Introduction to LTspice

Acknowledgment: LTspice material based in part by
Devon Rosner (6.101 TA 2014), Engineer, Linear Technology

SPICE

- Simulation Program with Integrated Circuit Emphasis
- Developed in 1973 by Laurence Nagel at UC Berkeley's Electronics Research Laboratory
- Dependent on user defined device models

Netlists

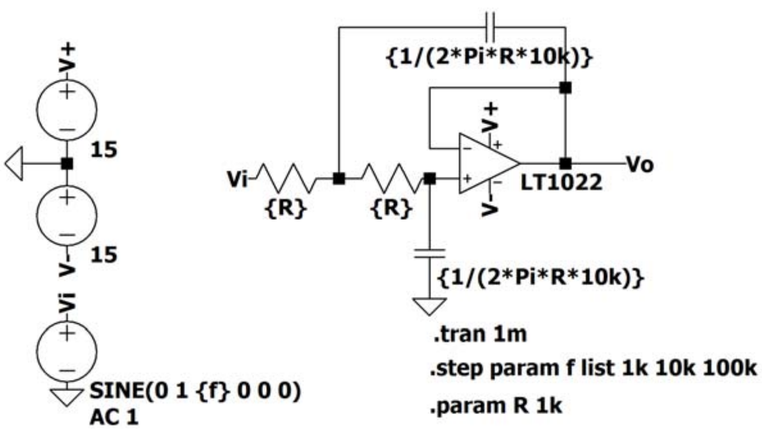
```
* C:\Users\Devon\Desktop\6101\6101SallenKey.asc
XU1 N002 Vo V+ V- Vo LT1022
V1 V+ 0 15
V2 0 V- 15
R1 Vi N001 {R}
R2 N001 N002 {R}
C1 N002 0 {1/(2*Pi*R*10k)}
C2 N001 Vo {1/(2*Pi*R*10k)}
V3 Vi 0 SINE(0 1 {f} 0 0 0) AC 1
.step param f list 1k 10k 100k
.param R 1k
.tran 1m
.lib LTC.lib
.backanno
.end
```

Components

Commands

LTspice

- Developed in 1998 by Mike Engelhardt at Linear Technology Corporation
- GUI, simulator, and schematic -> netlist for SPICE
- **FREE** and comes with tons of models

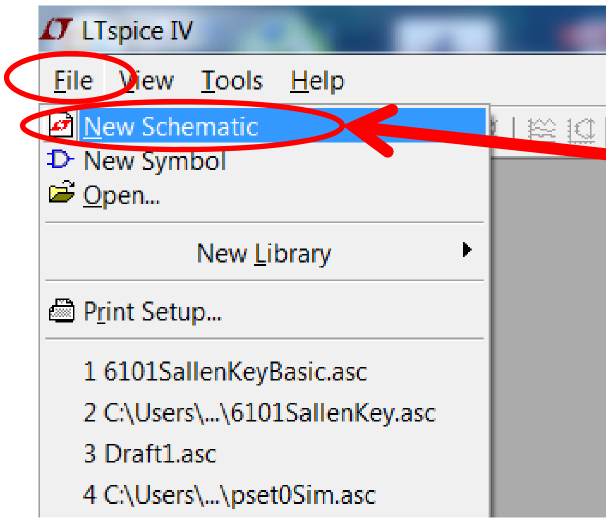



```
* C:\Users\Devon\Desktop\6101\6101SallenKey.asc
XU1 N002 Vo V+ V- Vo LT1022
V1 V+ 0 15
V2 0 V- 15
R1 Vi N001 {R}
R2 N001 N002 {R}
C1 N002 0 {1/(2*Pi*R*10k)}
C2 N001 Vo {1/(2*Pi*R*10k)}
V3 Vi 0 SINE(0 1 {f} 0 0 0) AC 1
.step param f list 1k 10k 100k
.param R 1k
.tran 1m
.lib LTC.lib
.backanno
.end
```

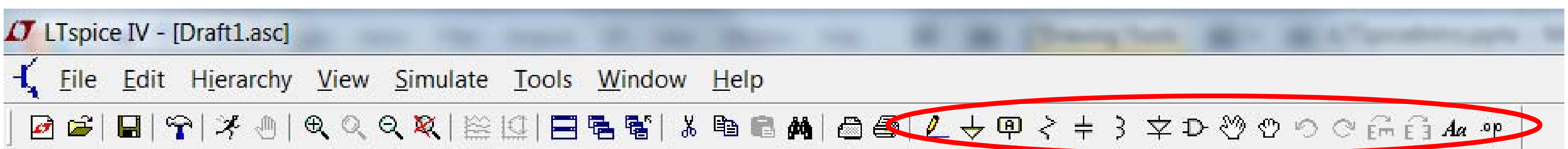
You do this

Ltspice makes this

Getting Started

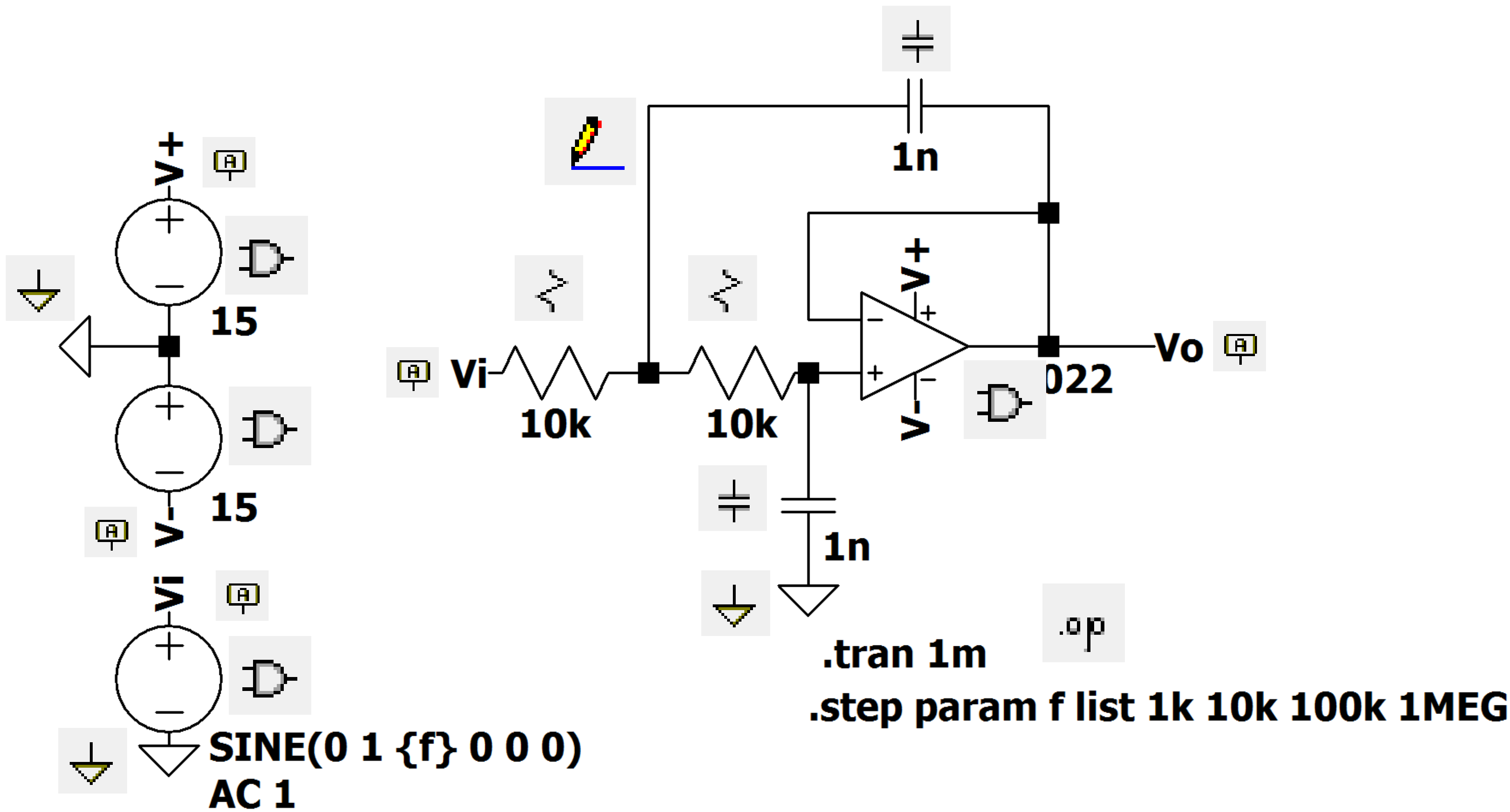


THAT'S IT!



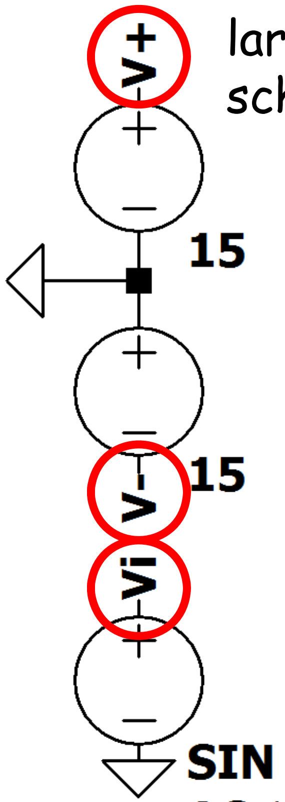
These buttons are where you will live

Component to Menu Item Matchup

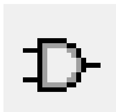


Net Labels

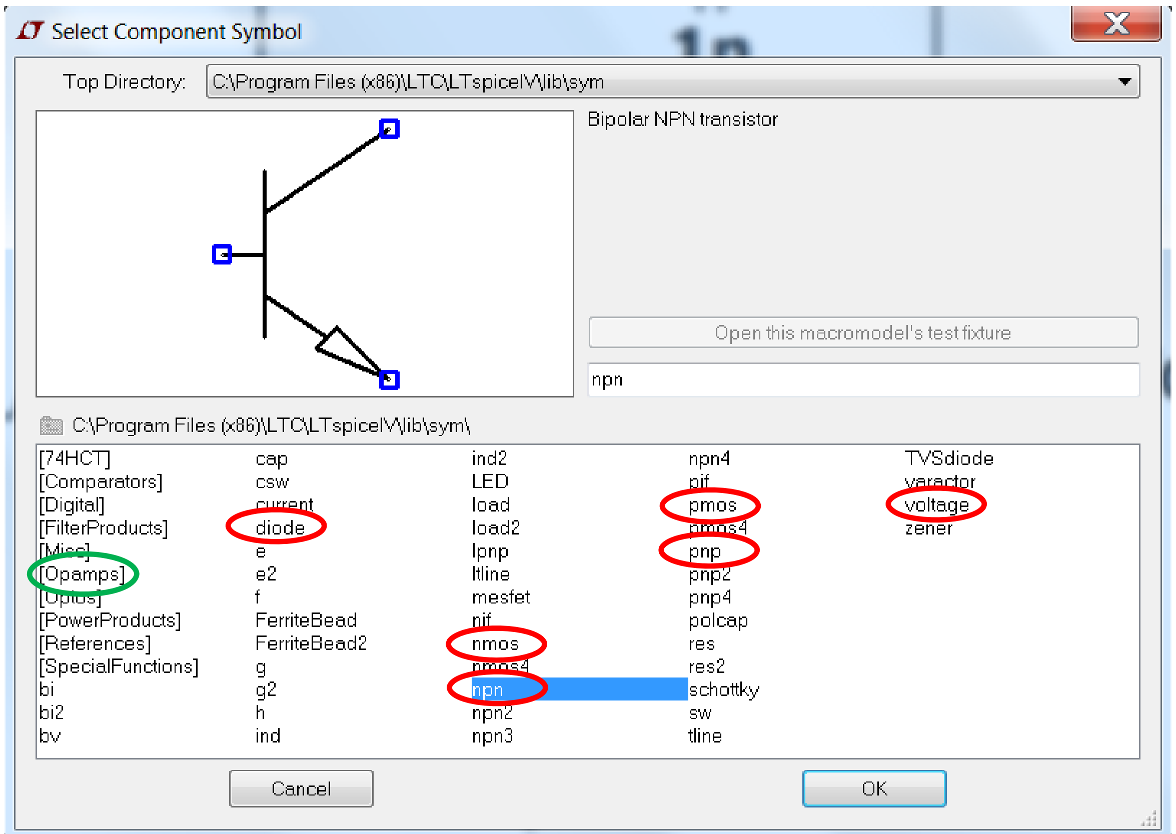
By labeling nets you can avoid a giant mess of wires. *Always* use these for at least your power supplies. When you start making large circuits, your power supplies will provide energy all over your schematic.



