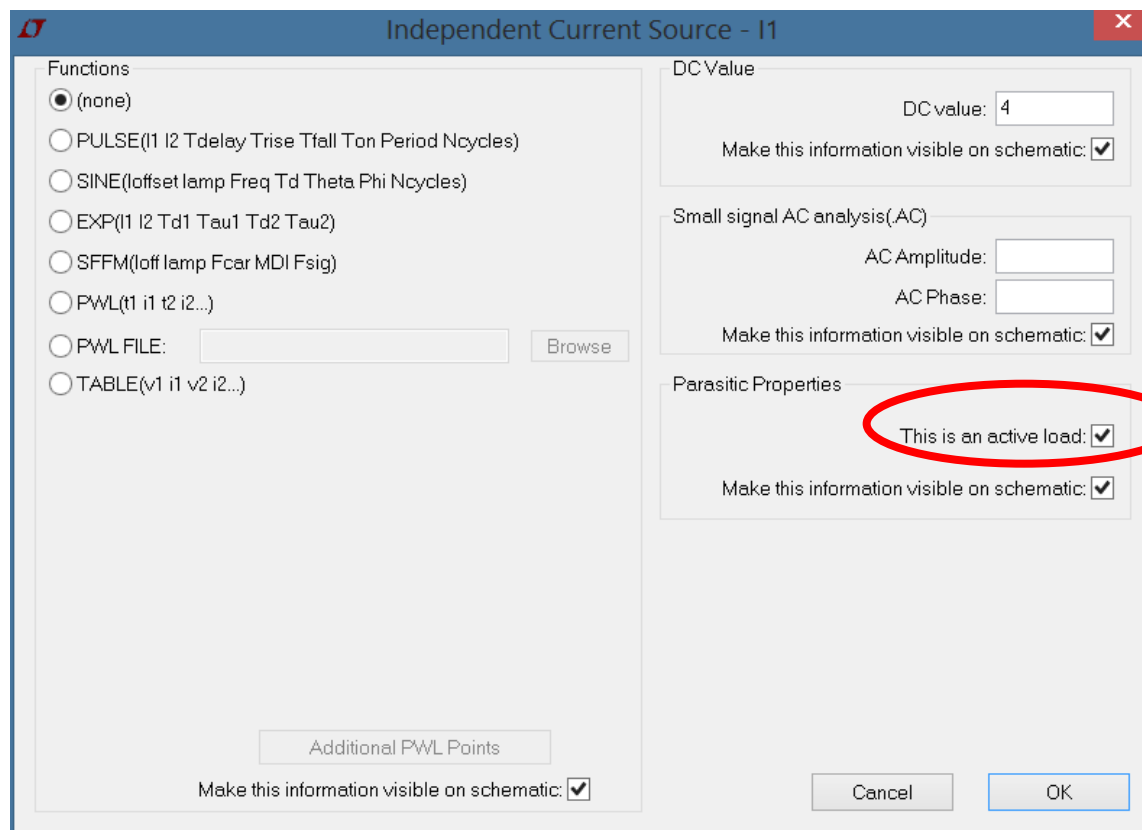
- ❖ Ideal current sources source current even with zero or negative differential across them
 - ❖ This results in the output capacitor initially charging to a negative value



Current Sources (cont.)

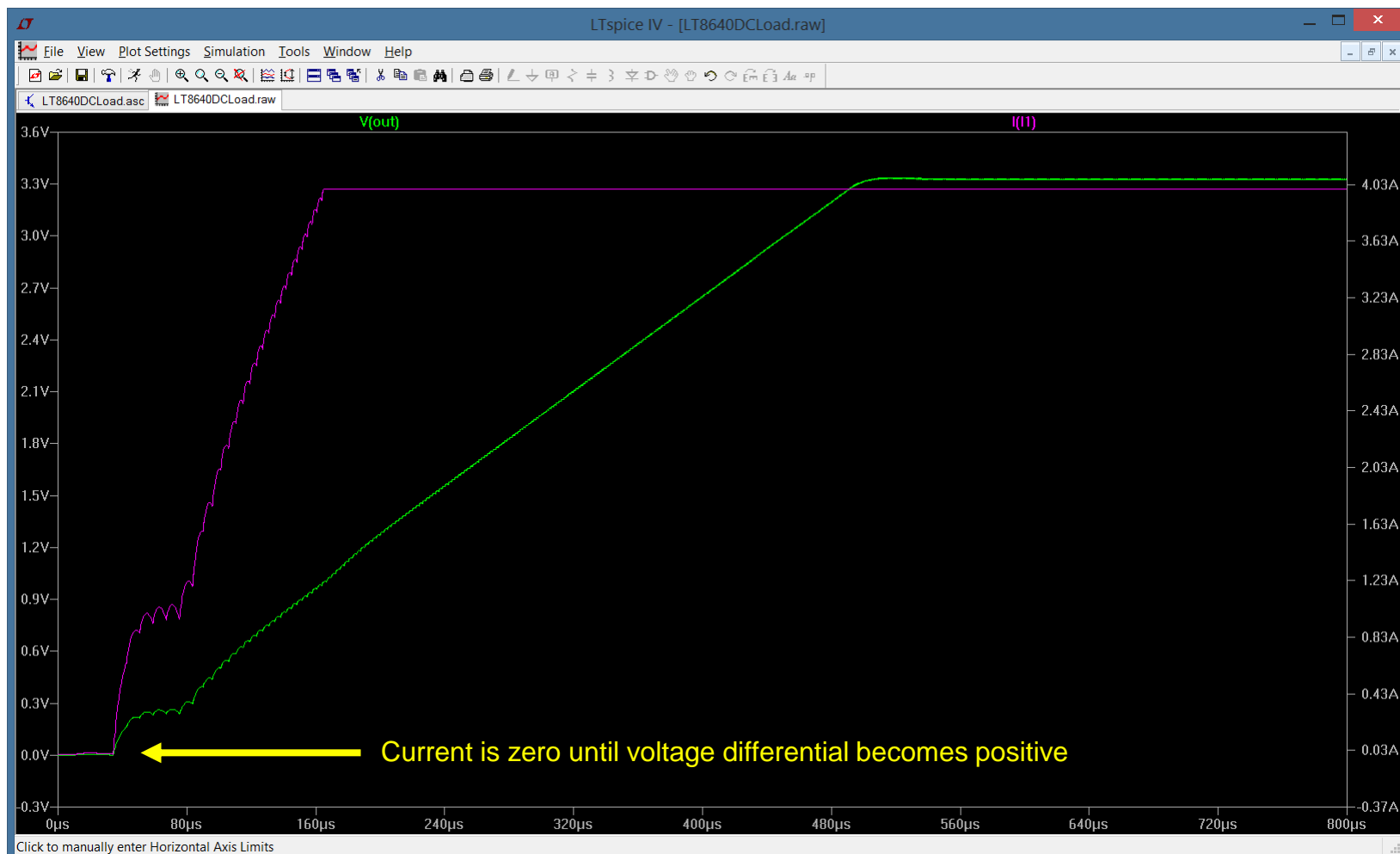
❖ Active load

- ❖ 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.)

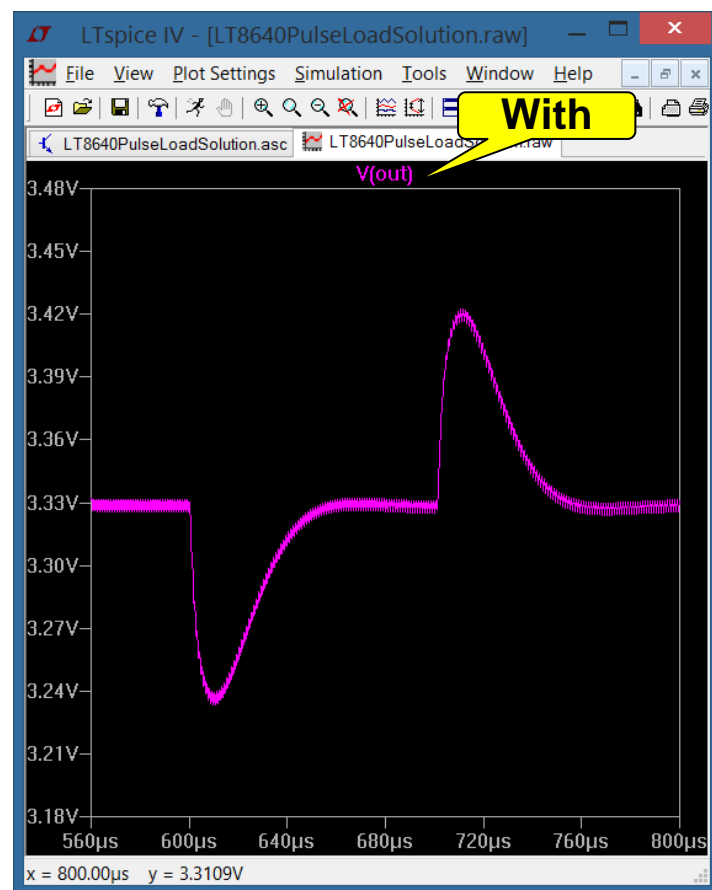
❖ Active load (cont.)



