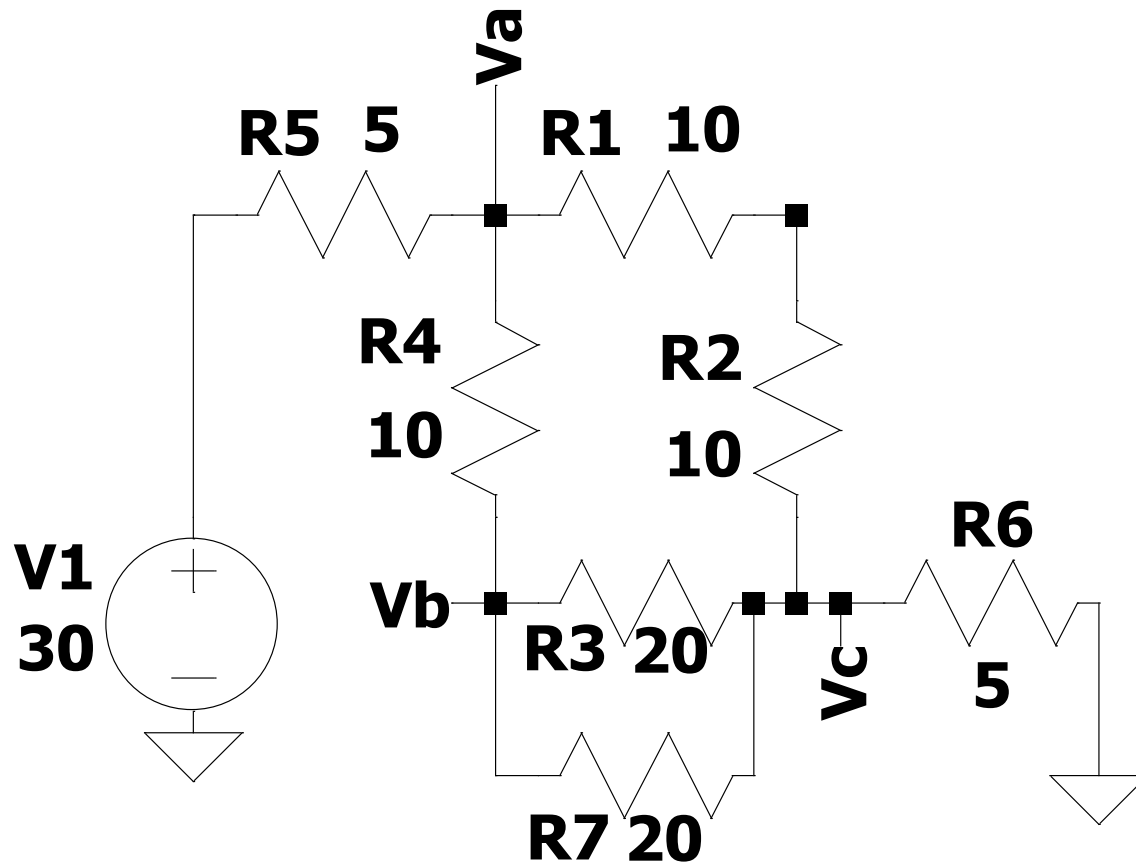


LTspice and SPICE Simulation

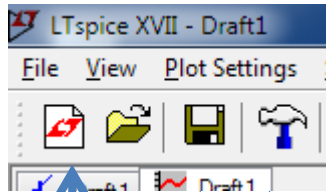
Simulation of Circuits

What is the Voltage at Vb?

SPICE is one of the alternatives to solving the problem by hand.



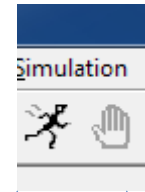
Menus: Left Side of Toolbar



Control Panel

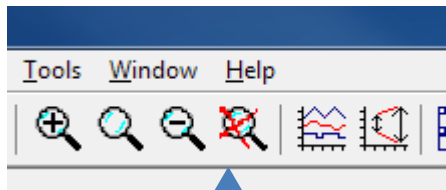
Save (Ctrl+S)

New File (Ctrl+N)



HALT Simulation

RUN Simulation



Auto-Fit Graph

Zoom Function(s)

Menus: Right Side of Toolbar



Command (Hotkey)

Spice **Directive** (S)

Mirror (Ctrl+E)

Rotate (Ctrl+R)

Undo (F9)/**Redo** (Shift+F9)

Move [Large] **Drag** [Small]

Generic Component (F2)

Diode (D)

Inductor (L)

Capacitor (C)

Resistor (R)

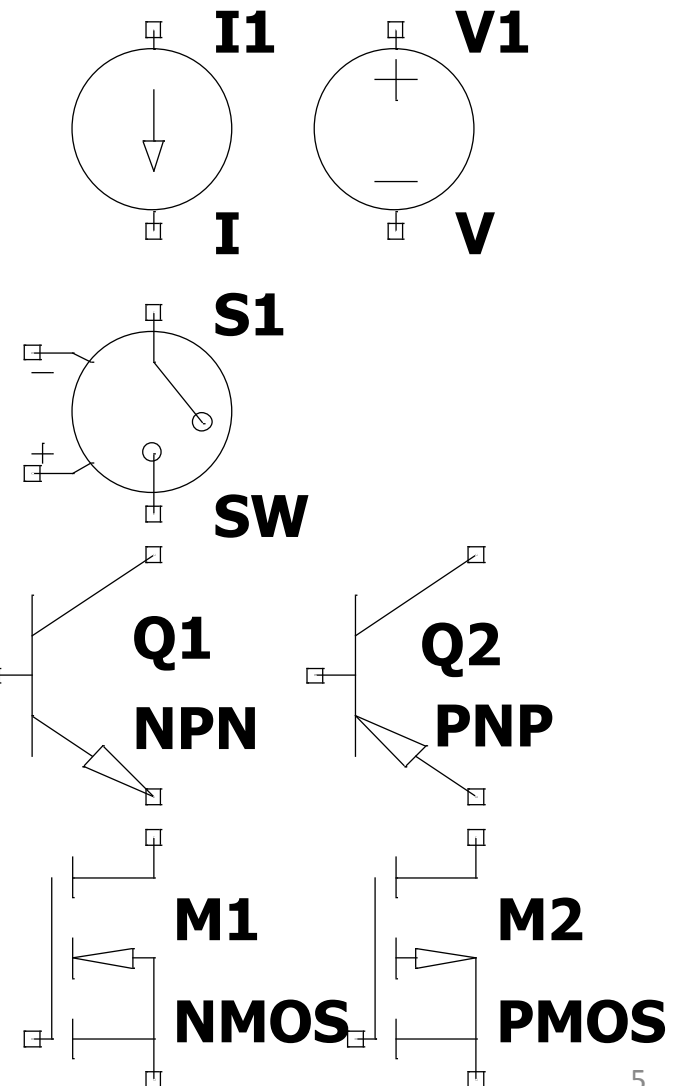
Label Net (F4)