Adding Other Components

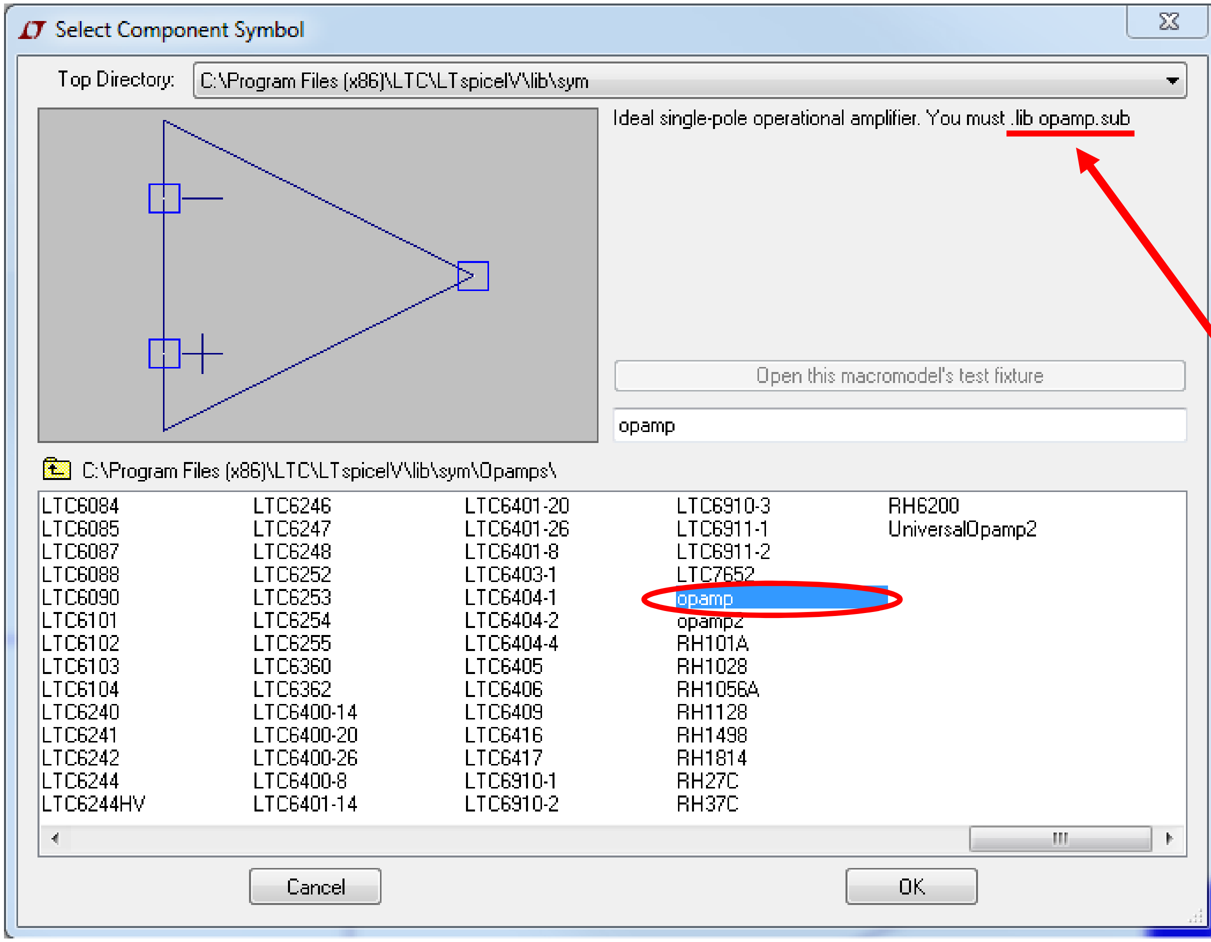


Devices besides basic resistors, capacitors, and inductors are found from this button



Op-Amps

There are no “ideal” op-amps in reality. BUT, there are in LTspice.

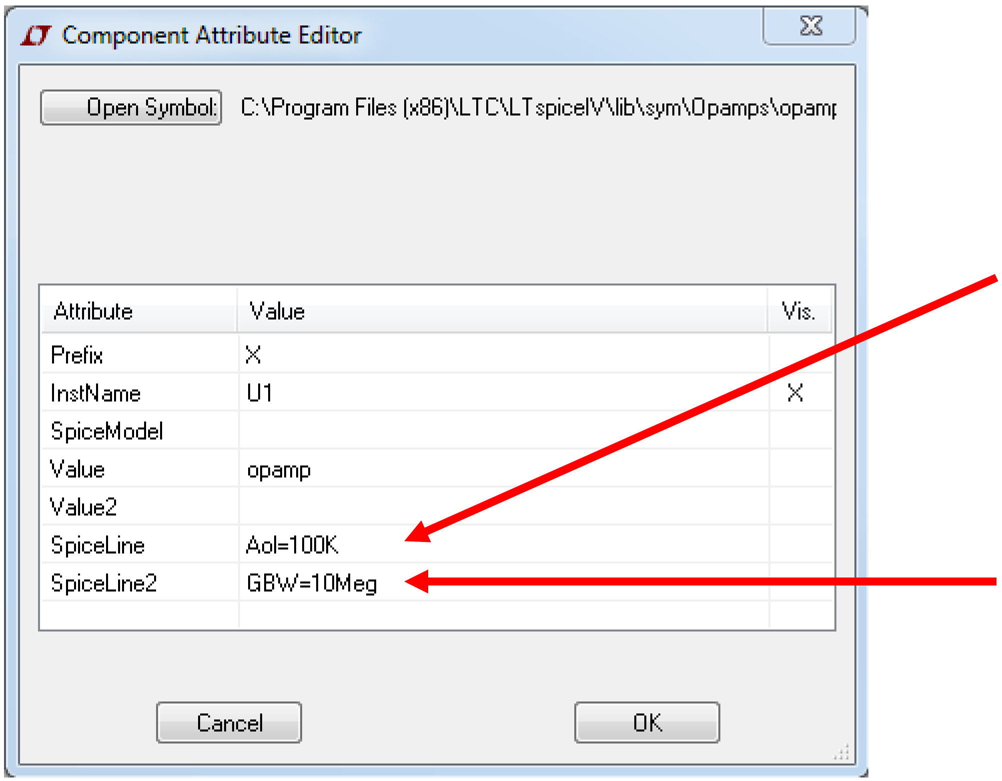


PAY CLOSE ATTENTION TO THE TEXT

You must literally include .lib opamp.sub in your netlist or schematic as a SPICE directive.

Op-Amps

Though listed as “ideal” there are still 2 parameters you can tweak.

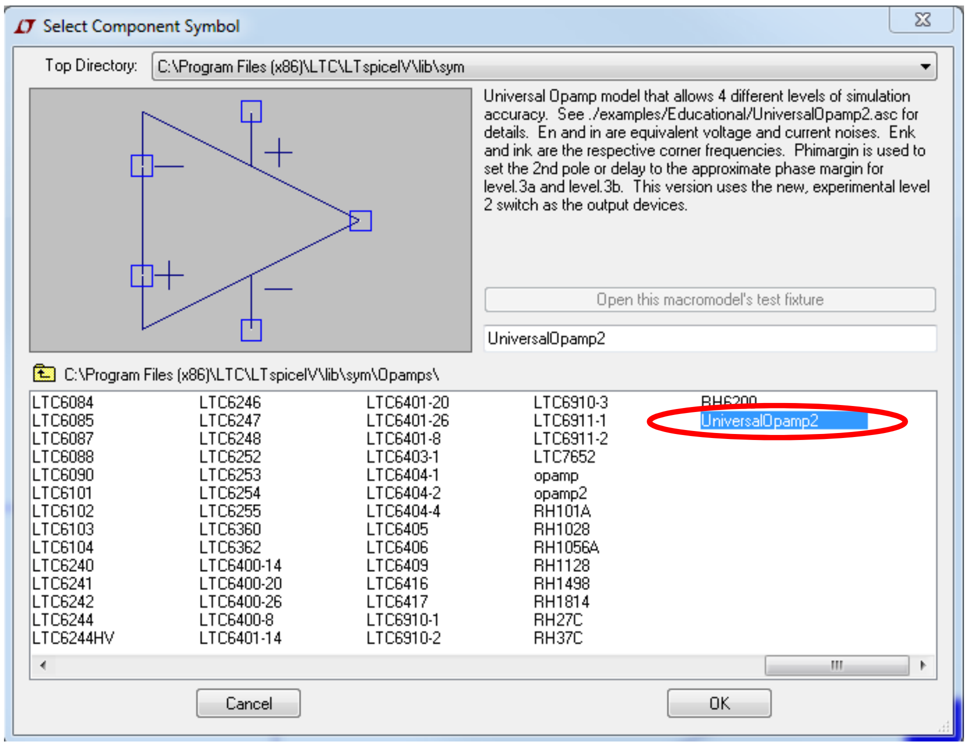


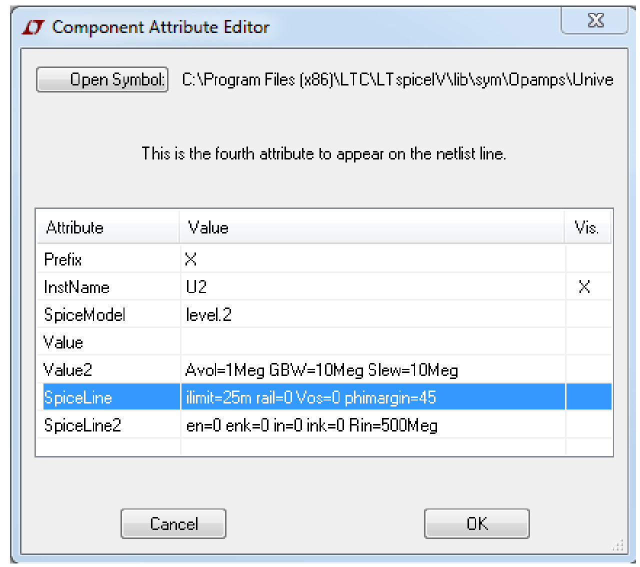
Open Loop Gain: As this number approaches infinity, the Op Amp becomes more “ideal”. Look at some Op Amp data sheets to see some real open loop gains.