Net Labeling

Advantages of Labeling

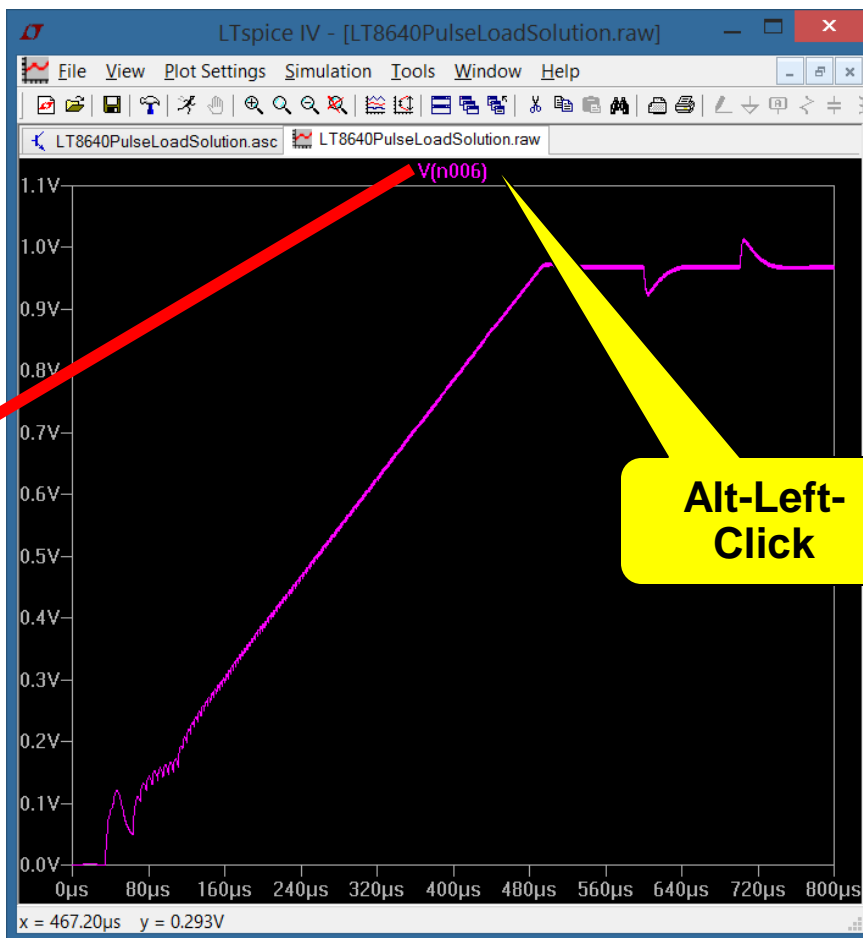
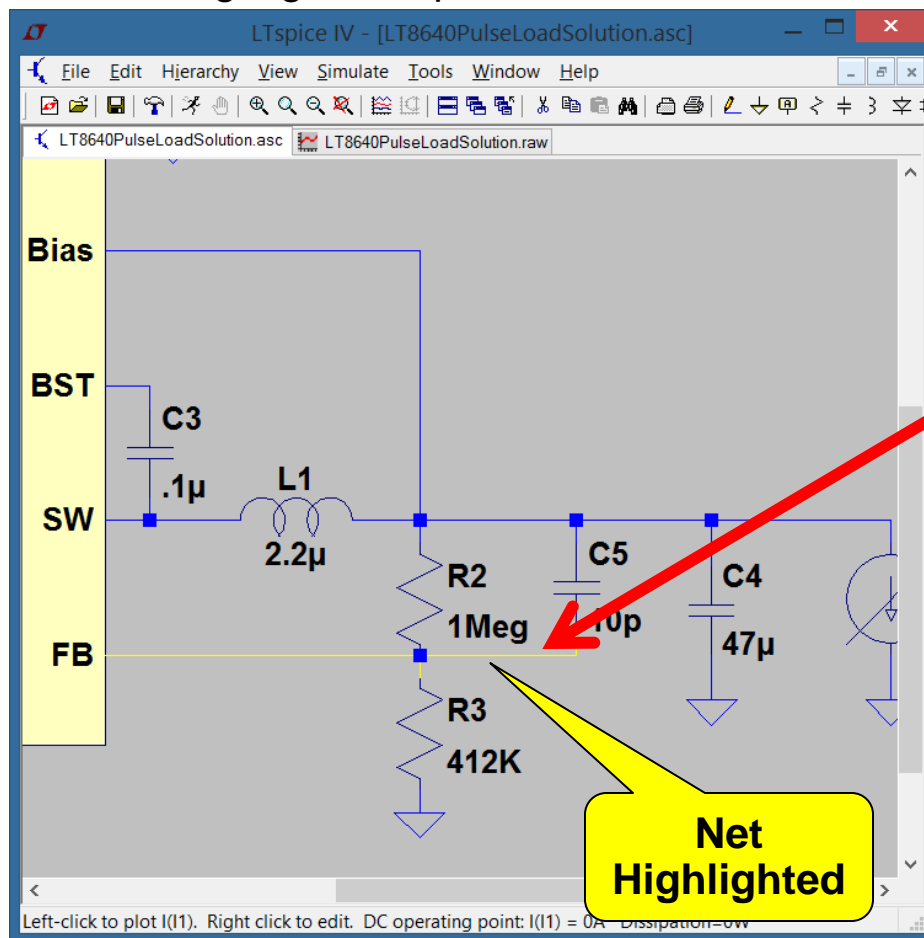
- ❖ Replaces arcane SPICE machine node names with easy to understand and remember human names
- ❖ Allows LTspice circuit nodes to match those on your production schematic, i.e. “TP15”



Labeling - Trick

❖ Highlight net from waveform viewer

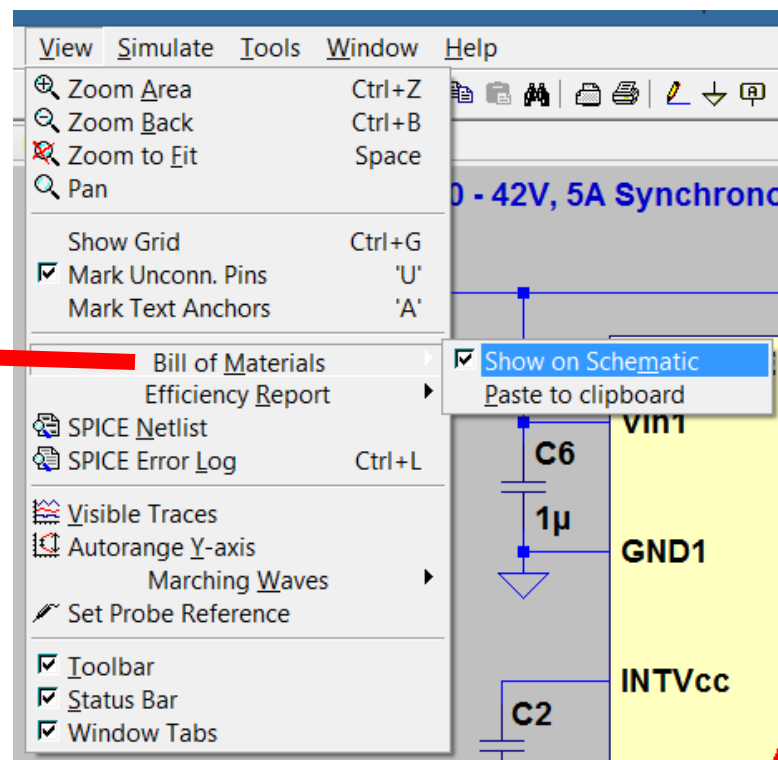
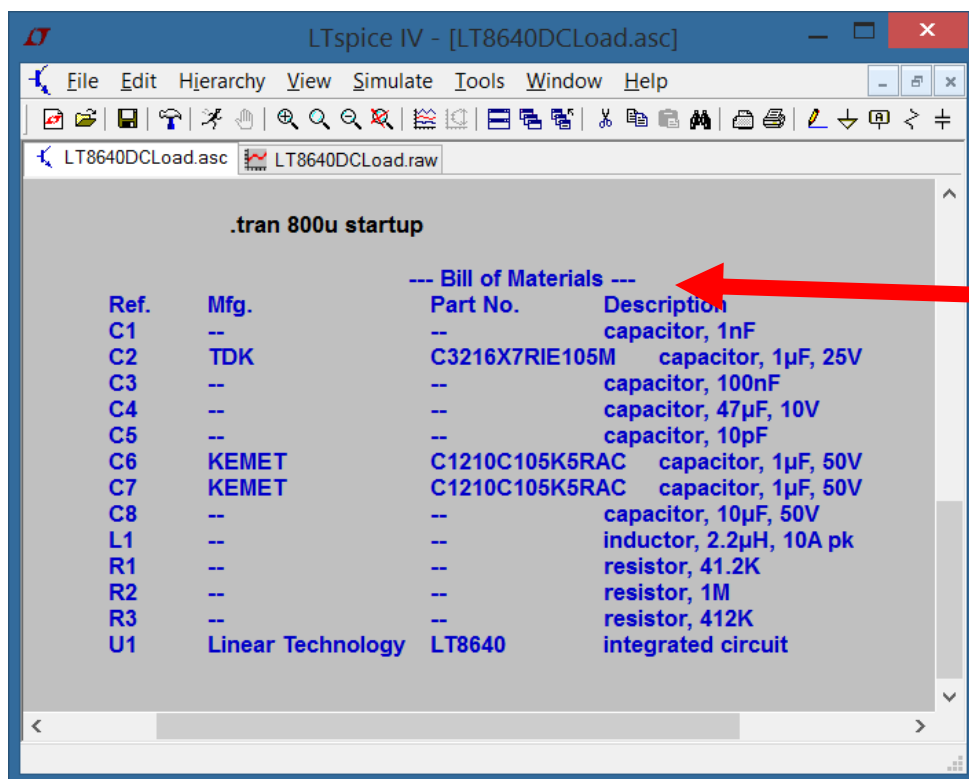
- ❖ **Alt-Left-Click** on the label in the waveform viewer (i.e. V(n006)) and it will now highlight that particular net on the schematic