Ground (G)

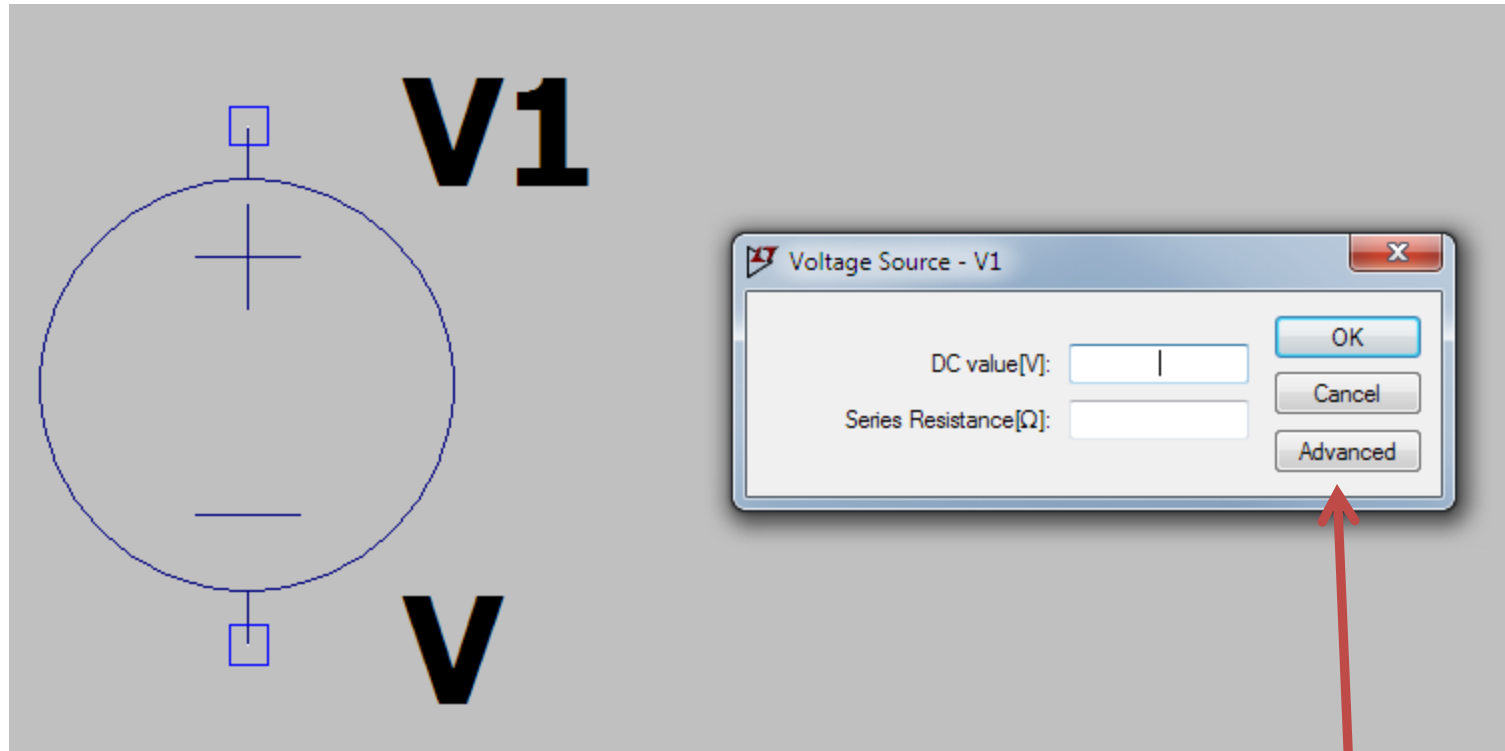
Draw Wire (F3)

Components: Common Names

- Sources
 - Current Source “current”
 - Voltage Source “voltage”
- Switches
 - Switch “sw”
- Bipolar Junction Transistors (BJT's)
 - NPN BJT “npn”
 - PNP BJT “pnp”
- Metal-Oxide Semiconductor Field Effect Transistors (MOSFET's)
 - N-Channel MOSFET “nmos”
 - P-Channel MOSFET “pmos”