Gain Bandwidth: As this number approaches infinity, the Op Amp becomes more “ideal”. To check if this is high enough, multiply your desired Closed Loop Gain by your highest desired output frequency.

Op-Amps

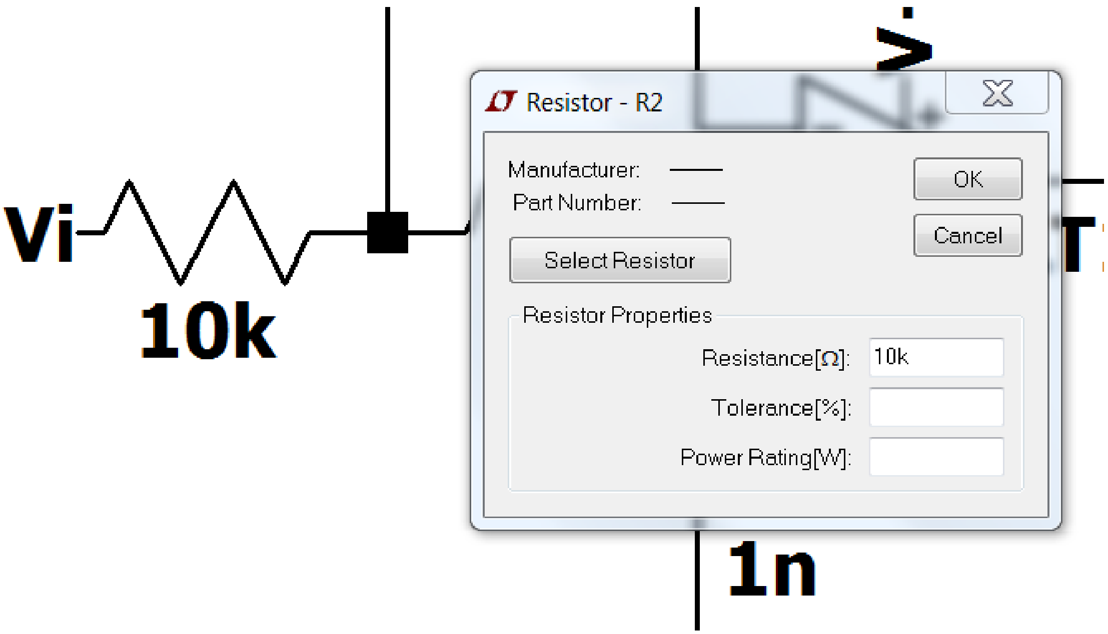
To more accurately model a real Op Amp not available in LTspice, UniversalOpamp2 has many tweakable parameters.





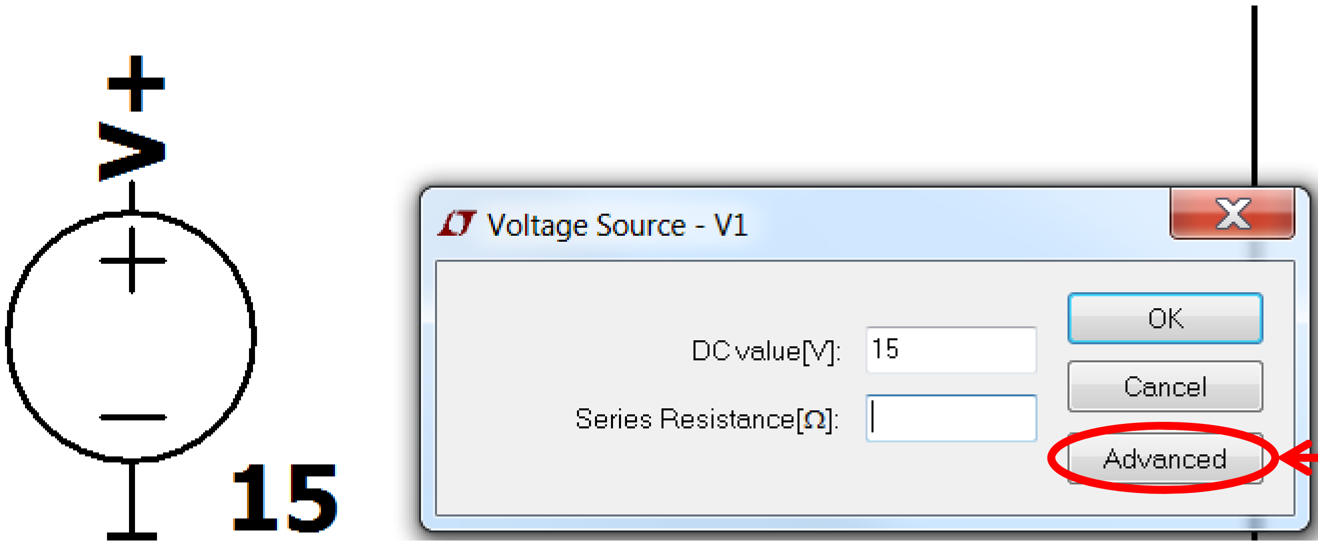
Open loop gain, gain bandwidth, slew rate, current limit, rail-rail voltage, input voltage offset, phase margin, Rin, etc.

Editing Components



Just right click the component

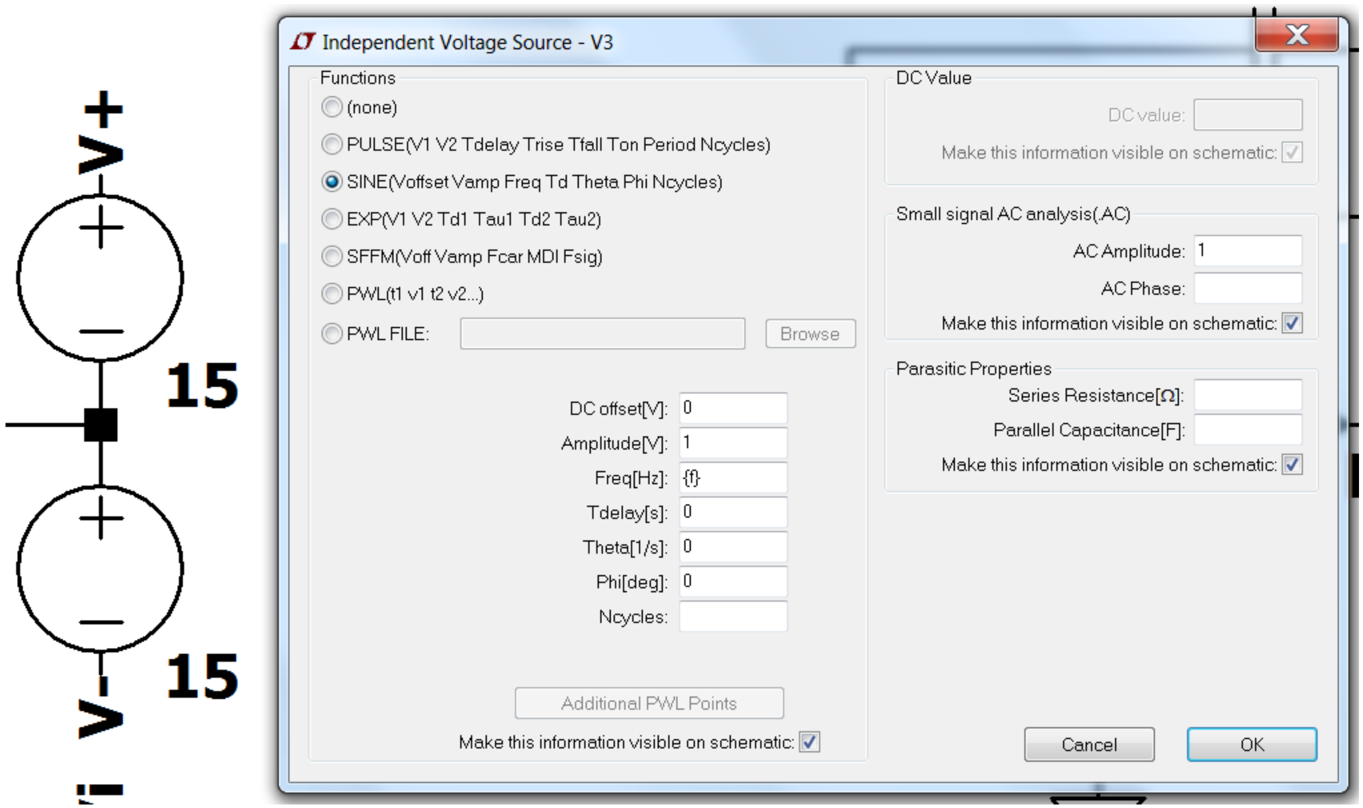
Editing Components



But what about this?

This is the basic voltage source menu. Use this for DC sources such as power supplies or bias voltages.

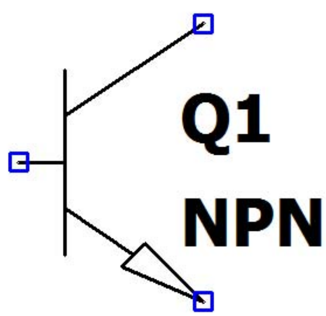
Editing Components

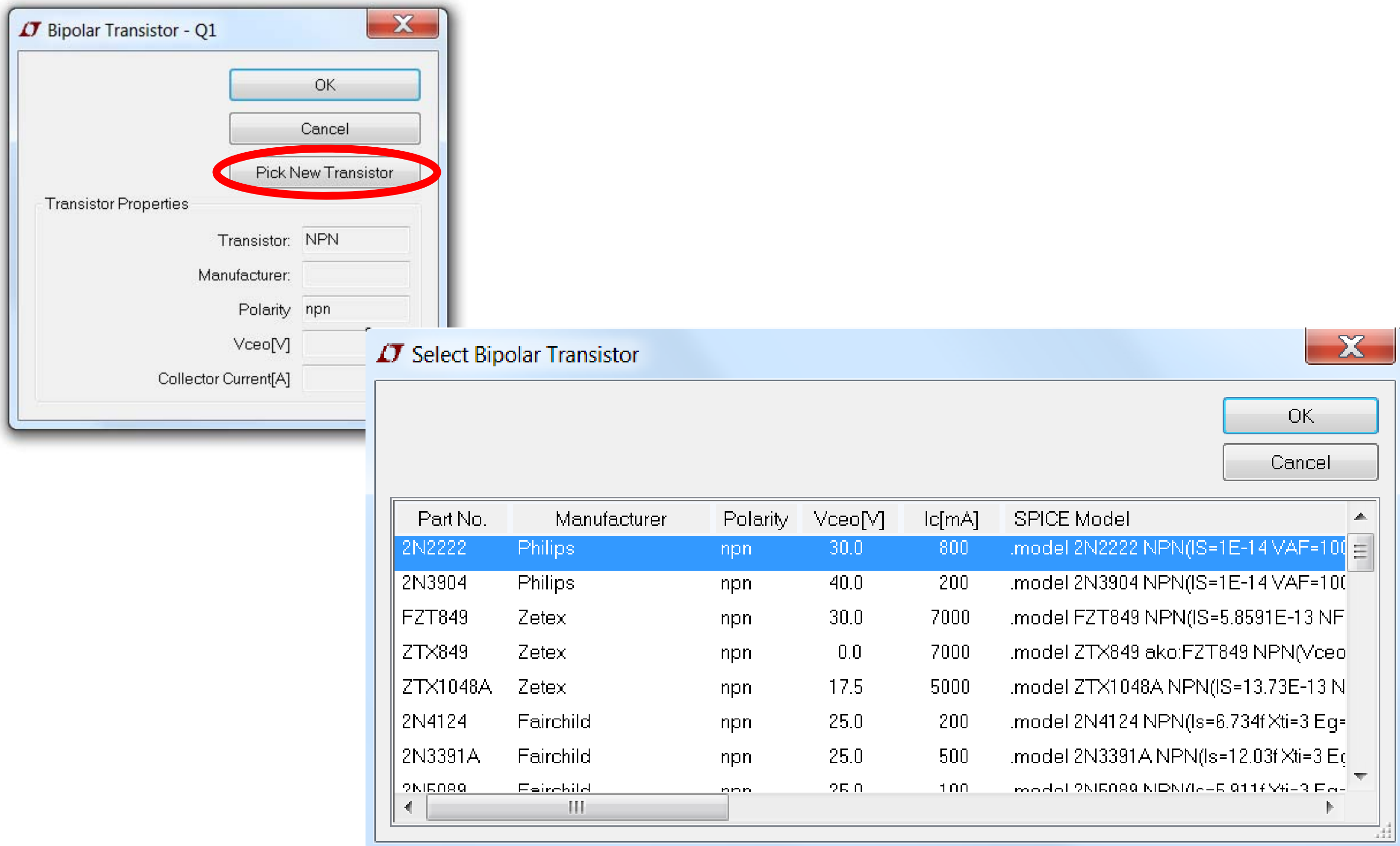


Voltage sources can produce many test signals.
PWL can be used to construct any signal.