Generating a BOM and Efficiency Report

BOM

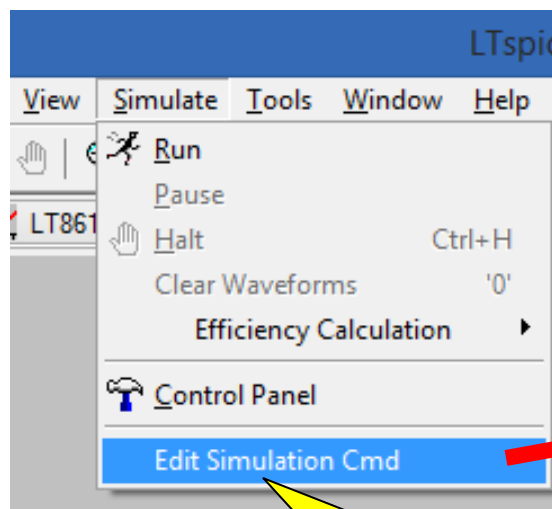
- ❖ Under View select Bill of Material
 - ❖ Displayed on Diagram
 - ❖ Paste to Clipboard



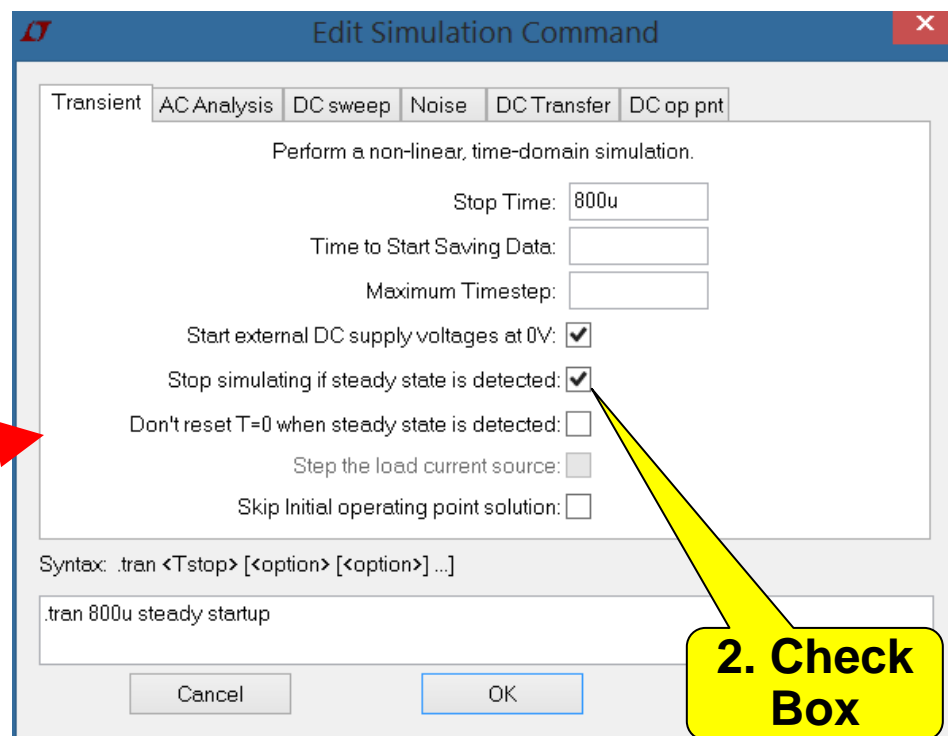
Steps to Computing Efficiency

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” ...