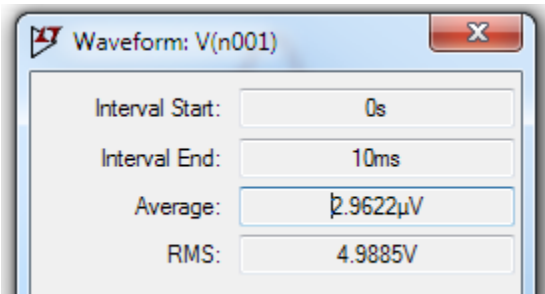


Components: Voltage Sources

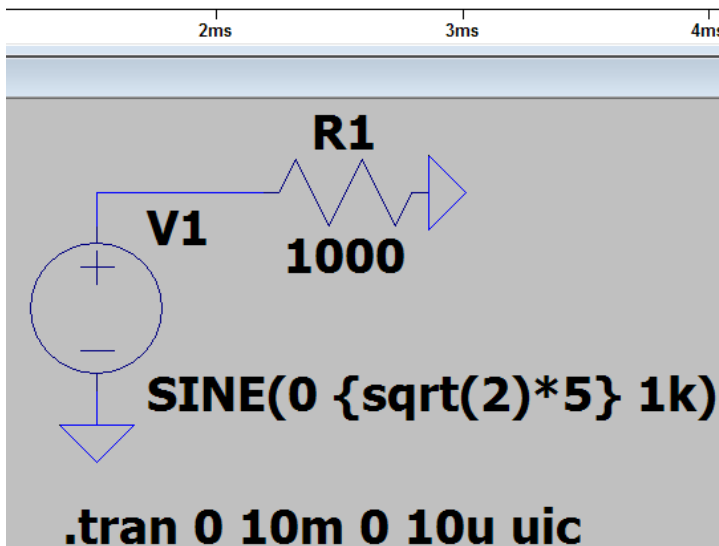
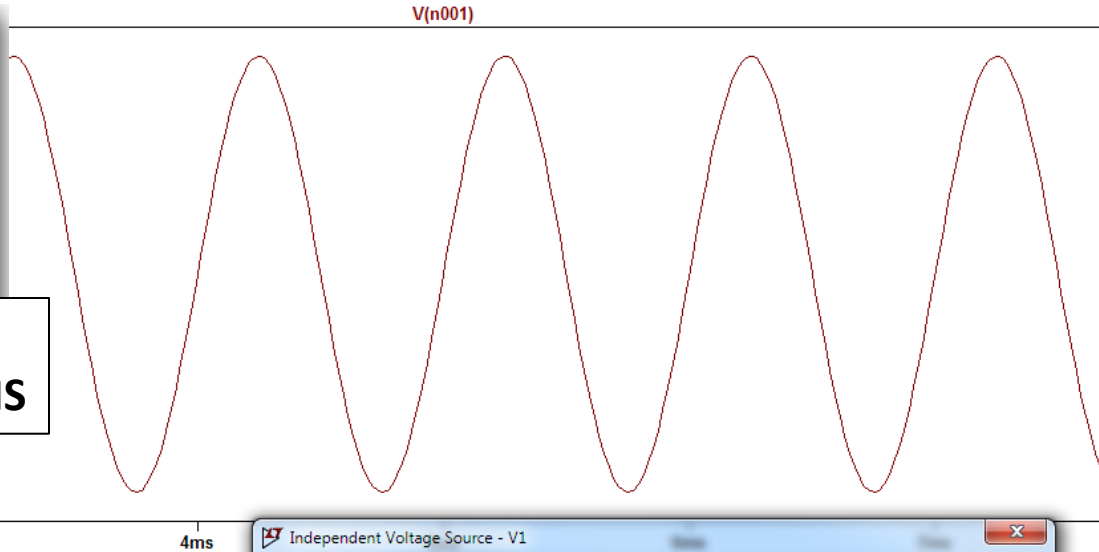


Interesting Settings Under **“Advanced”**

Voltage Sources: Sine Waves



(Ctrl+Click) Trace Name to show plot **Average** and **RMS**



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: ☒

DC offset[V]: 0

Amplitude[V]: {sqrt(2)*5}

Freq[Hz]: 1k

Tdelay[s]:

Theta[1/s]:

Phi[deg]:

Ncycles:

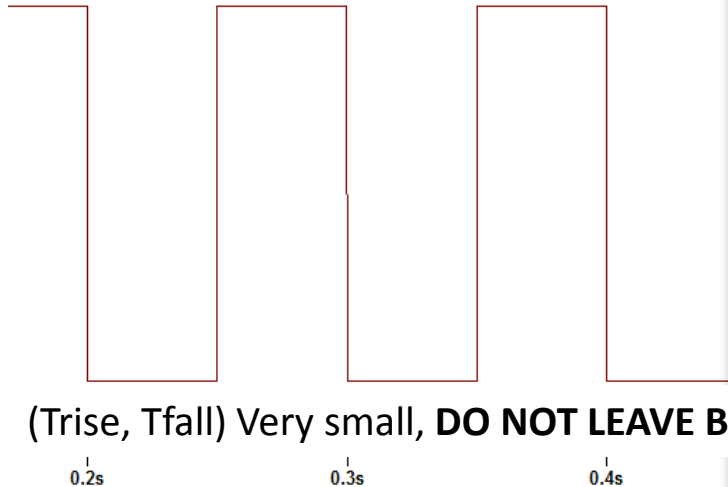
Additional PWL Points

Make this information visible on schematic: ☒

Cancel OK

Amplitude is **PEAK Voltage**
Peak Voltage = 1.414*RMS V
Use {...} to enclose expressions

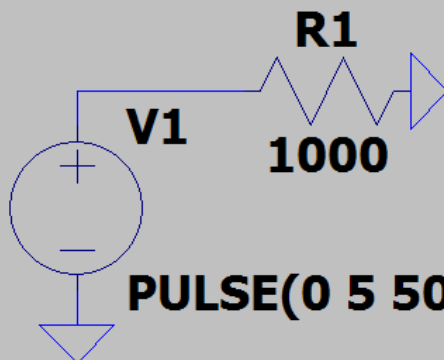
Voltage Sources: Square Wave



(Trise, Tfall) Very small, **DO NOT LEAVE BLANK**

The dialog box for configuring an Independent Voltage Source (V1). It shows the following settings:

- Functions:** ☒ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- 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: ☒)
- Parameters:** Vinitial[V]: 0, Von[V]: 5, Tdelay[s]: 50m, Trise[s]: 10n, Tfall[s]: 10n, Ton[s]: 50m, Tperiod[s]: 100m, Ncycles:
- Buttons:** Additional PWL Points, Make this information visible on schematic: ☒, Cancel, OK