Selecting Device Model

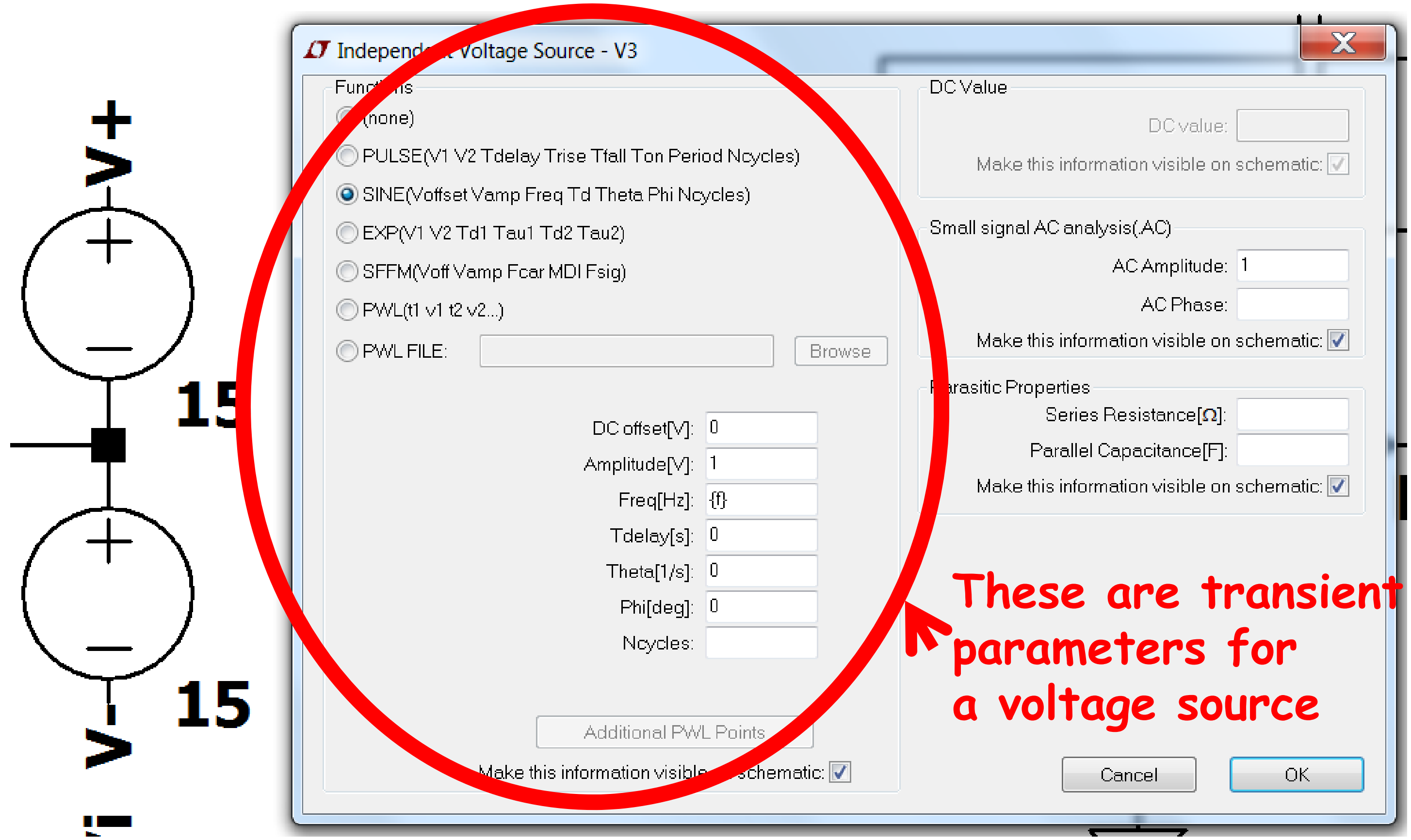
There are no “ideal” BJT’s, MOSFET’s, etc. You can select a model (provided by LTspice), download models, or create your own.



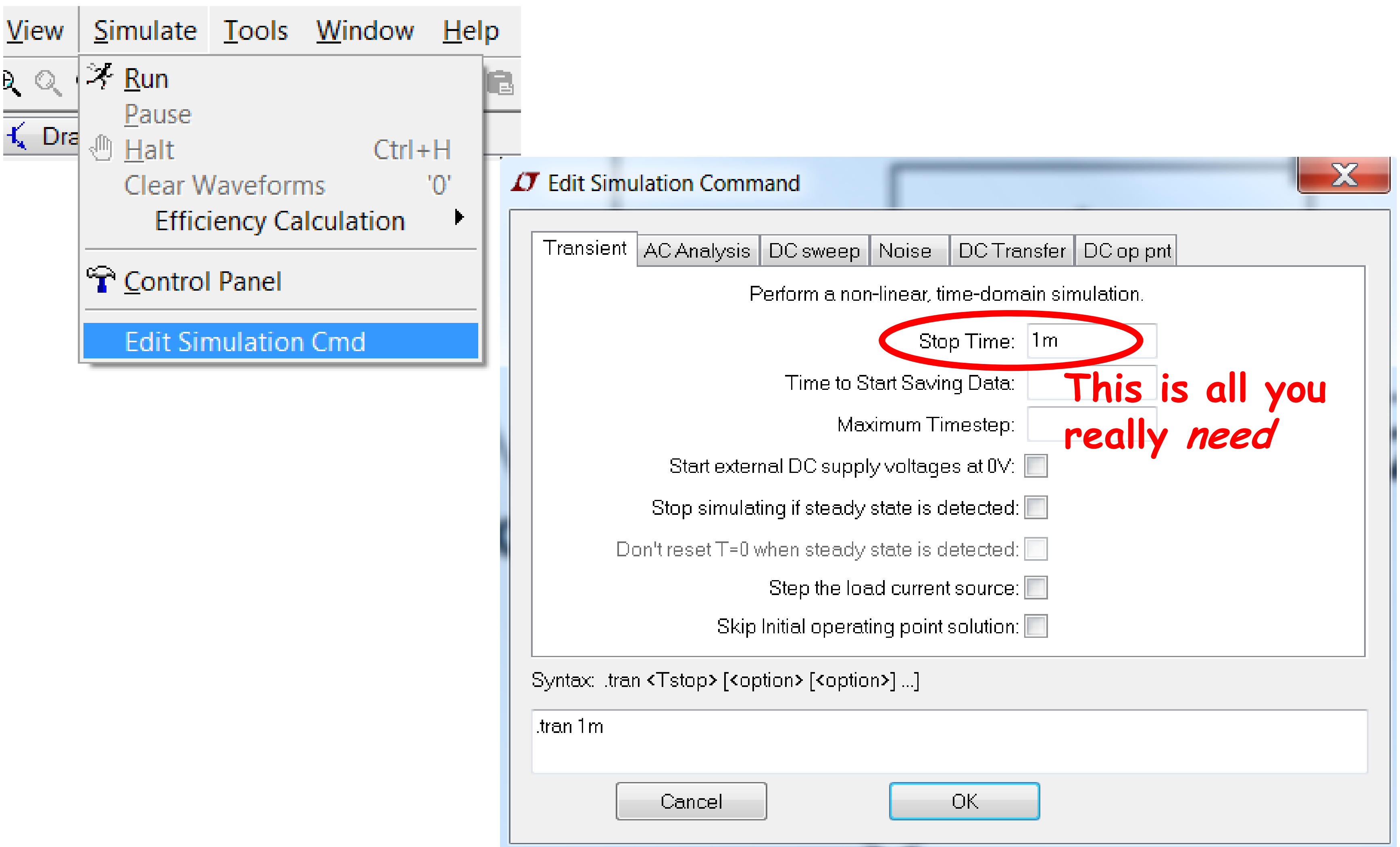


Simulation: Transient

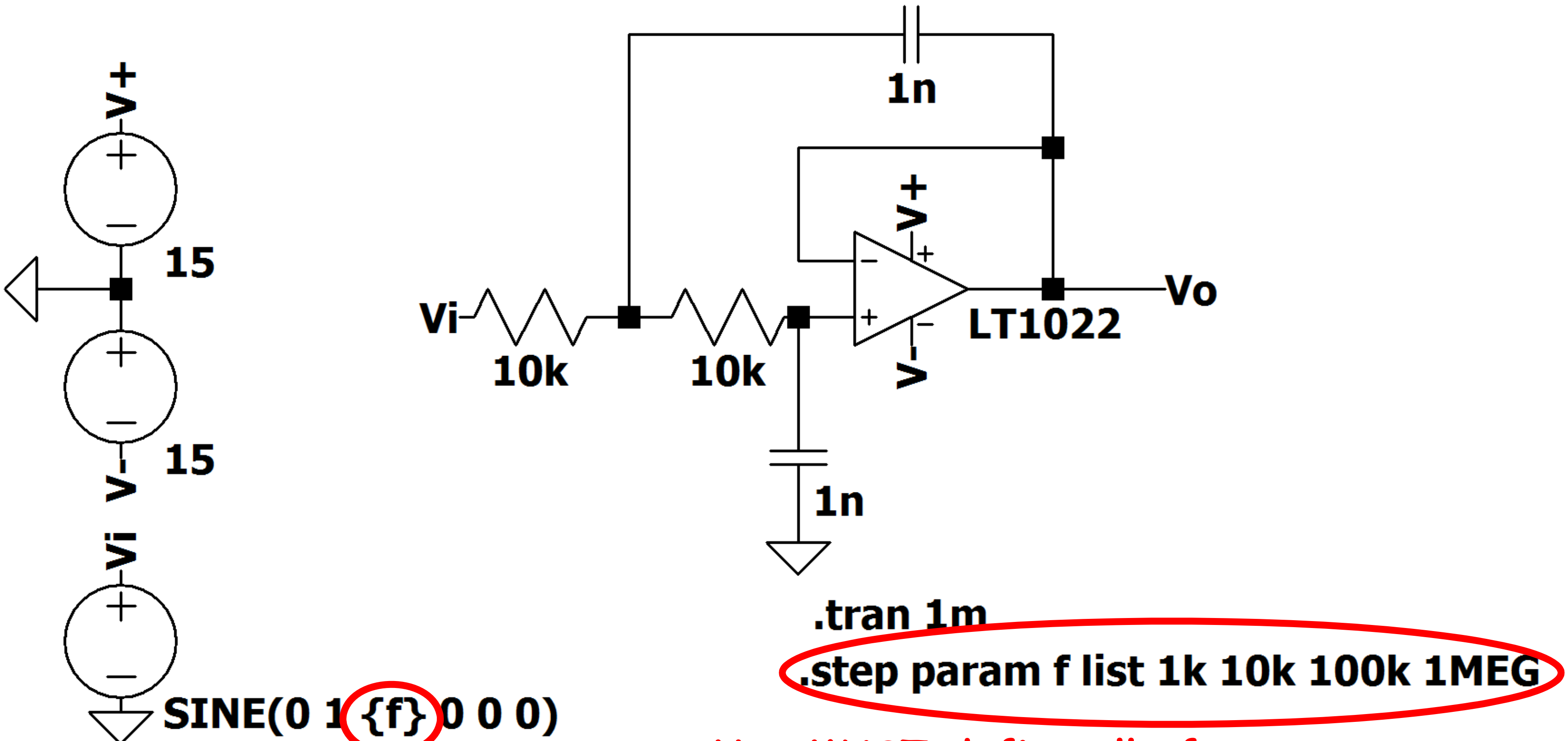
Transient simulation gives Voltage and/or Current vs. *time*.