**1. Simulate menu,
select “Edit
simulation Cmd”**

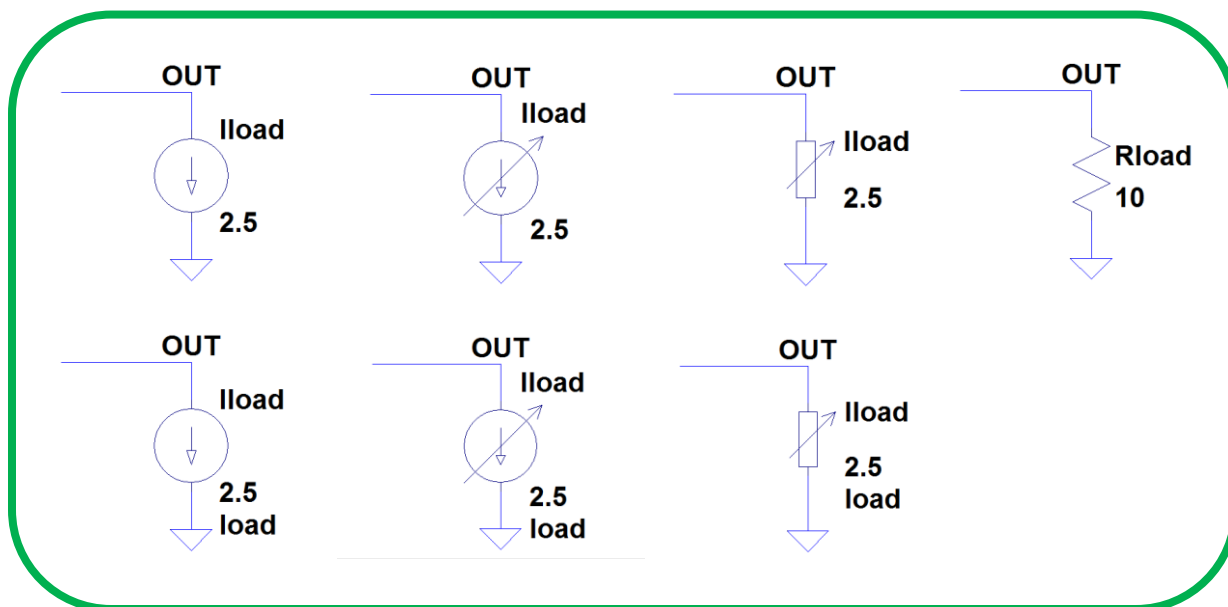


**2. Check
Box**

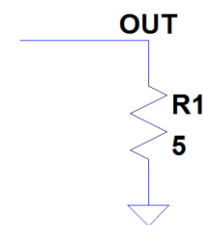
Steps to Computing Efficiency

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**

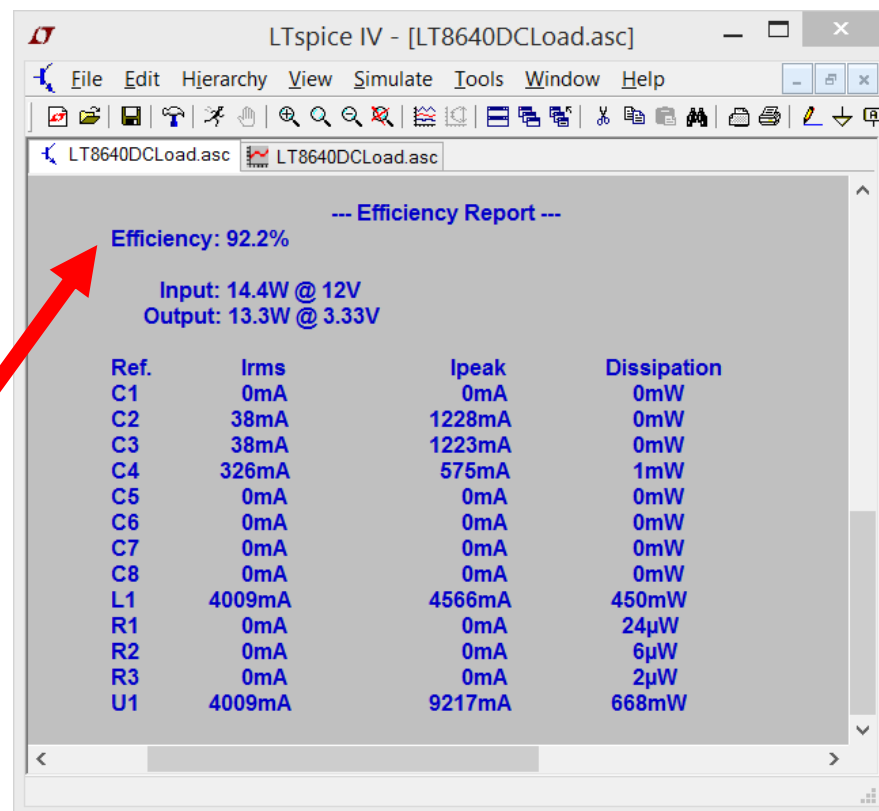
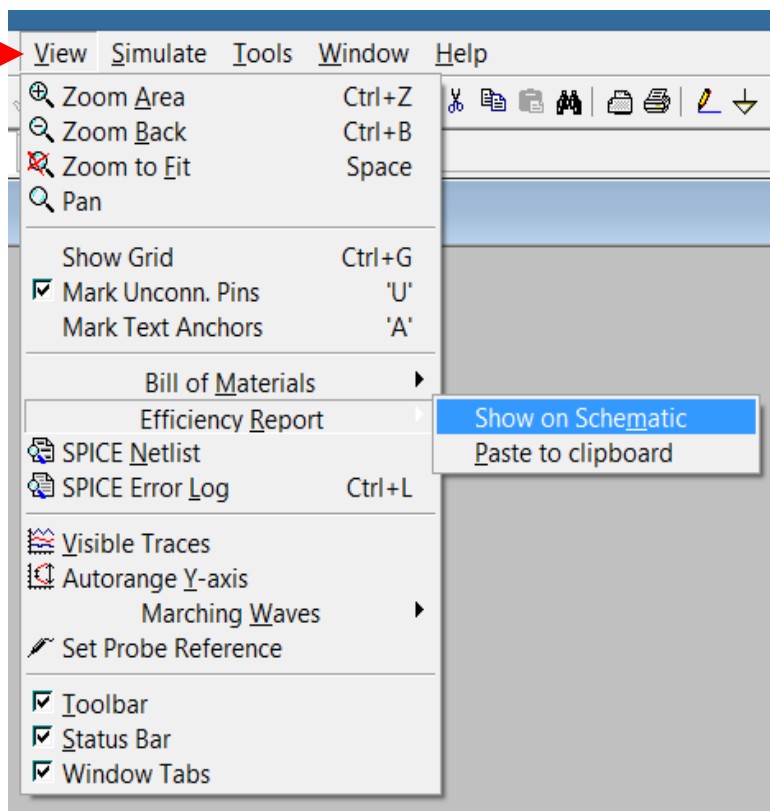
*If there is more than one current source

**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

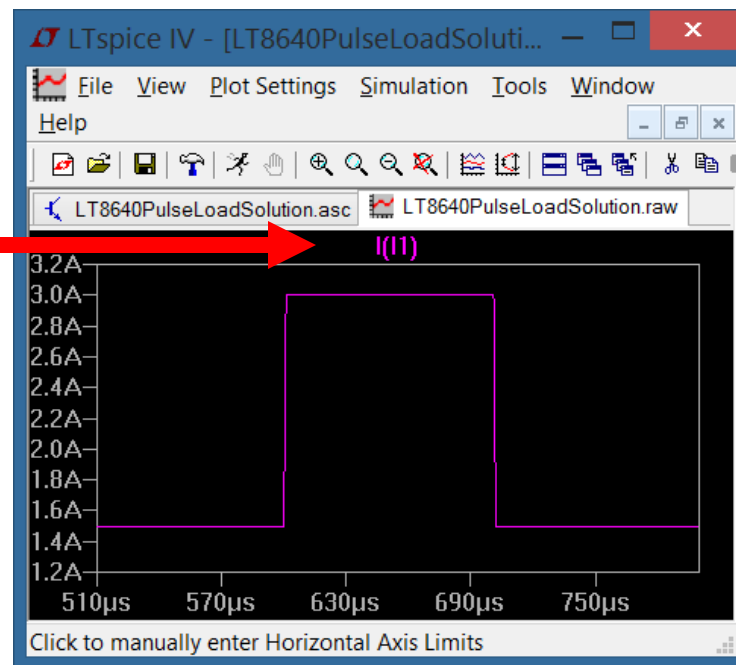
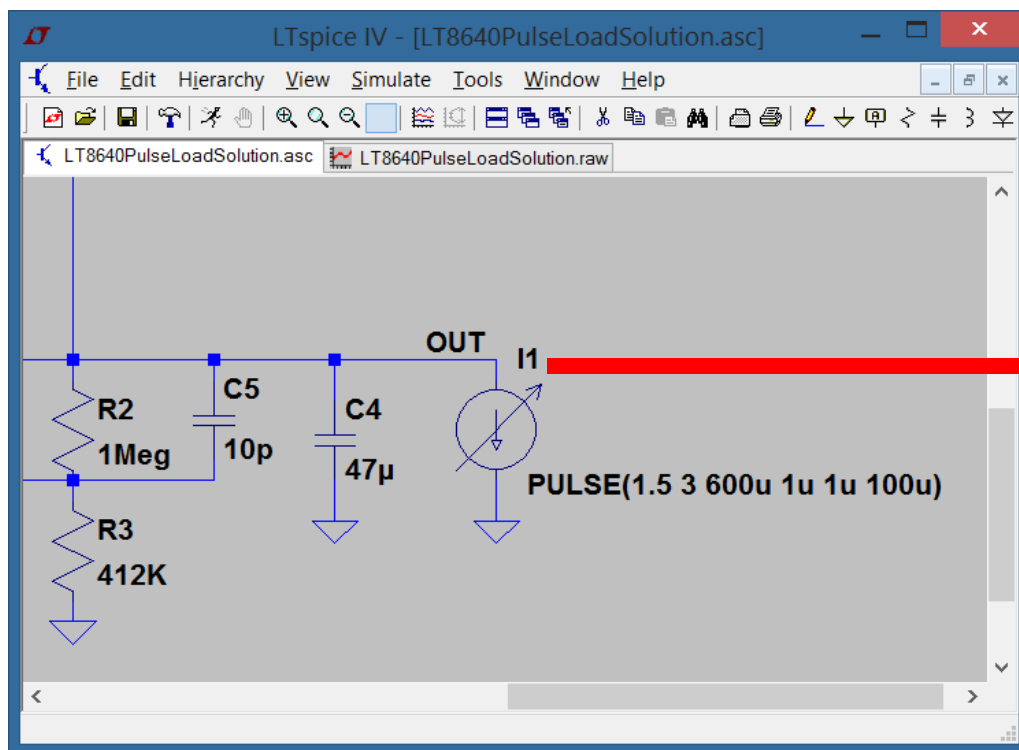
- ❖ LTspice will not always be able to determine steady state, but this is rare!
 - ❖ Workaround: **Alt-Left-Click** & Calculator
- ❖ Probe the various nodes and verify the circuit is stabilized
 - ❖ If not edit the .tran statement and increase the Stop Time parameter. Re-run simulation
- ❖ For multiple 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



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”

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...)

☐ PWL FILE:

☐ TABLE(v1 i1 v2 i2...)