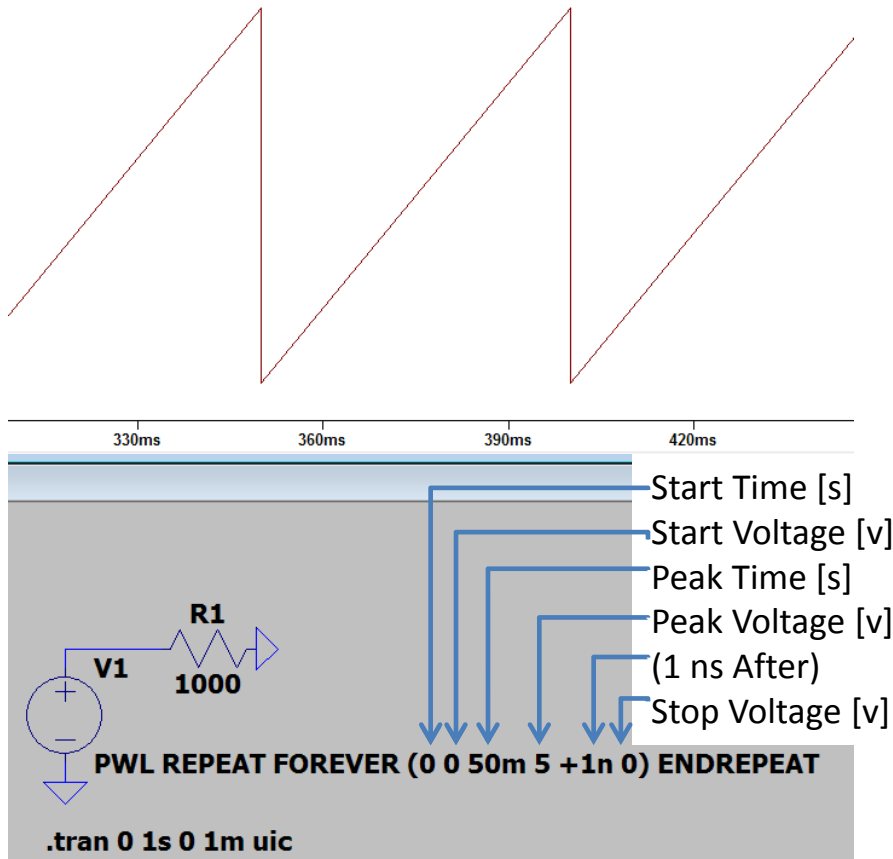


PULSE(0 5 50m 10n 10n 50m 100m)

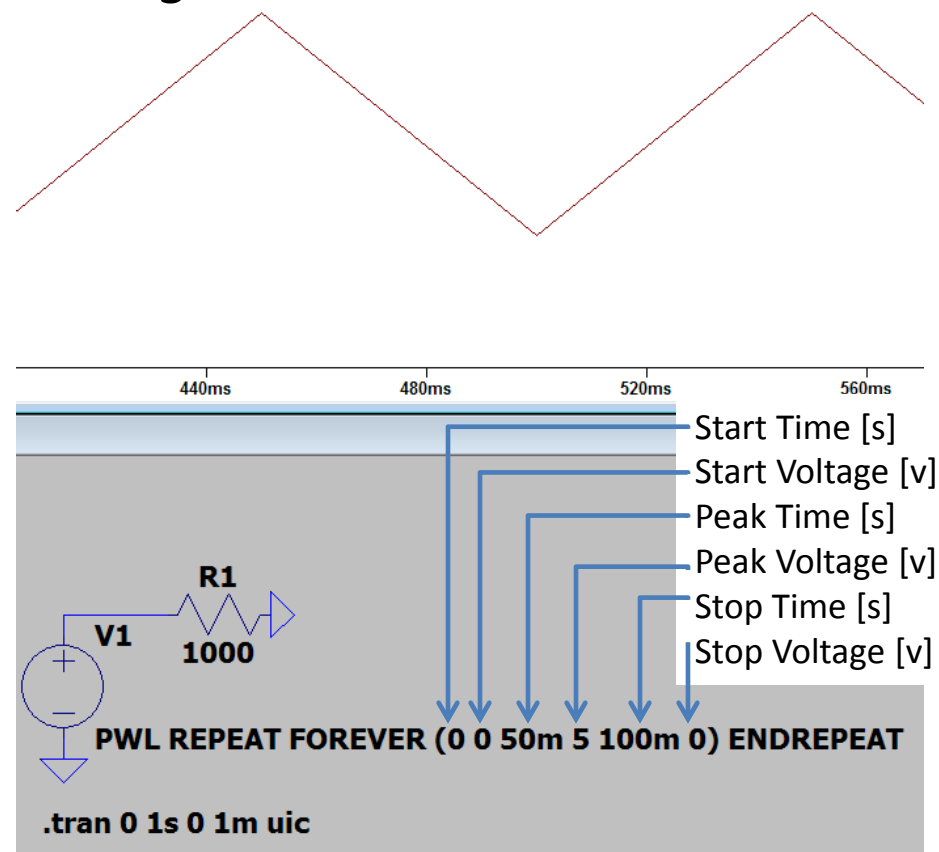
.tran 0 1s 0 1m uic

Voltage Sources: Triangular Waves

Sawtooth Wave

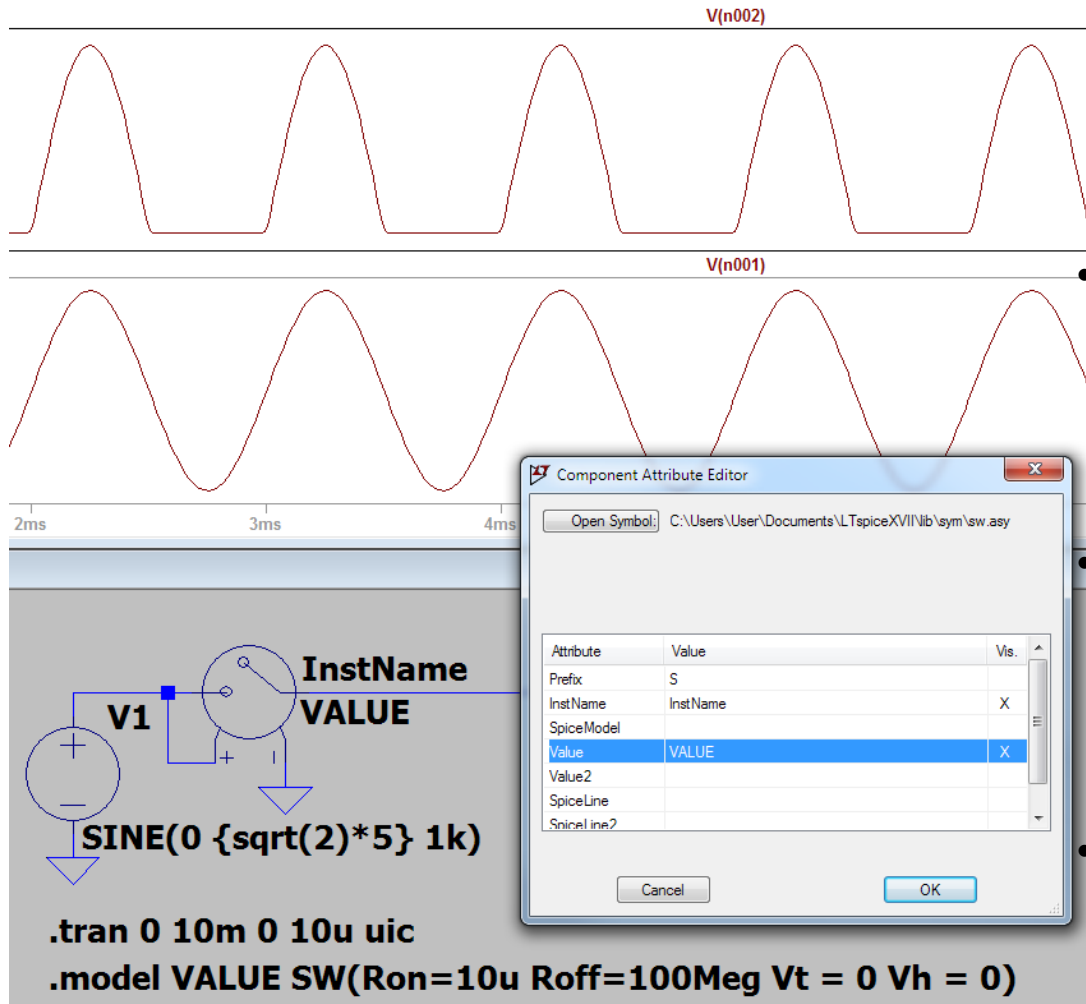


Triangular Wave



- <http://www.analog.com/en/technical-articles/ltspice-generating-triangular-sawtooth-waveforms.html>