Simulation: Transient



Random Tangent: Parameters

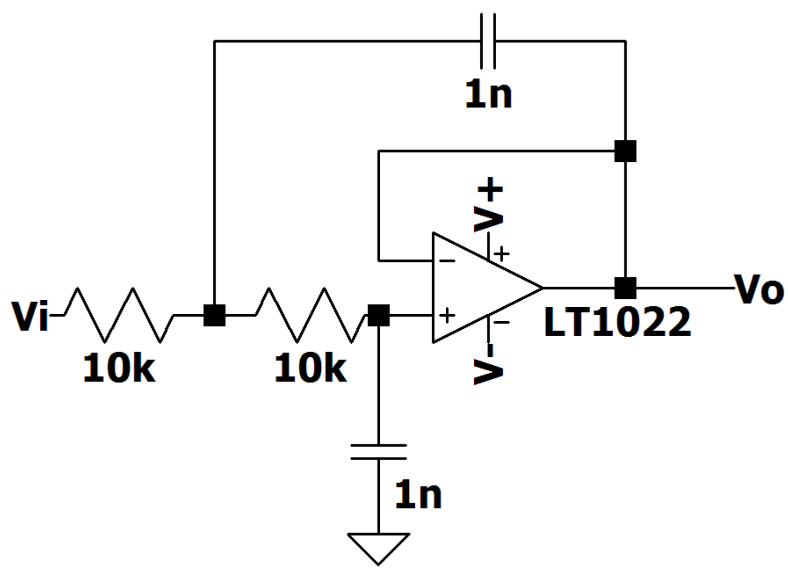


This is a parameter

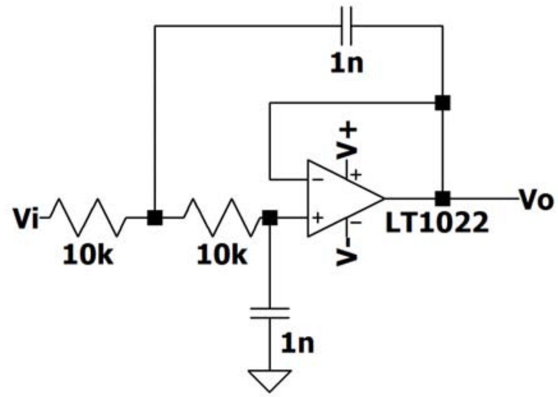
You MUST define all of your parameters. The "list" command allows you to choose multiple values (simulation simulates each value separately).

What Should My Circuit Do?

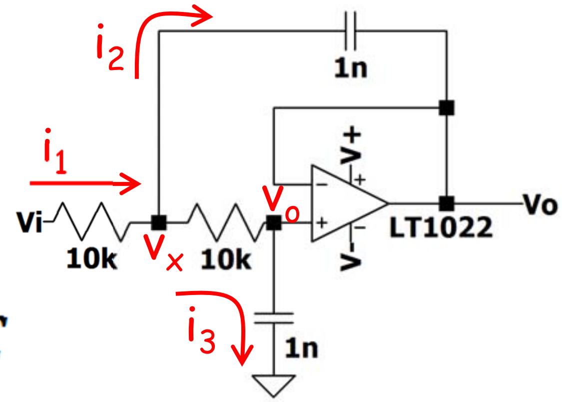
- The very first step to any simulation is to know how your circuit *should* behave. Simulation is a verification tool NOT A CIRCUIT SOLVER.
- So how should this circuit behave?



Here's Where You Write the Solution



Here's Where You Write the Solution



$$i_1 = \frac{v_i - v_x}{R}$$

$$i_2 = (v_x - v_o)sC$$

$$i_3 = \frac{v_x - v_o}{R} = v_o sC$$

$$v_x = v_o(sCR + 1)$$

$$i_1 = i_2 + i_3$$

$$\frac{v_i - v_o(sCR + 1)}{R} = (v_o(sCR + 1) - v_o)sC + v_o sC$$

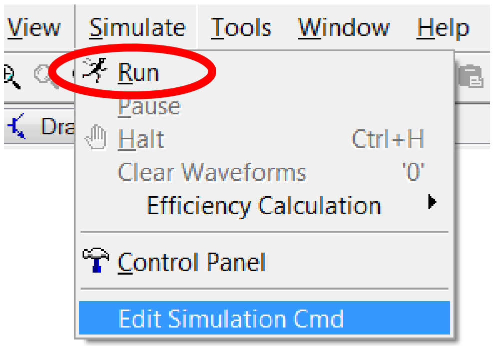
$$\frac{v_o}{v_i}(s) = \frac{1}{(sCR + 1)^2}$$

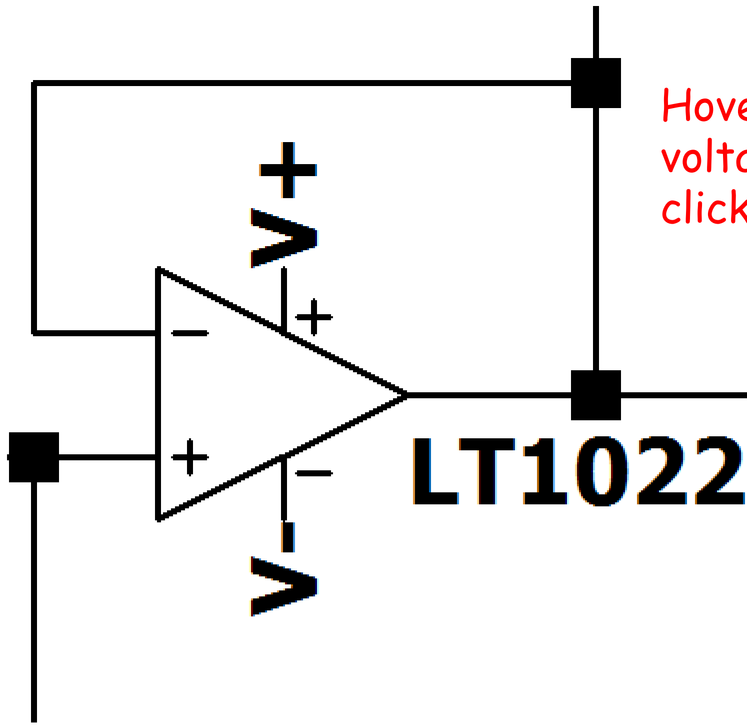
A DOUBLE POLE!!

Expected Behavior

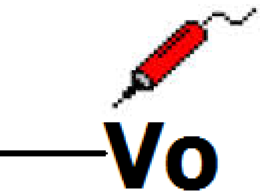
- Double pole is at: $f = \frac{1}{2\pi RC} = 16kHz$
- We expect frequencies up to this point to be large, but frequencies above to quickly drop off due to the -40 dB/decade characteristic of the double pole

Transient Simulation



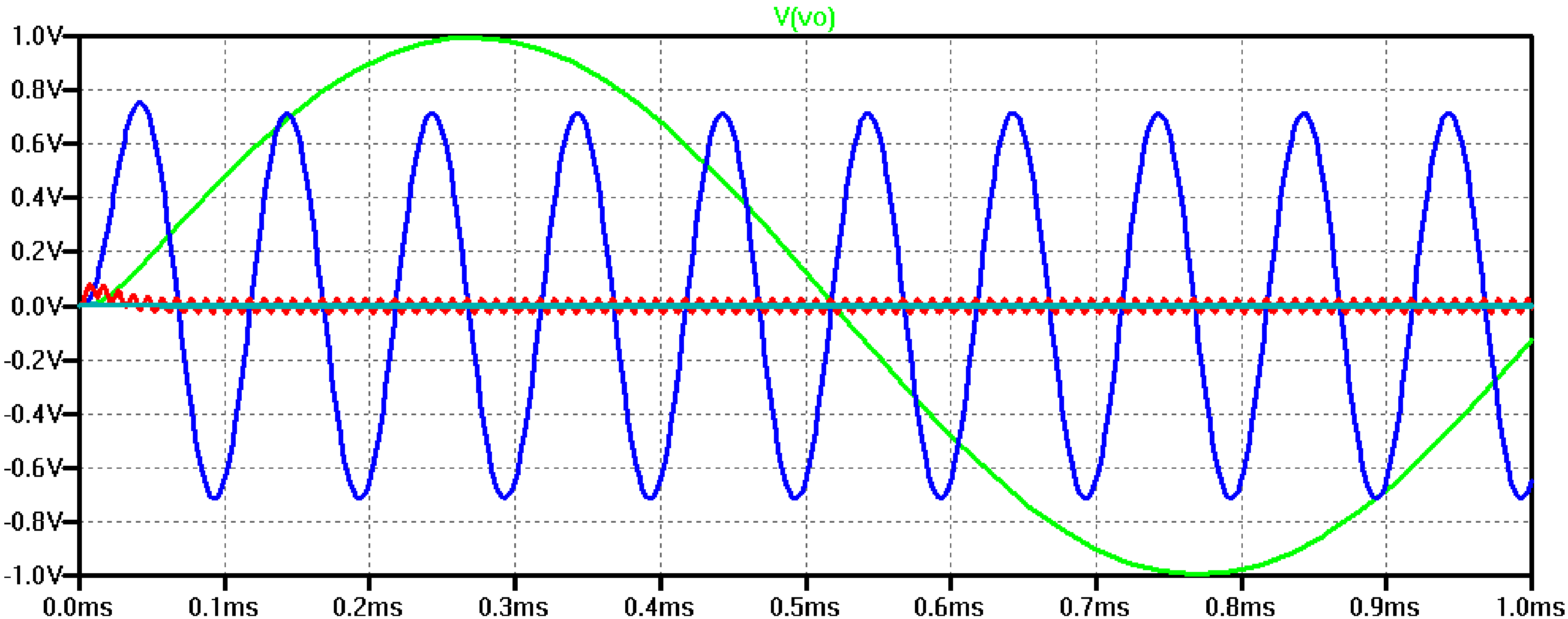


Hover over the desired voltage node to be probed and click when you see this symbol