I1[A]:

I2[A]:

Tdelay[s]:

Trise[s]:

Tfall[s]:

Ton[s]:

Tperiod[s]:

Ncycles:

Make this information visible on schematic: ☒

DC Value

DC value:

Make this information visible on schematic: ☒

Small signal AC analysis(.AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic: ☒

Parasitic Properties

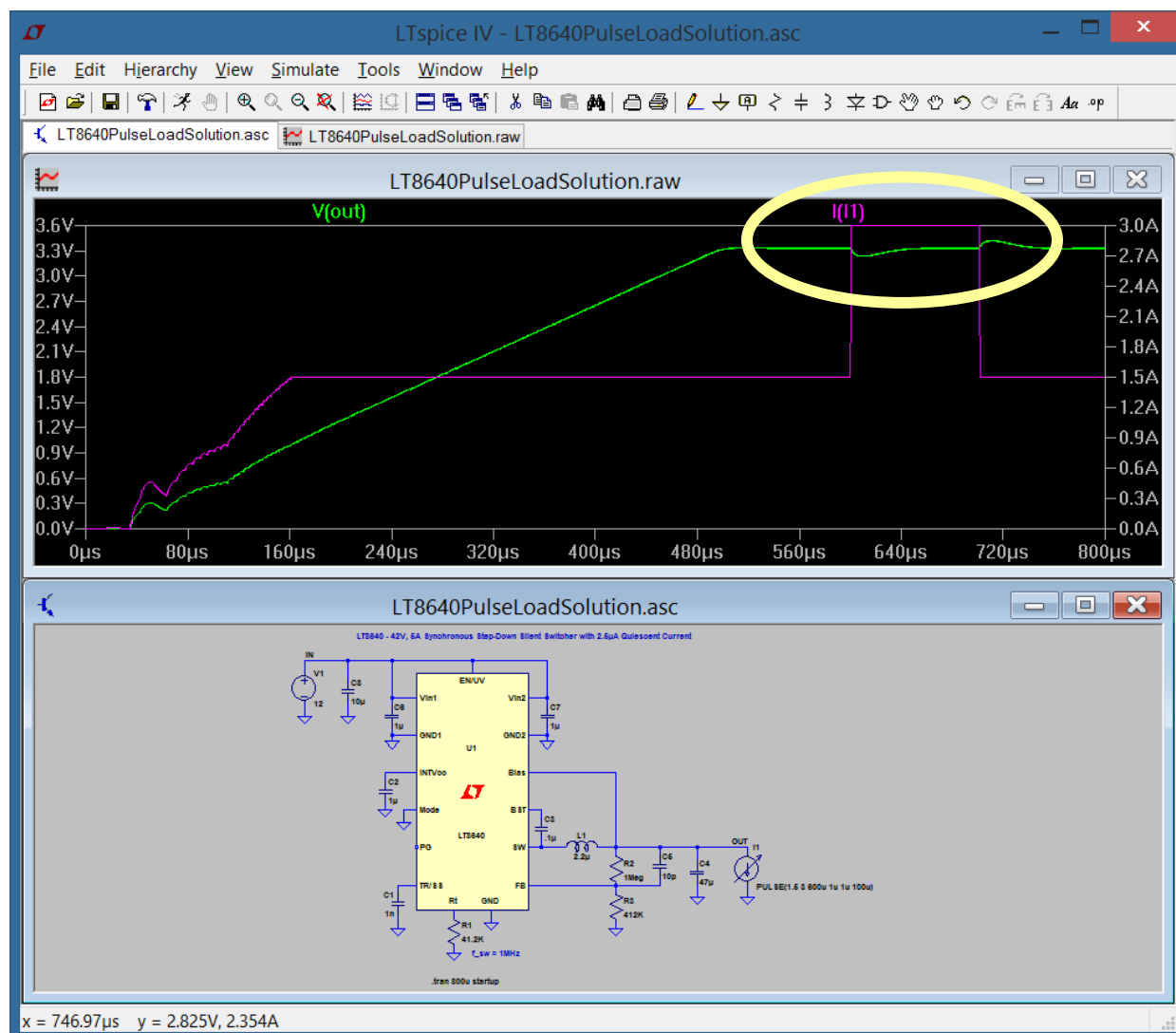
(*) This is an active load: ☒

Make this information visible on schematic: ☐