Components: Switches



Spice Directive is **REQUIRED**

- “.model VALUE SW()”
 - Simplest Switch
 - “VALUE” is the Value (Name) of the Switch

“SW(...)” Additional Parameters:

- On Resistance: “Ron=”
- Off Resistance: “Roff=”
- Trigger Voltage: “Vt=”
- Hysteresis Voltage: “Vh=”

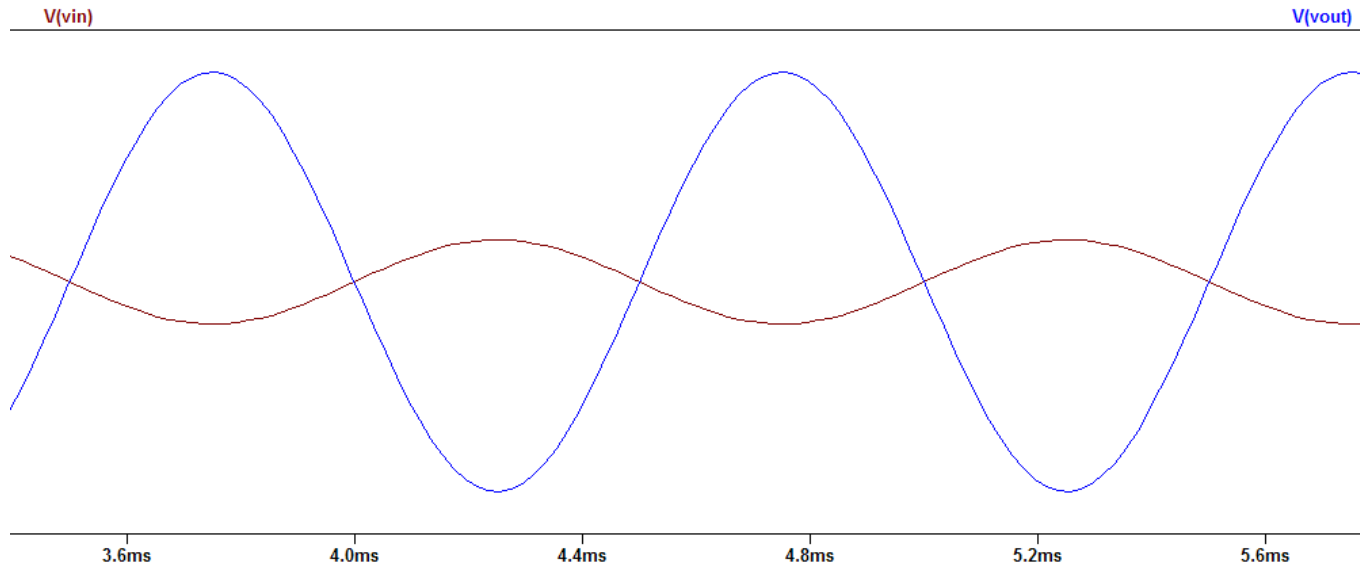
Component Attribute Editor

- (Right-Click) the Component

.model VALUE SW()

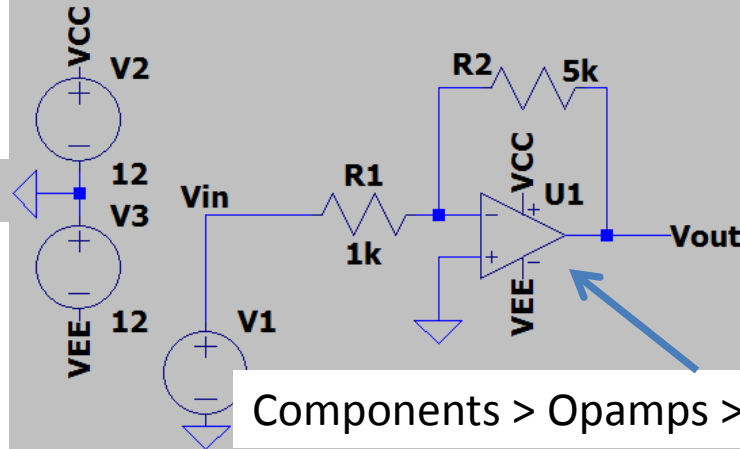
<http://www.analog.com/en/technical-articles/ltpspiceiv-voltage-controlled-switches.html>

Components: Op-Amps



Net Names can be used to connect terminals without wires.