**This is the current probe 

Transient Simulation

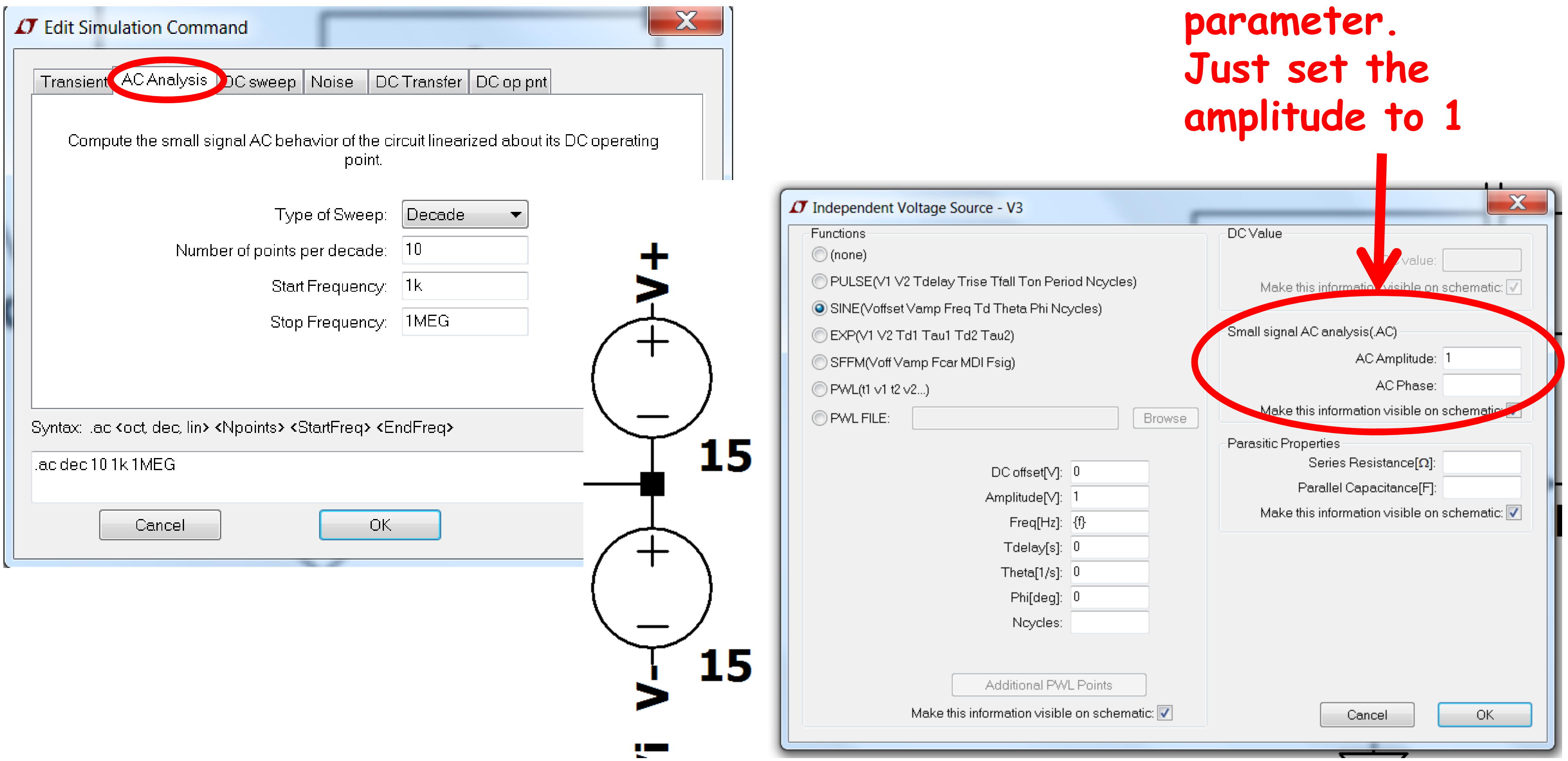


- 1 kHz
- 10 kHz
- 100 kHz
- 1 MHz

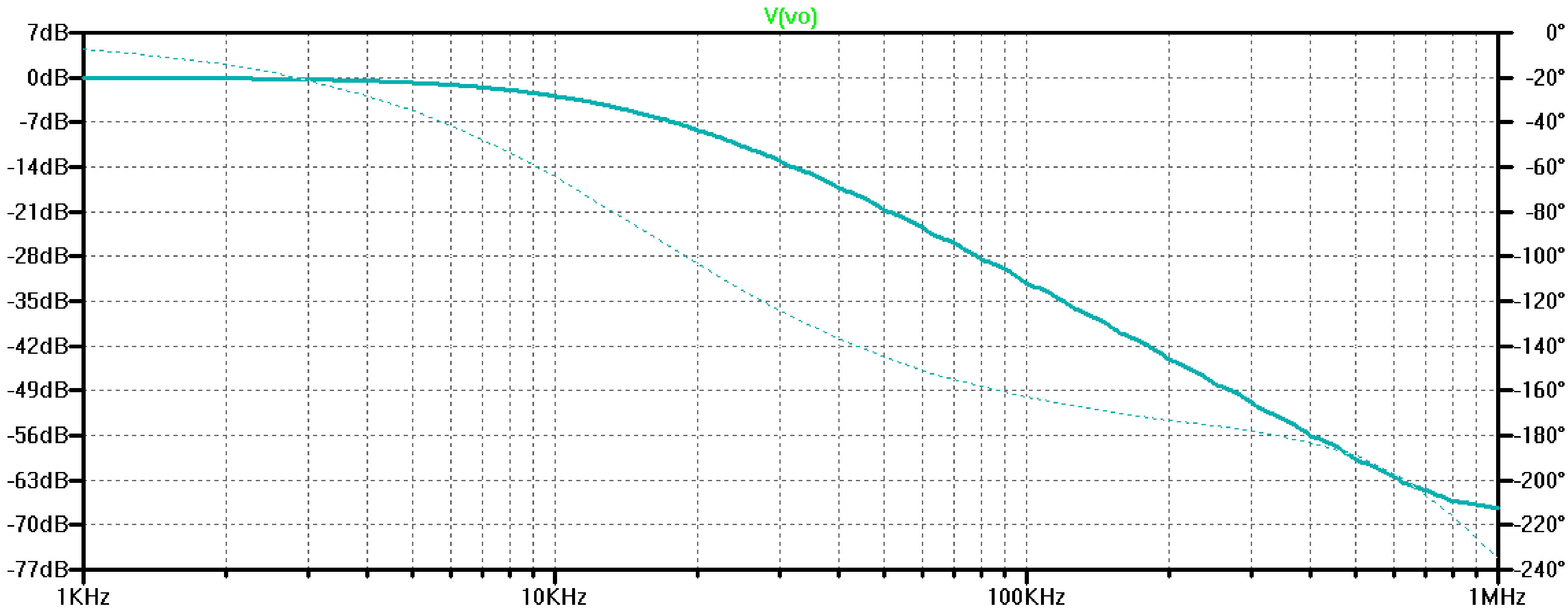
AC Simulation

AC simulation gives Voltage and/or Current vs. __frequency__.

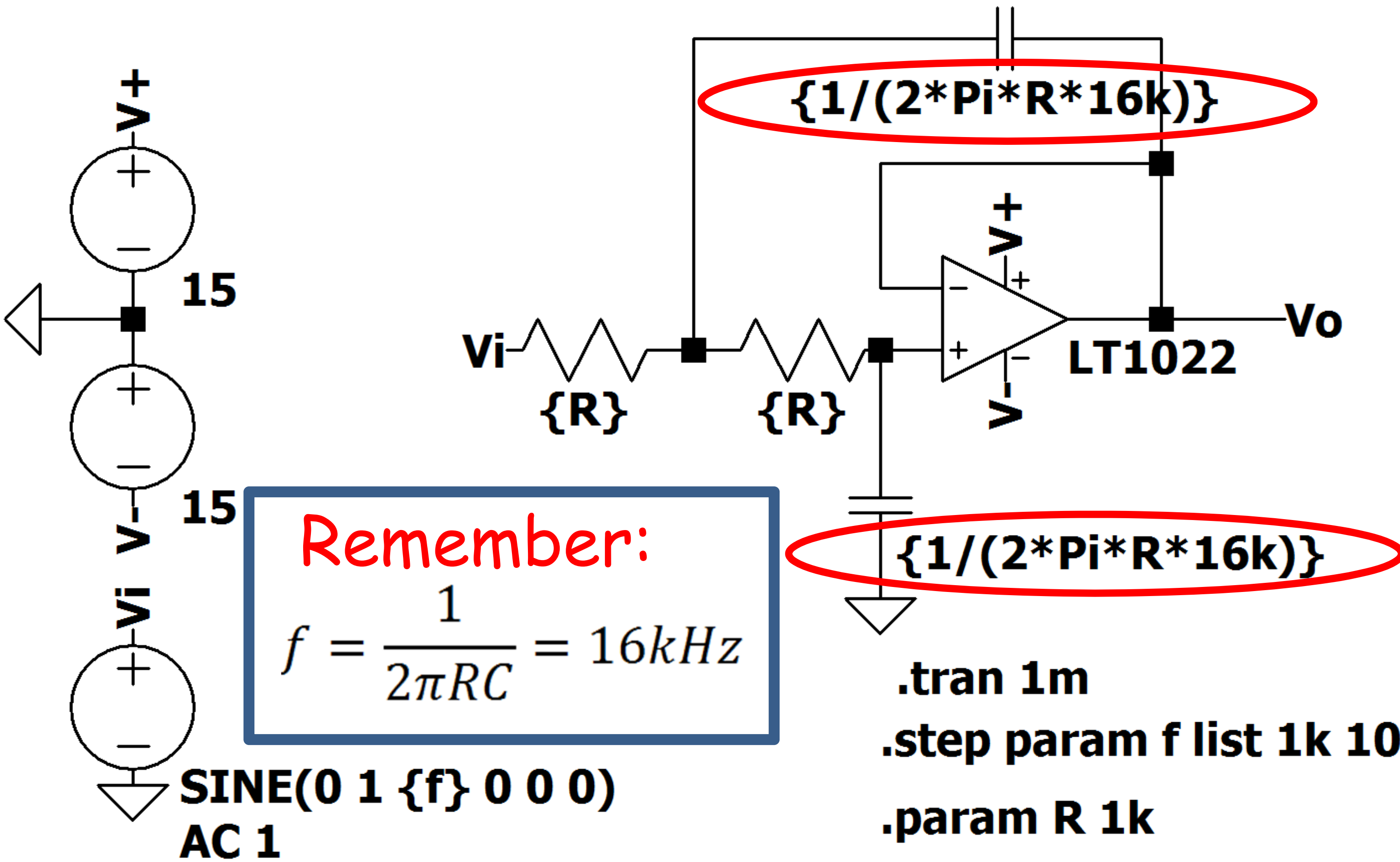
This is the AC parameter. Just set the amplitude to 1



AC Simulation



Extra Fun: Math in LTspice



Remember:

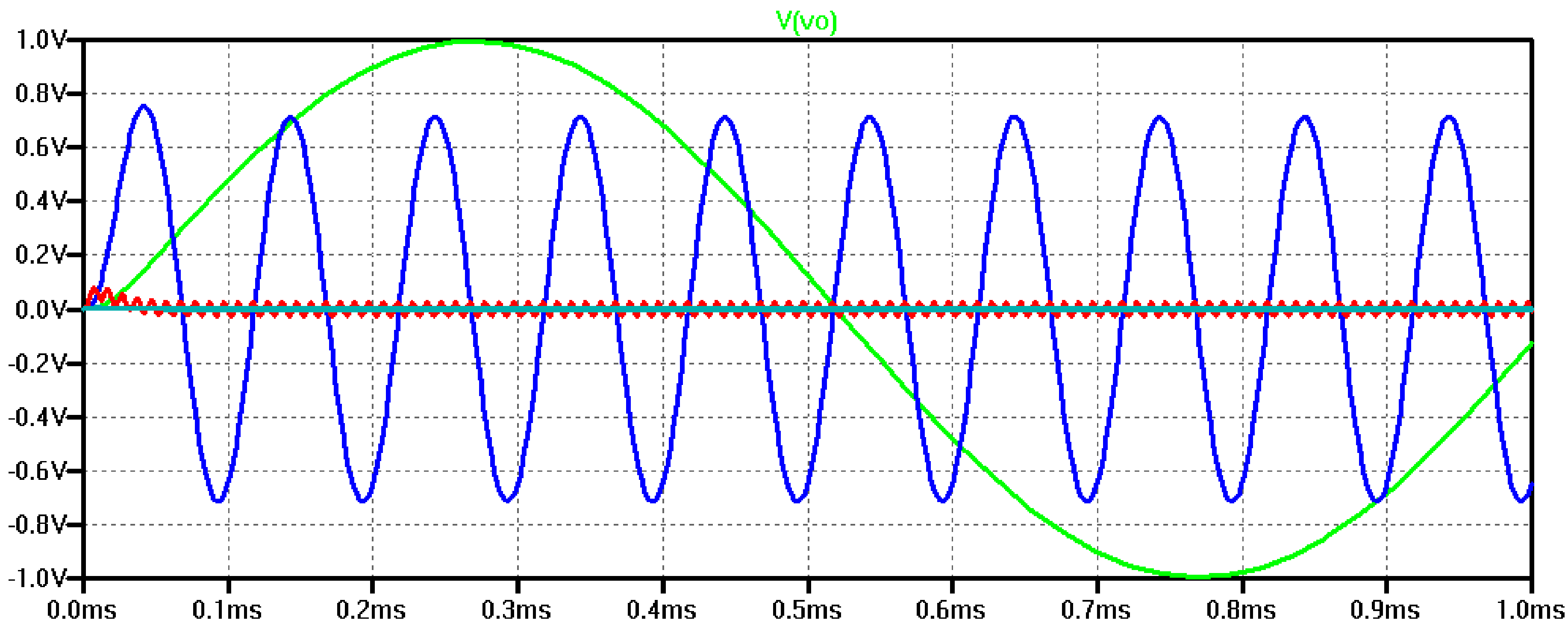
$$f = \frac{1}{2\pi RC} = 16kHz$$

.tran 1m

.step param f list 1k 10k 100k 1MEG

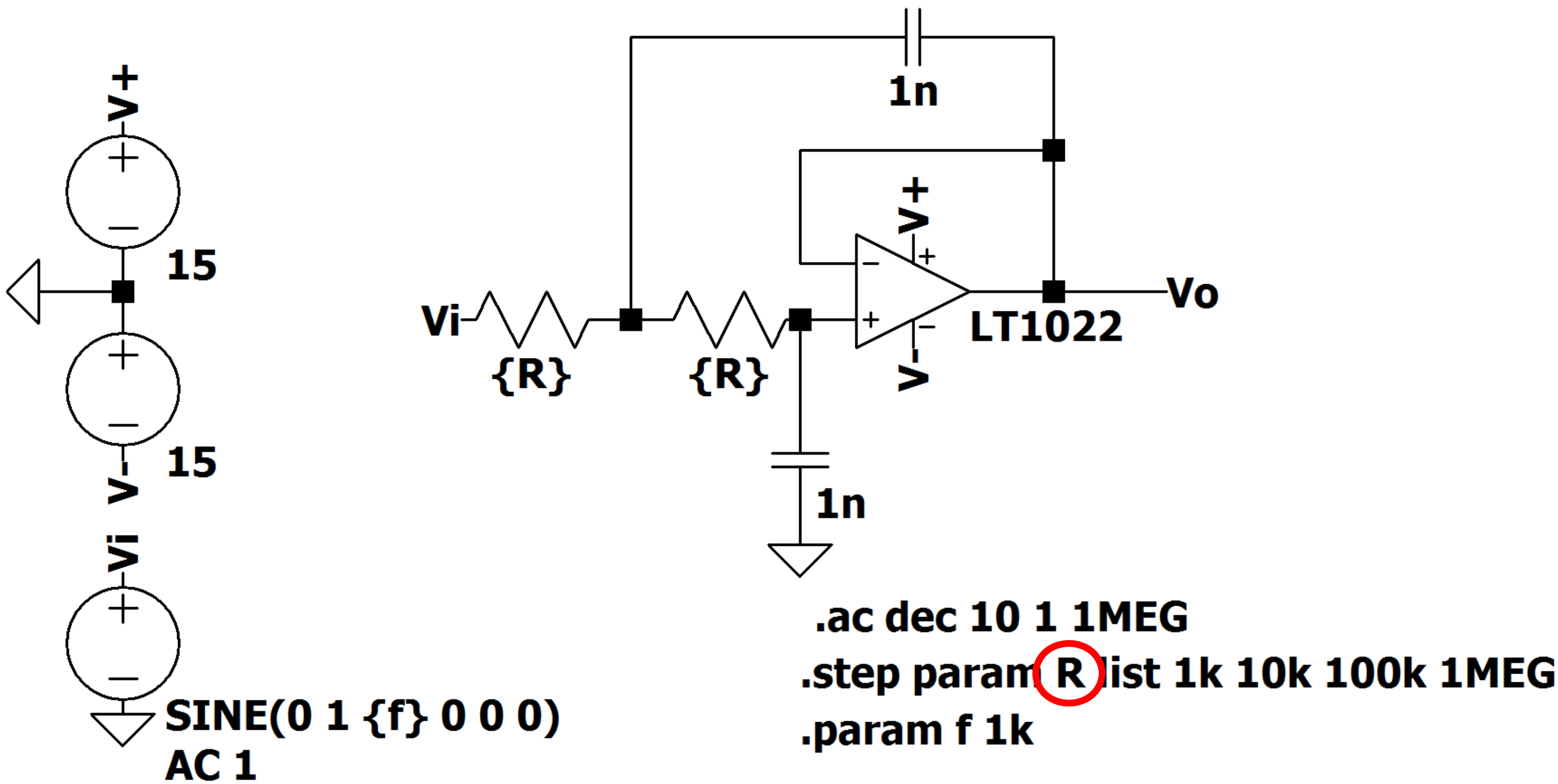
.param R 1k

Transient Simulation



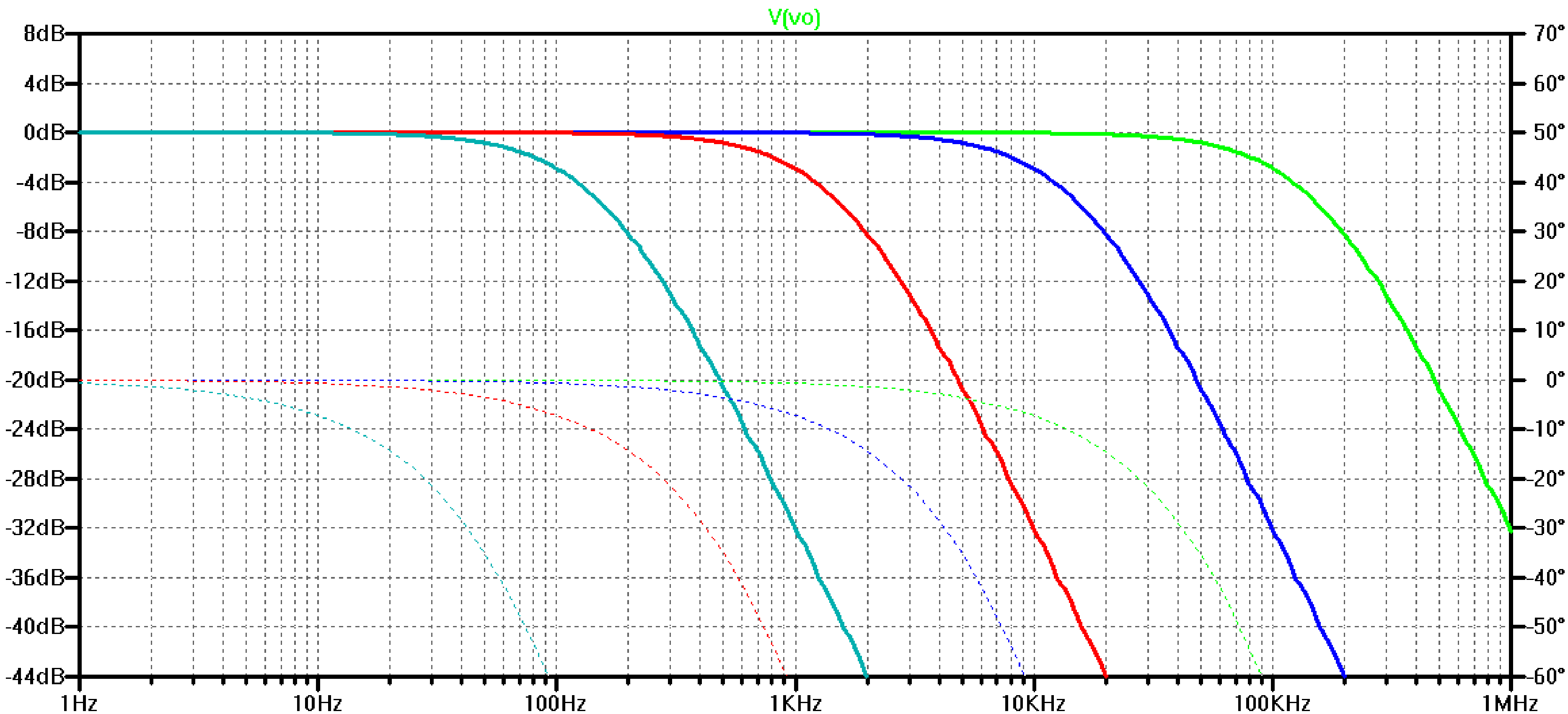
It's the same as before!