(*) 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



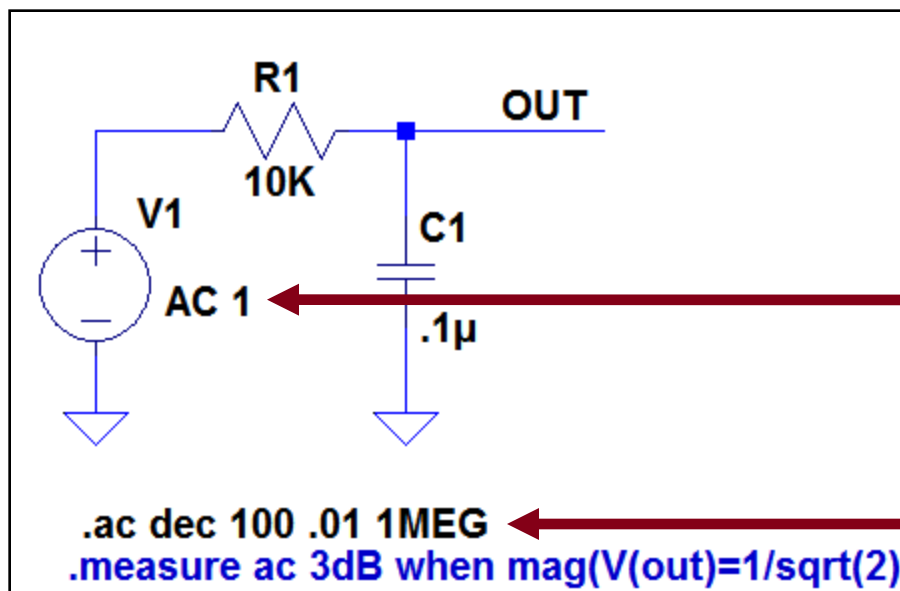
AC Analysis

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`

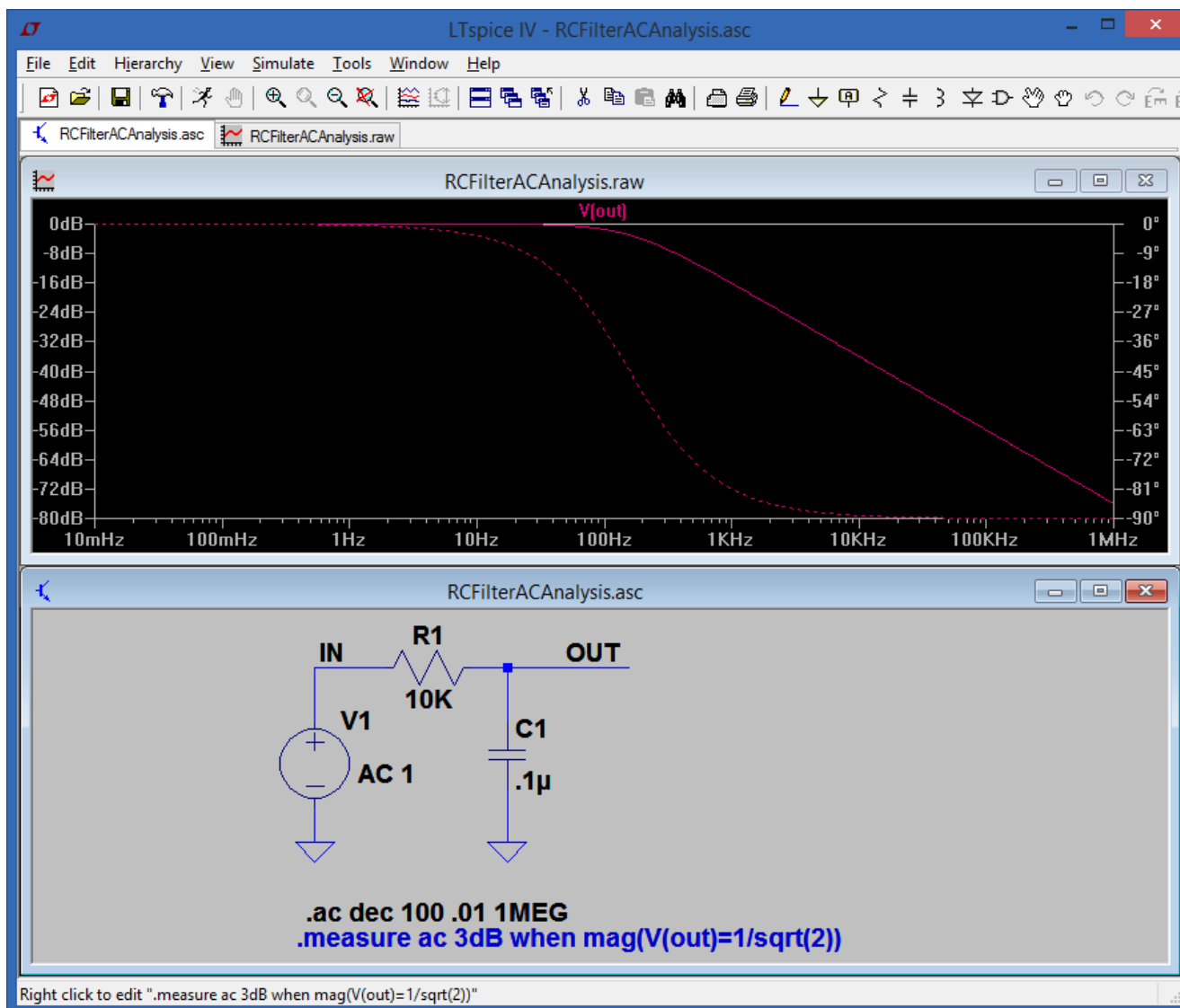


-3dB point:
 $1/(2 \cdot \pi \cdot R \cdot C) = 159\text{Hz}$

AC amplitude must be set to 1 for accurate results

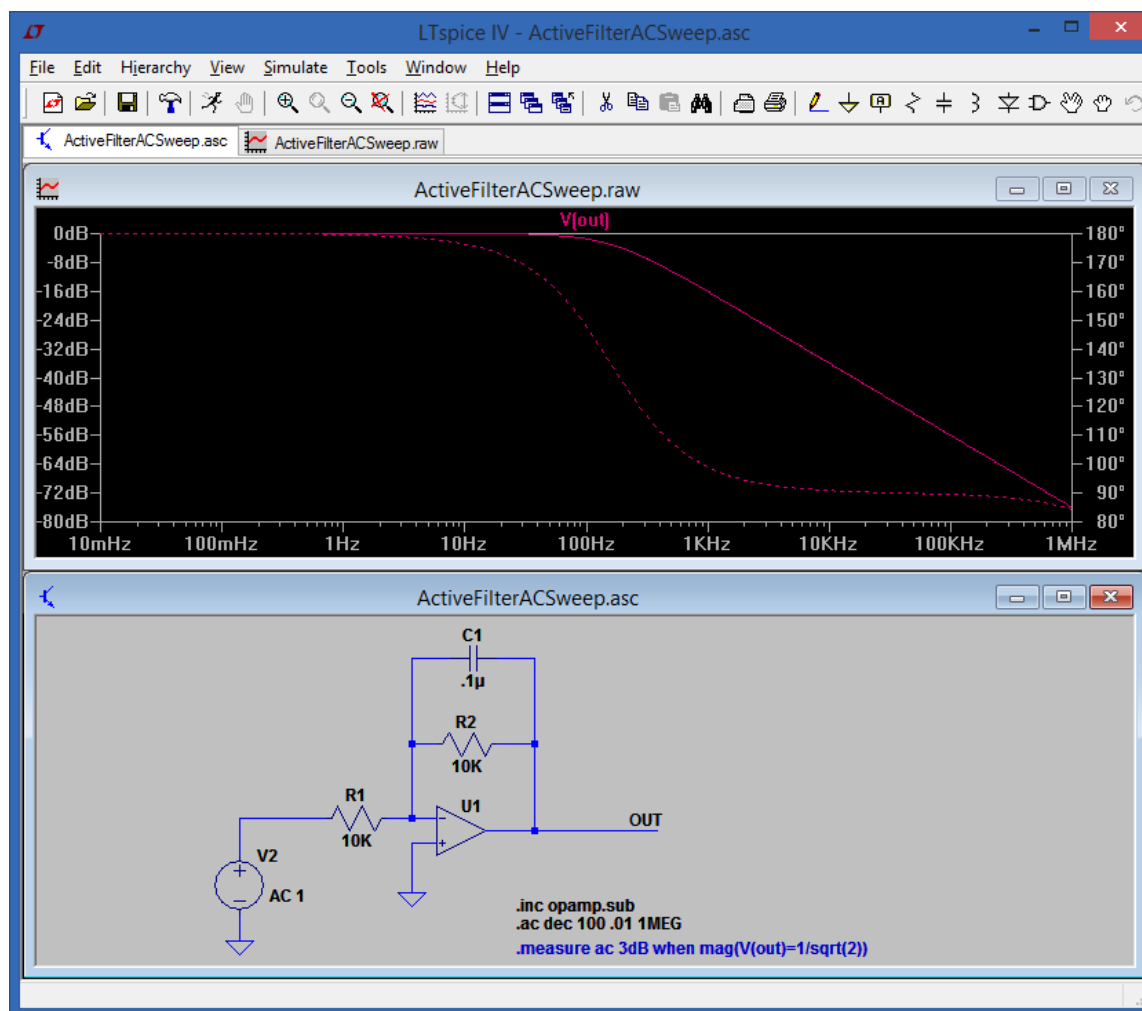
Right-click on .tran command and select “AC Analysis”