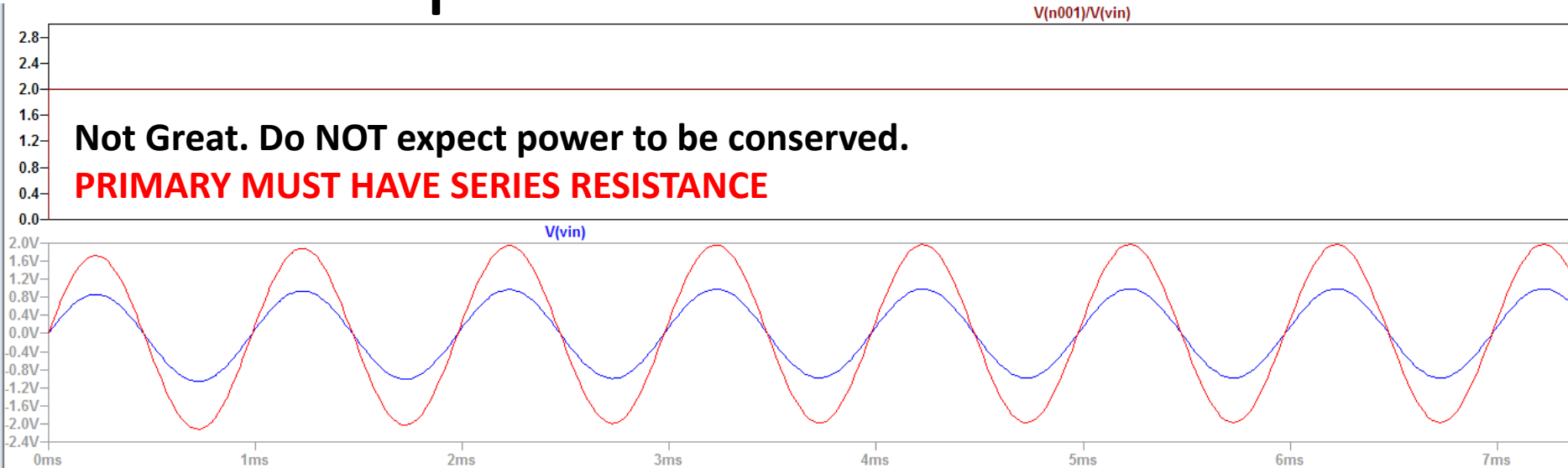
Voltage Sources can be cascaded in series. This particular arrangement is a common for $\pm V$.



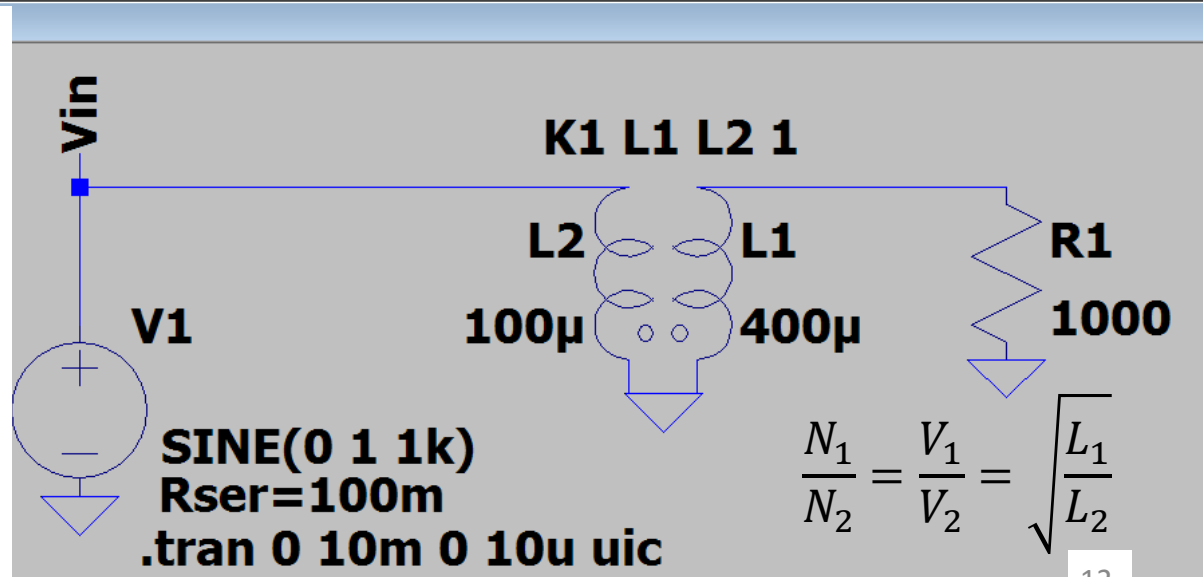
Components > Opamps > **UniversalOpamp2**

.tran 0 10m 0 10u uic

Components: Transformers

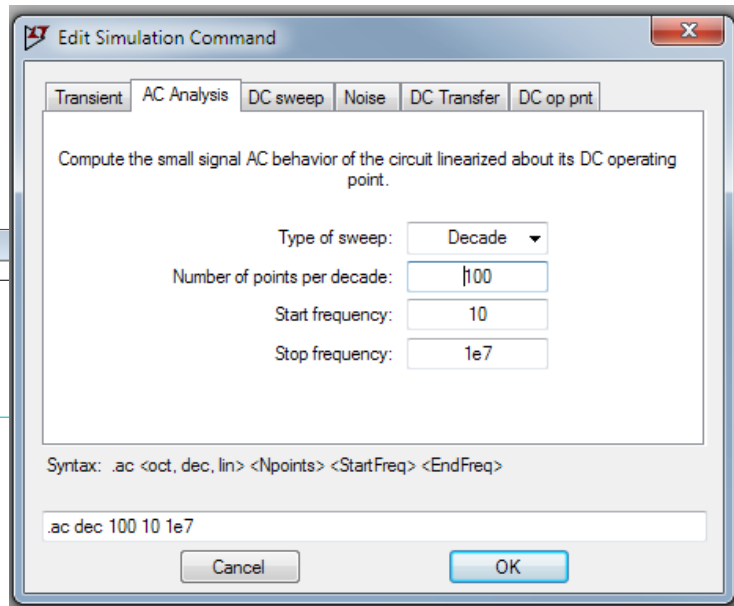


- Inductors are coupled using a spice directive
 - “K# L# L# Coupling”
 - Multiple Inductors can be mutually coupled by listing them before the coupling coefficient.
- Coupling ranges from 0 (none) to 1 (complete)

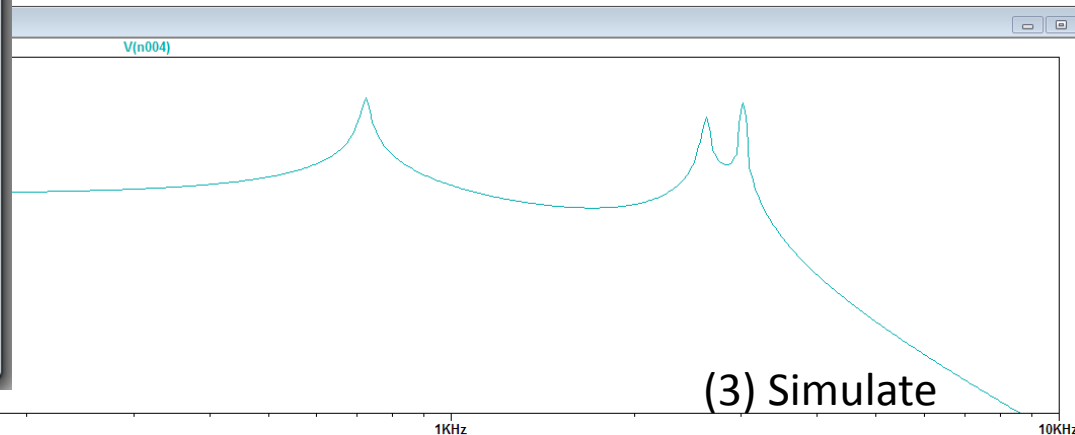


Simulation: AC Analysis

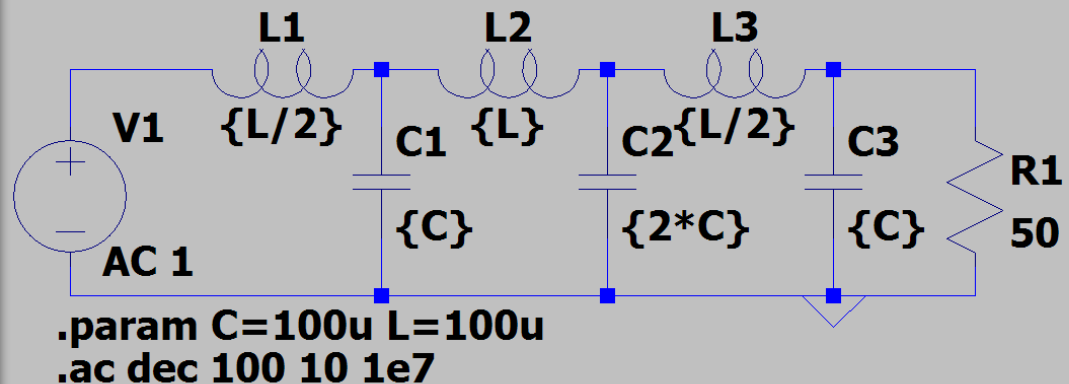
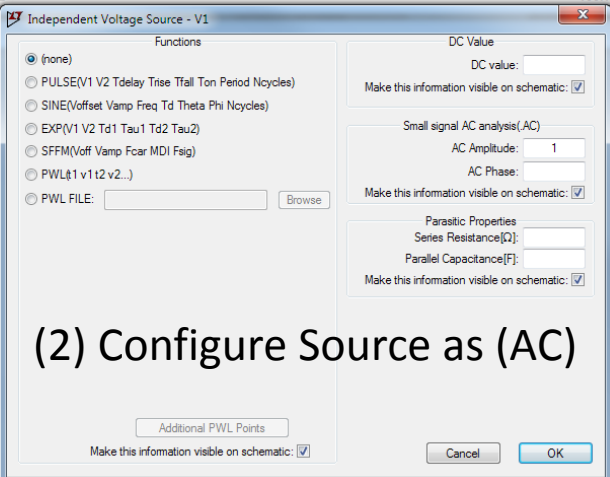
(1) Configure Simulation as “AC Analysis”



(3) Simulate



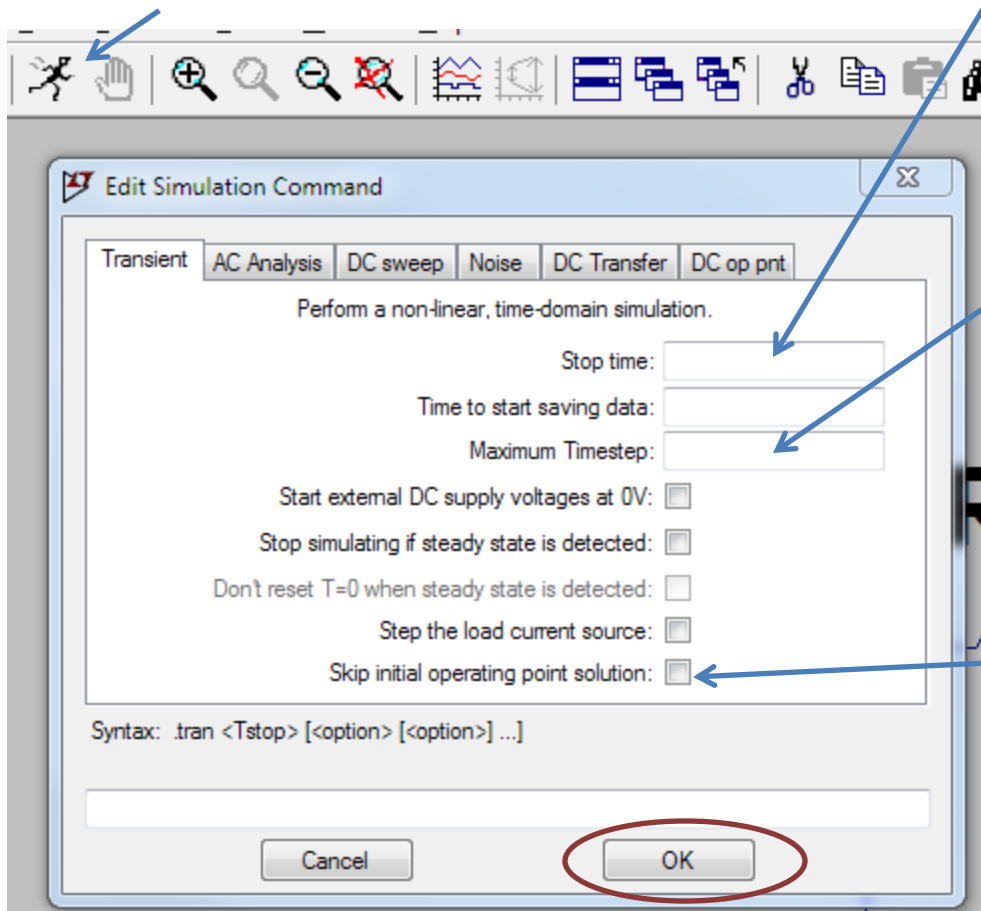
(2) Configure Source as (AC)



Simulation: Transient Analysis

Simulate the Presented Assuming Sources Were Just Connected

(1) Click Here



- When to Stop [S]: (Keep this small)
 - A few periods/few time constants
 - Shorter will capture transients
 - Longer takes longer to simulate
 - (100's ms) likely sufficient
- Time between successive calculations
 - Smaller times are more accurate
 - Much smaller than resolution
 - (1 us) good to 100 kHz
 - (100 ns) good to 1 MHz
 - (10 ns) good to 10 MHz
 - ...
- Skip initial operating point solution
 - Translated: "Don't guess the steady-state solution before simulating"
 - **Highly Recommended.**