Simulating AC Analysis – RC Result



Simulating AC Analysis – Active Filter

- ❖ Single pole active filter using an opamp



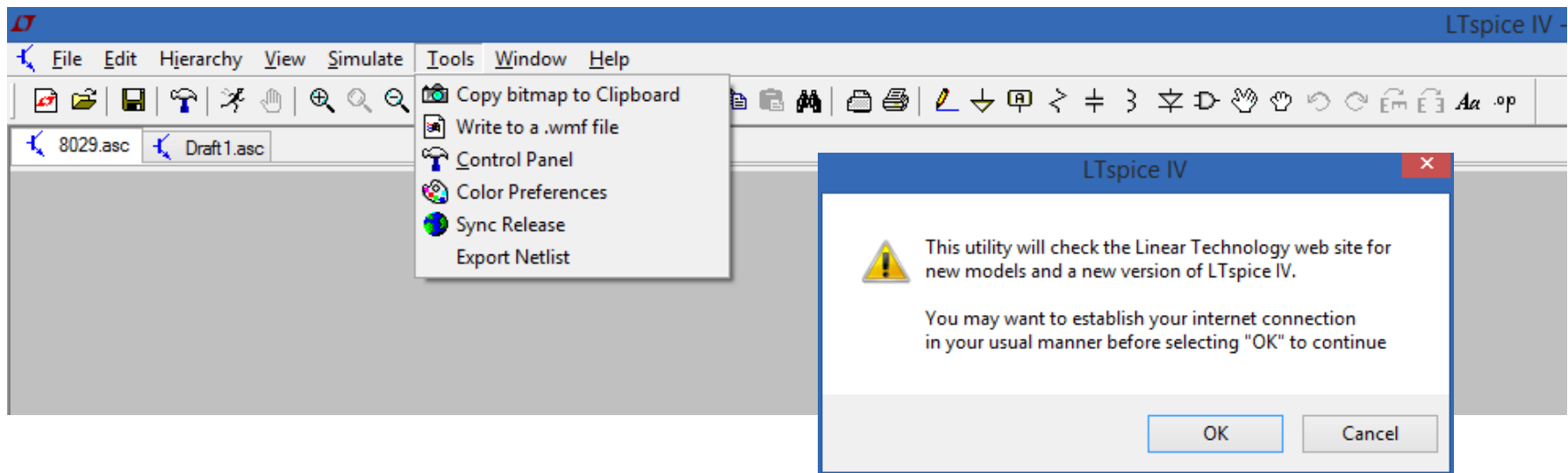
More Information and Support

Reminder to Periodically Sync Release

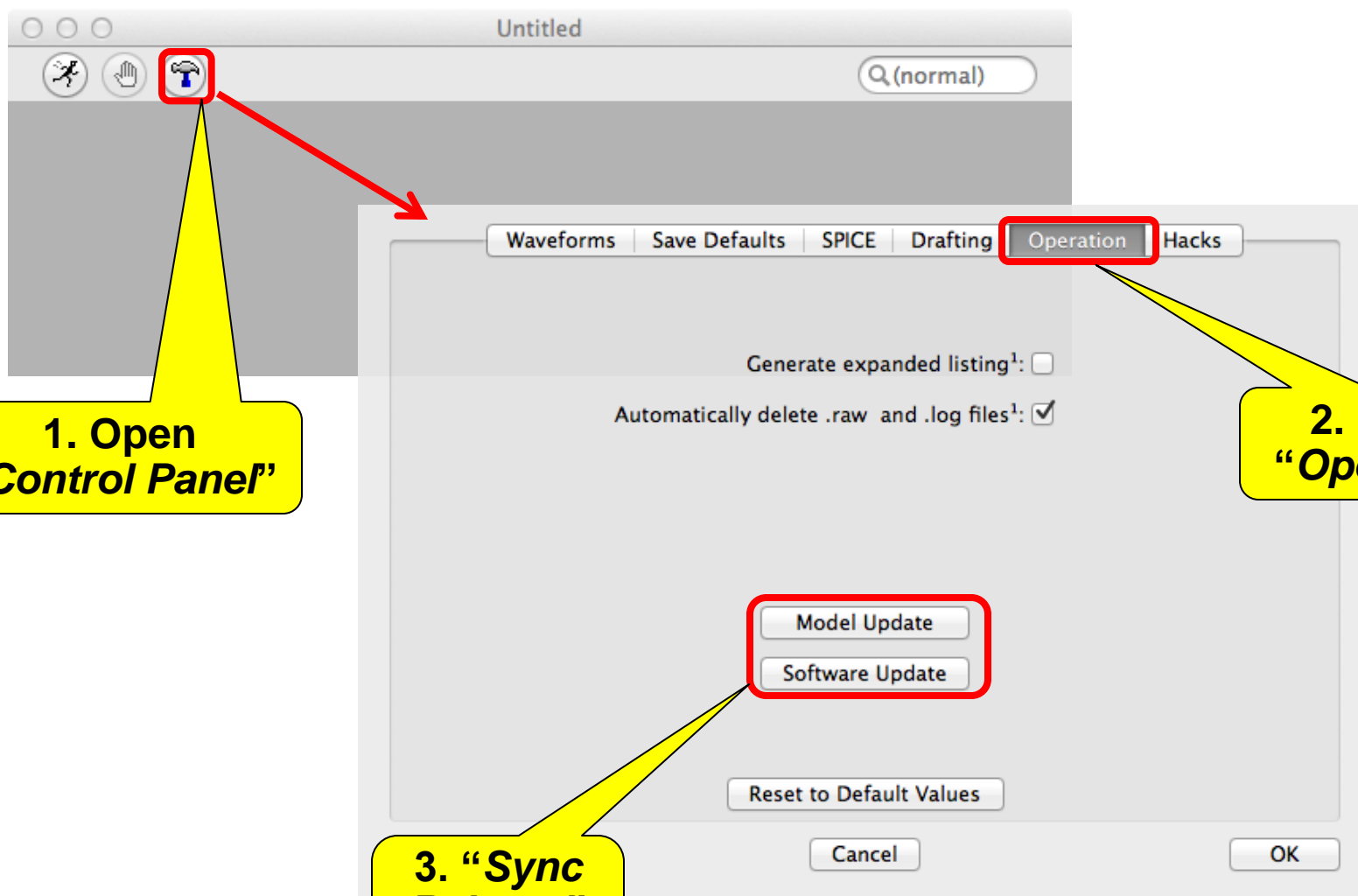
- ❖ It is important to sync your release of LTspice once a month to get the latest updates
 - ❖ Software updates and bug fixes
 - ❖ Models
 - ❖ Sample circuits and examples

Reminder to Periodically Sync Release (Windows)

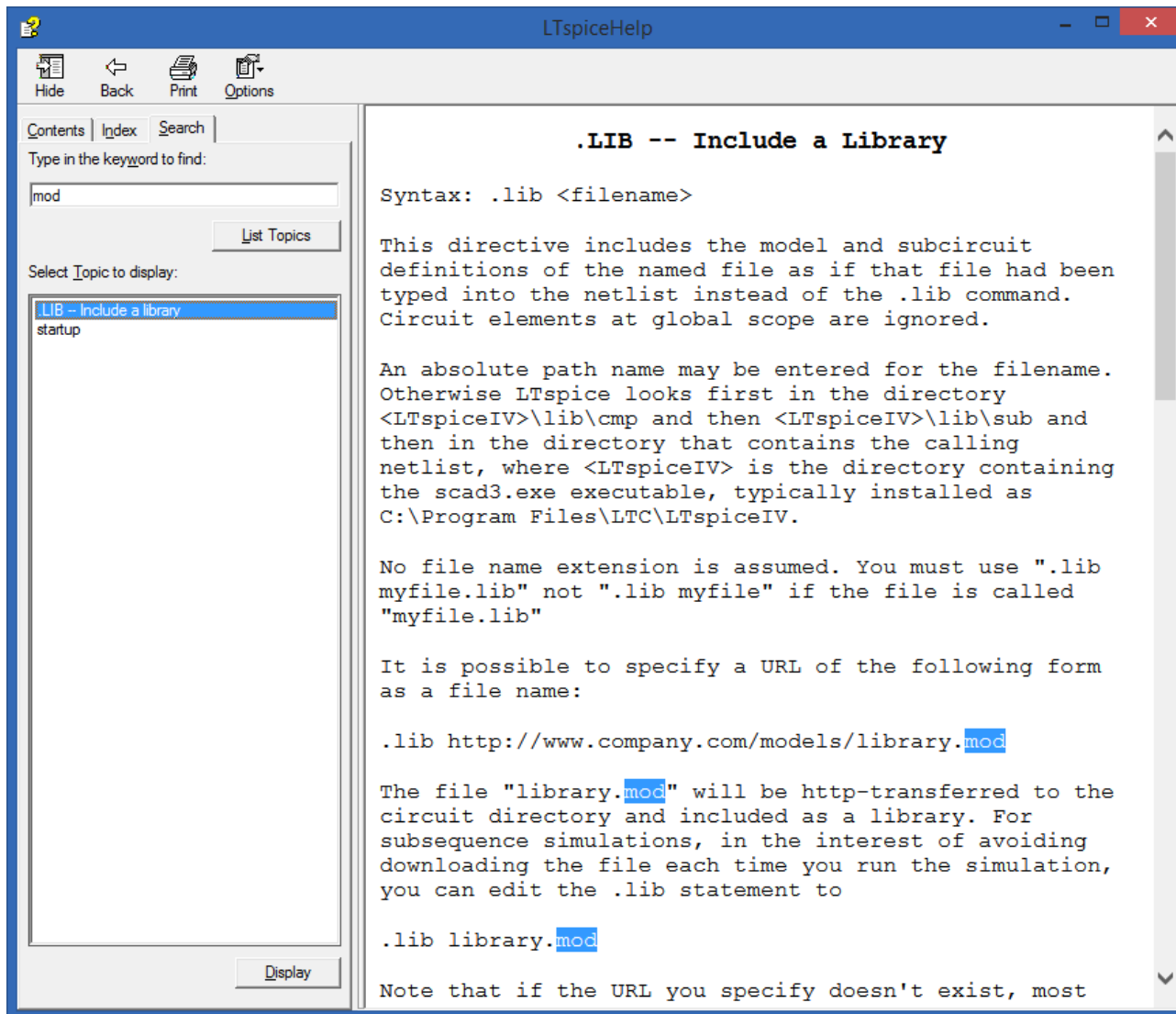
- ❖ Vista, Win7, and Win8 users (any UAC-enabled OS)
 - ❖ You must “Run as administrator” scad.exe or its shortcut even if you are logged in as an administrator



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:\ ... \LTspice\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

LTspice - Yahoo Groups

Search Conversations Search Groups

BROWSE GROUPS

Featured Groups

Business & Finance

Computers & Internet

Cultures & Community

Entertainment & Arts

Family & Home

Games

Government & Politics

Health & Wellness

Hobbies & Crafts

Music

Recreation & Sports

Regional

Religion & Beliefs

Romance & Relationships

Schools & Education

Science

Terms

Privacy

Guidelines

Feedback

Help

Blog

LTspice IV

Public Group 46345 members

Conversations Photos Files More

About Group + Join Group

Group Description

Dedicated to the exchange of information about LTspice. LTspice IV (formerly SwitcherCAD III) is a complete and fully functional SPICE program that is available free of charge from Linear Technology.
(download here: <http://www.linear.com/designtools/software/>)

Before using the software, please read the **License Agreement/Disclaimer** found in the introductory LTspice Help pages.

Please report bugs directly to LTC. Find the address in the **Help About** dialog box (Menu command Help -> About LTspice IV). Include any relevant files and a note on how to duplicate the problem.

For general questions, please read the program's **Help utility**, the group **FAQ** (<http://groups.yahoo.com/group/LTspice/files/%20FAQ>), browse the group's **Conversations -> Messages**, or try a **Search** (top of this webpage) of previous posts before adding your question. This archive holds more than a decade of answered questions.

If you post questions about a specific circuit, it is most helpful to also upload the

<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

The screenshot displays a Windows File Explorer window with the address bar showing the path: **Computer > BOOTCAMP (C:) > Program Files (x86) > LTC > LTspiceIV**. This path is highlighted with a red box. A callout box points to this path with the text: **\Example\Educational subdirectory**.

The left pane shows the folder structure, with **LTspiceIV** expanded to show subfolders like **examples**, **Educational**, **jigs**, **lib**, **PLLWizard**, **Mentor Graphics**, and **Microsoft Analysis Services**. The **Educational** folder is selected.

The right pane lists files in the **Educational** subdirectory, including **phaseshift.asc**, **phaseshift2.asc**, **phono.asc**, **Pierce.asc**, **PLL.asc**, **PLL2.asc**, **relax.asc**, **ring.wav**, **Royer.asc**, **SampleAndHold.asc**, and **S-naram.asc**. A red arrow points from a callout box to the **relax.asc** file. The callout box contains the text: **Design examples demonstrating LTspice capabilities**.

An inset window shows the LTspice schematic for **Transformer.asc**. The title is **A transformer with two windings, 1 to 3 turns winding ratio**. The circuit includes a voltage source **V1** connected to a resistor **R1** (value 10), which is connected to the primary winding **L1** (value 100μ). The secondary winding **L2** (value 900μ) is connected to ground. The schematic is titled **Transformer.asc** and includes the text: **This example schematic is supplied for informational/educational purposes only.**

Other Resources – LTspice Videos

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

Linear Technology Video Channel – LTspice

video.linear.com/LTspice

Linear Technology

Search

中文网站 日本サイト QUALITY CAREERS CONTACT MyLinear

PRODUCTS SOLUTIONS DESIGN SUPPORT PURCHASE COMPANY

> Software & Simulation > LTspice

DOCUMENTATION

Video

- LTspiceIV Overview

LTspice

- LTC3565 Demo Circuit - 1.25A, 4MHz, Synchronous Step-Down Converter (Li-Ion to 1.8V @ 1.25A)
- LTspice Getting Started Guide

Application

LTspice

Matching Videos

The Following Videos match the search criteria selected above.

LTspice IV: Overview

LTspice IV: Schematic Editor

LTspice IV: Waveform Viewer

LTspice IV: Overview

« Prev Next »

LTspiceIV Overview

with Gabino Alonso, Strategic Marketing

Note: Additional Topics are Listed in the Matching Videos (See Left Panel) or at video.linear.com/LTspice.

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.

This video provides an overview of the advantages of using LTspiceIV in an analog design. Topics include the benefits of using LTspice, how easy it is to get started and where to go to download LTspice.

MYLINEAR LOGIN CART SHARE

LINEAR TECHNOLOGY

NOW PART OF ANALOG DEVICES

Click, Relax & Enjoy !

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

Other Resources - LTwiki

LTwiki

The screenshot shows the LTwiki website in a web browser. The browser's address bar displays "ltwiki.org". The page title is "LTwiki - Wiki for LTspice". The main content area is titled "Main Page" and includes a welcome message, a list of frequently asked questions for beginners, and a list of links to various resources. The left sidebar contains navigation, search, and toolbox sections.

Navigation:

- Main page
- Community portal
- Current events
- Recent changes
- Random page
- Help

Search:

Search

Toolbox:

- What links here
- Related changes
- Special pages
- Printable version
- Permanent link

Main Page Content:

Welcome to LT Wiki!

LTwiki is for [LTspice](#), SPICE, and Electronics help. You'll find unique material from beginner's tips to undocumented LTspice features! This site has no affiliation with the [Linear Technology Corporation](#).

Contributors Welcome! Just [create an account](#) first. This prevents anonymous spammers from ruining the wiki.

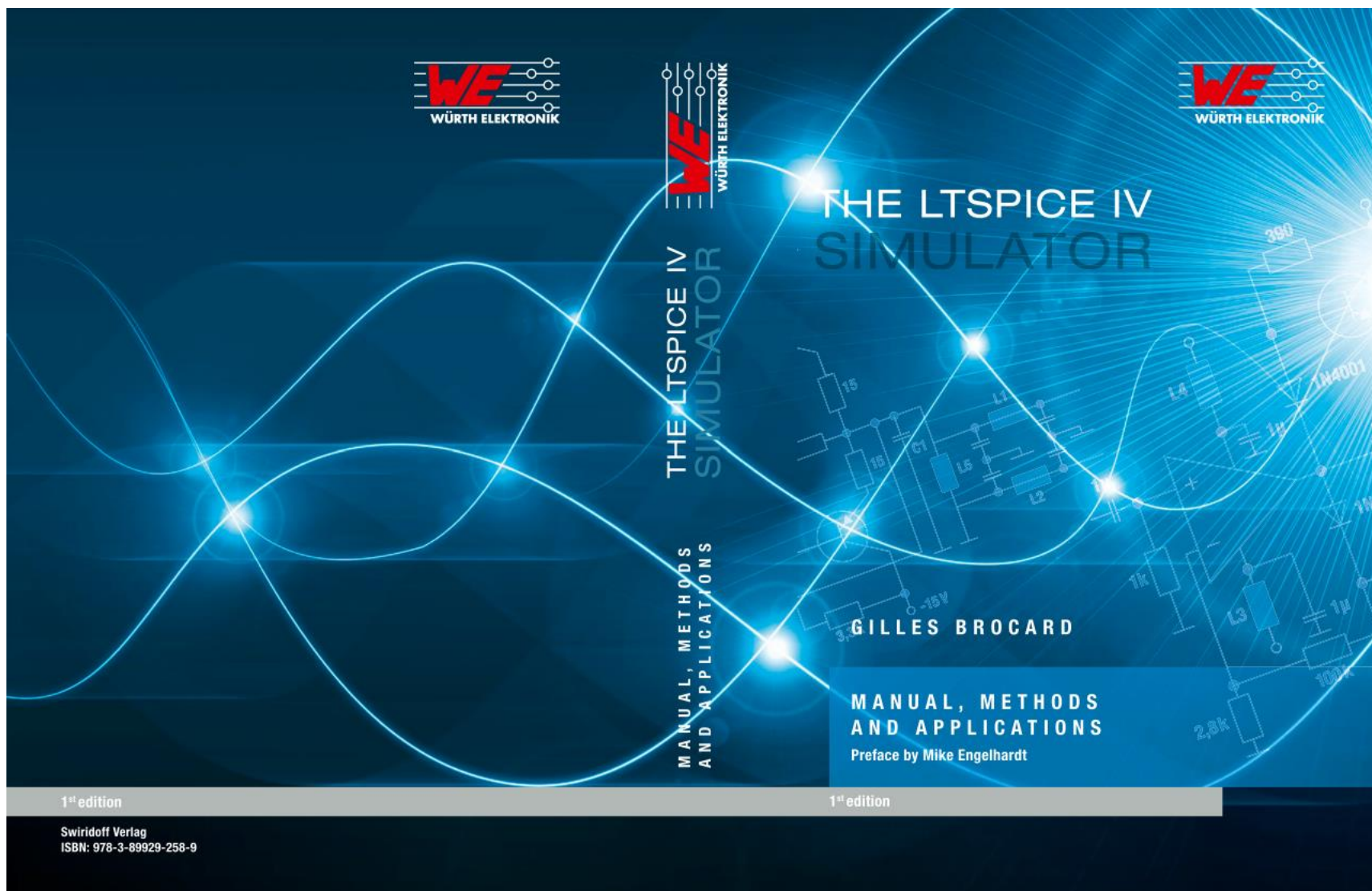
Most frequently asked questions for beginners

- [Adding a permanent component to LTspice](#)
- [Adventures with Analog](#)
- [B sources \(complete reference\)](#)
- [B sources \(common examples\)](#)
- [Components Library](#)
- [Control Panel](#)
- [Convergence problems?](#)
- [LTspice Annotated and Expanded Help*](#)
- [LTspice Hot Keys](#)
- [LTspice Tools and Applications](#)
- [Simulation Command](#)
- [SPICE and LTspice Courseware and Tutorials](#)
- [SPICE Model Links](#)
- [SPICE Application Notes and White Papers](#)
- [Tutorials relevant to Design and Modelling](#)
- [Transformers](#)
- [Undocumented LTspice](#)
- [LTspice Library API](#)

**based on original LTspice help (chm) file ©Linear Technology Corporation used by permission*

This page was last modified on 9 December 2012, at 03:03. This page has been accessed 103,951 times. [Privacy policy](#) [About LTwiki - Wiki for LTspice](#) [Disclaimers](#)

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!

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