Simulation: Simple Plotting

The image shows a screenshot of the LTspice XVII software interface. At the top, a plot window displays a voltage waveform over time. The vertical axis (y-axis) ranges from 0V to 16V, and the horizontal axis (x-axis) ranges from 0ms to 100ms. Two dialog boxes are overlaid on the plot:

- Vertical Axis Dialog:** This dialog is titled "Vertical Axis" and contains the following settings:
 - Axis Limits: Top: 16V, Tick: 2V, Bottom: 0V.
 - Logarithmic: ☐ (unchecked).
 - Buttons: OK, Cancel.
- Horizontal Axis Dialog:** This dialog is titled "Horizontal Axis" and contains the following settings:
 - Quantity Plotted: time.
 - Axis Limits: Left: 0s, tick: 10ms, Right: 100ms.
 - Logarithmic: ☐ (unchecked).
 - Buttons: Cancel, OK.

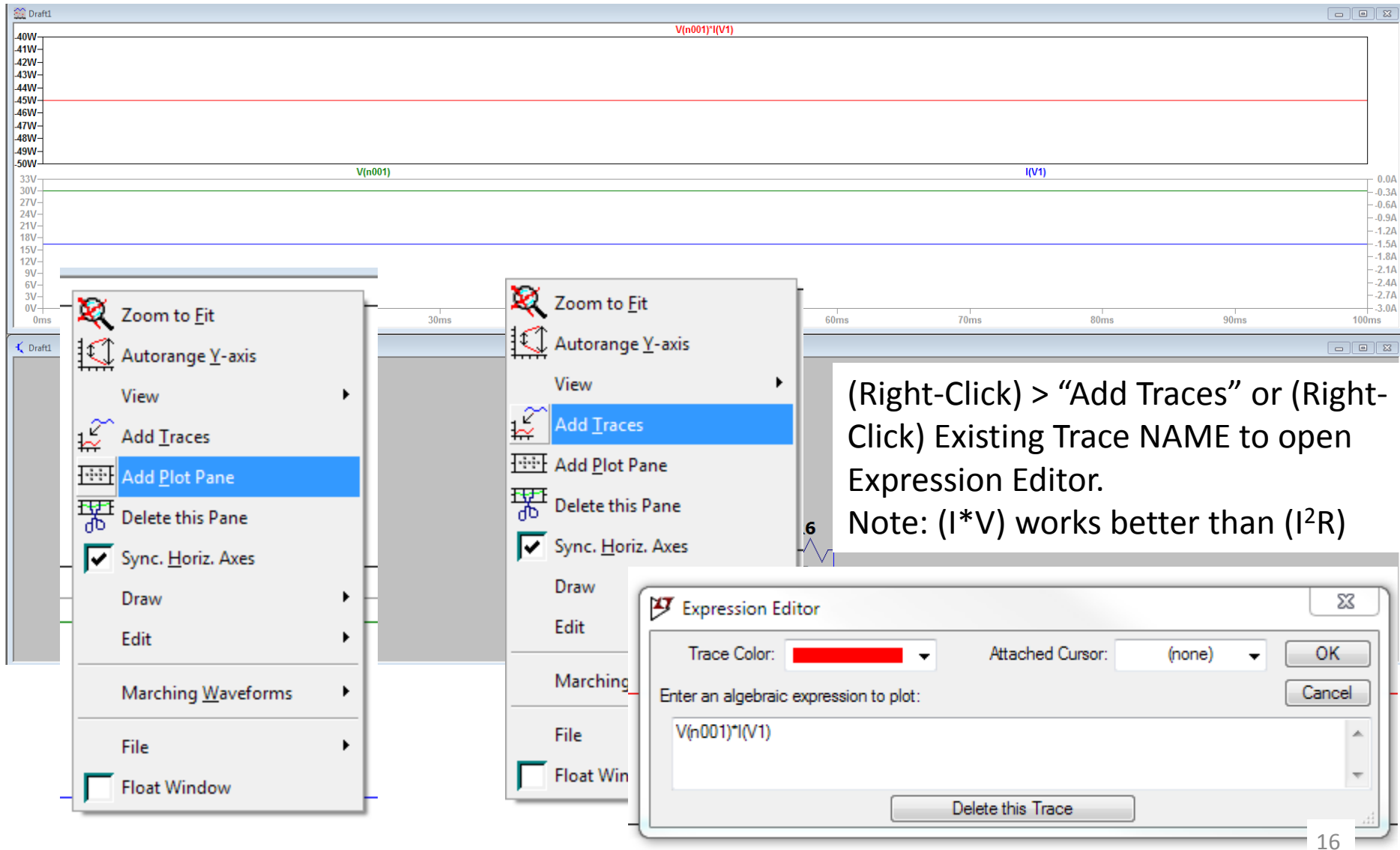
Below the plot, a circuit diagram is shown. It consists of a DC voltage source V1 (30V) connected in series with a resistor R5 (5Ω). This is followed by a parallel combination of two branches. The first branch contains a resistor R4 (10Ω) in series with a resistor R1 (10Ω). The second branch contains a resistor R3 (20Ω) in series with a resistor R2 (10Ω). After the parallel combination, there is a resistor R6 (5Ω) in series with a resistor R7 (20Ω). The circuit is connected to ground. The simulation command at the bottom is ".tran 0 100m 0 100n uic".

A: Click on Wires to add Voltages →

B: Click on Components to add Currents (*Note the Current Arrow*) →

NOTE: Double Clicking Current/Wire Clears all OTHER traces.

Simulation: Advanced Plotting



Simulation: Known Issues

- Discontinuities
 - Discontinuous functions and their derivatives tend to break the simulation.
 - E.g. Ideal Diodes, Perfect Switches
 - “.options gmin=1e-10” [1]
 - “.options abstol=1e-10” [1]
 - “.options reltol=0.003” [1]
 - “.options cshunt=1e-15” [1]
 - Magnetic Components
 - Shorting voltage sources into inductors
 - Transformers
 - <http://www.ltwiki.org/index.php?title=Transformers>

Additional Resources

- Analog Devices LTspice Download
 - <http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
- (Unofficial) LTspice Wiki
 - http://www.ltwiki.org/index.php?title=Main_Page
- Great Walk Through(s)
 - <http://denethor.wlu.ca/ltspice/#IIIE>