Even More Fun



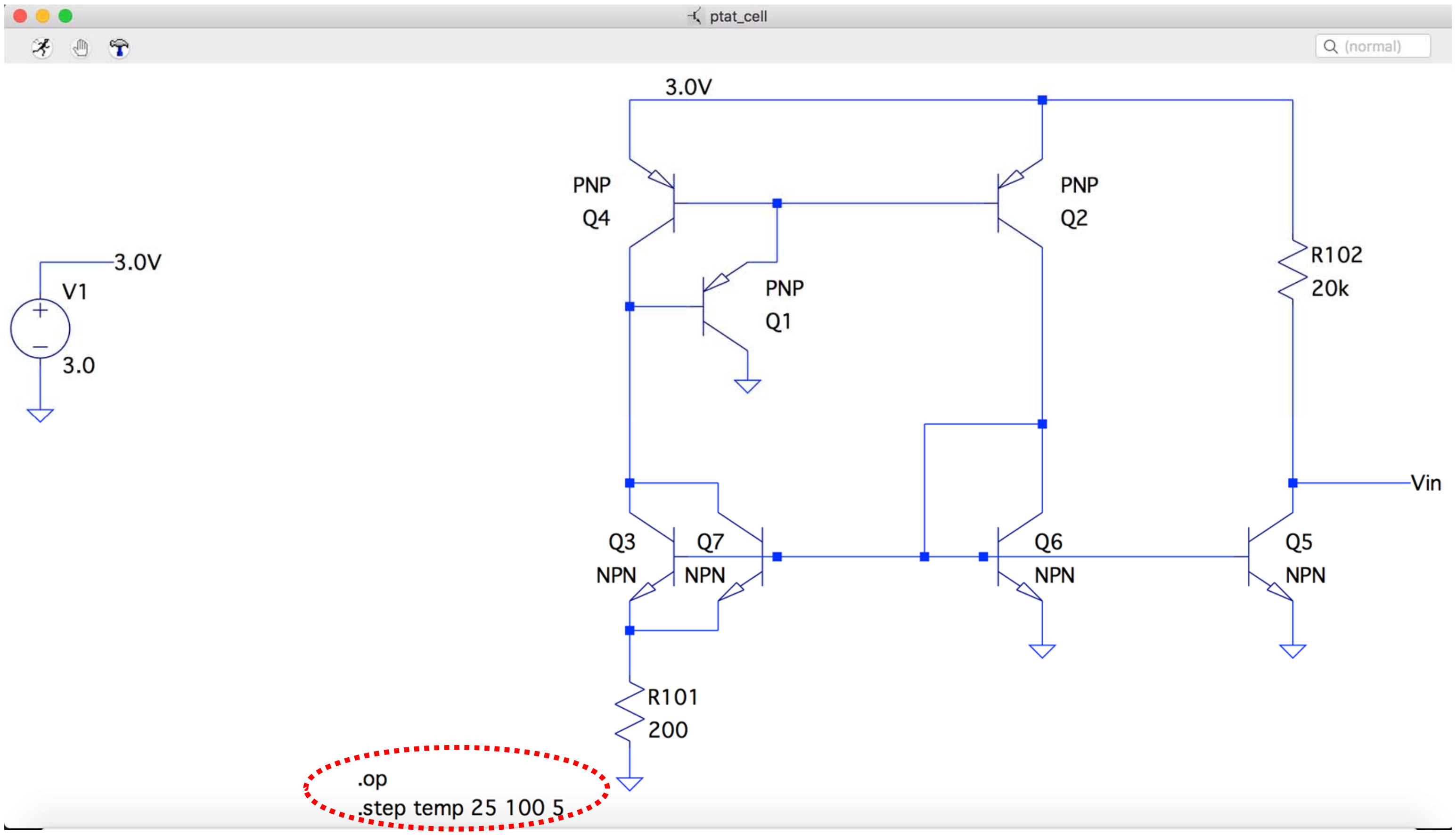
*Note: You can try out some math functions in the simulator window, too! (ex: V(Vo)/V(Vi)).

AC Simulation

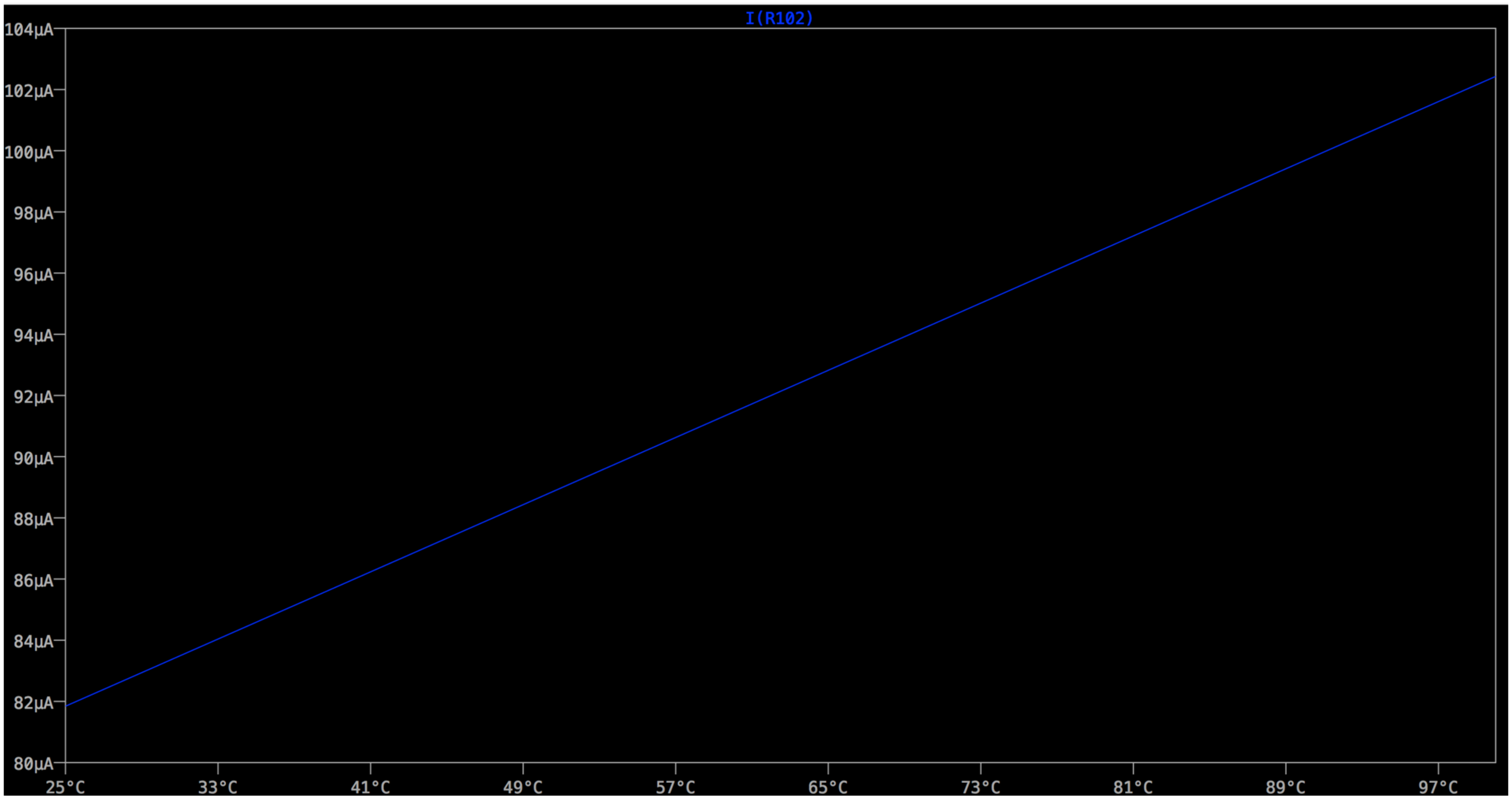


Temperature as a Variable

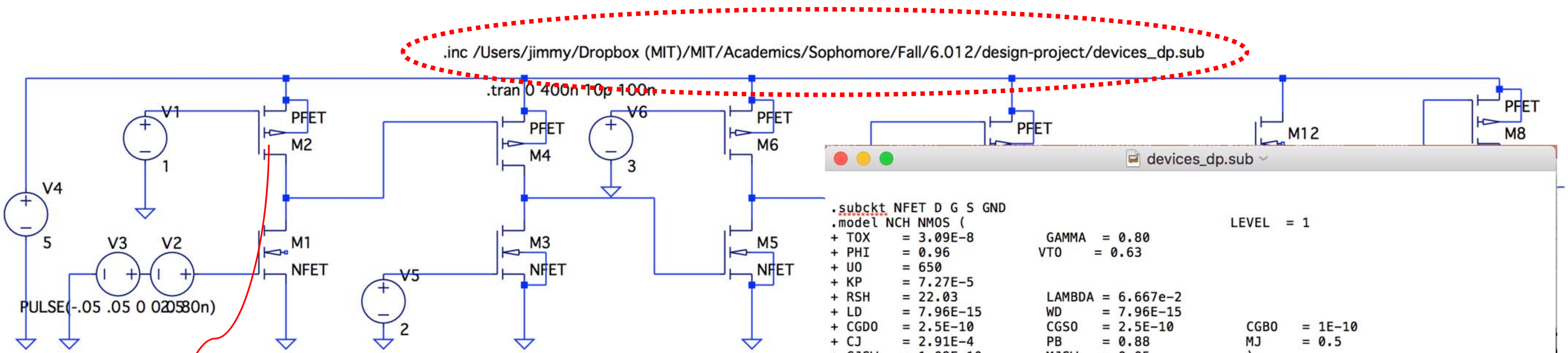
- PTAT current source



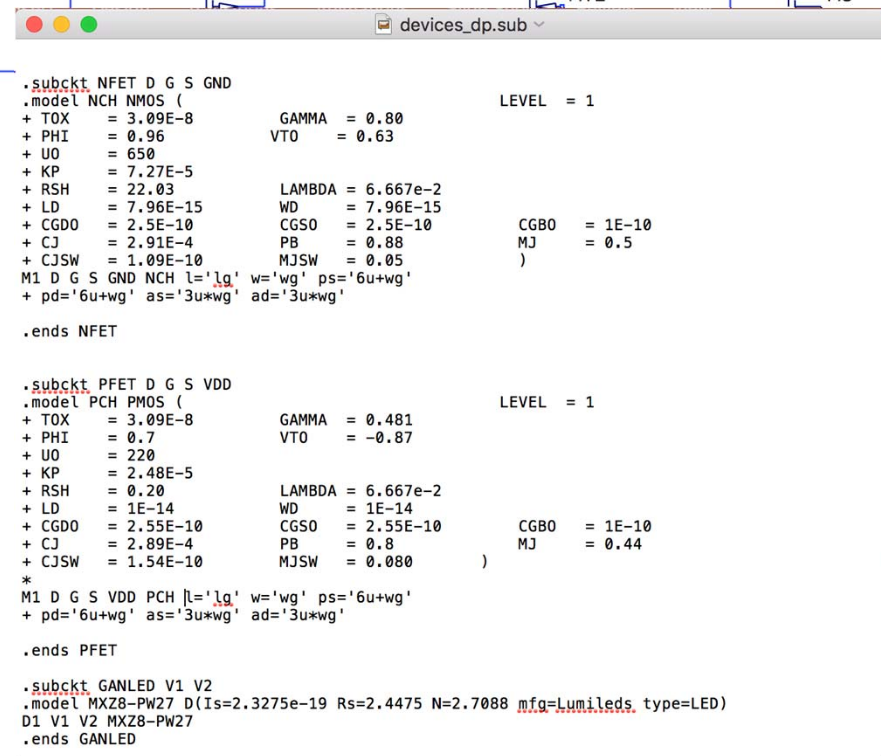
Temperature as a Variable



Including External Models

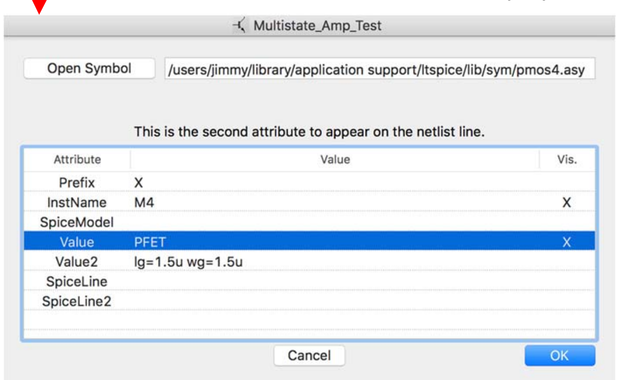


- PFET model
- Includes parameters to describe MOS device physics

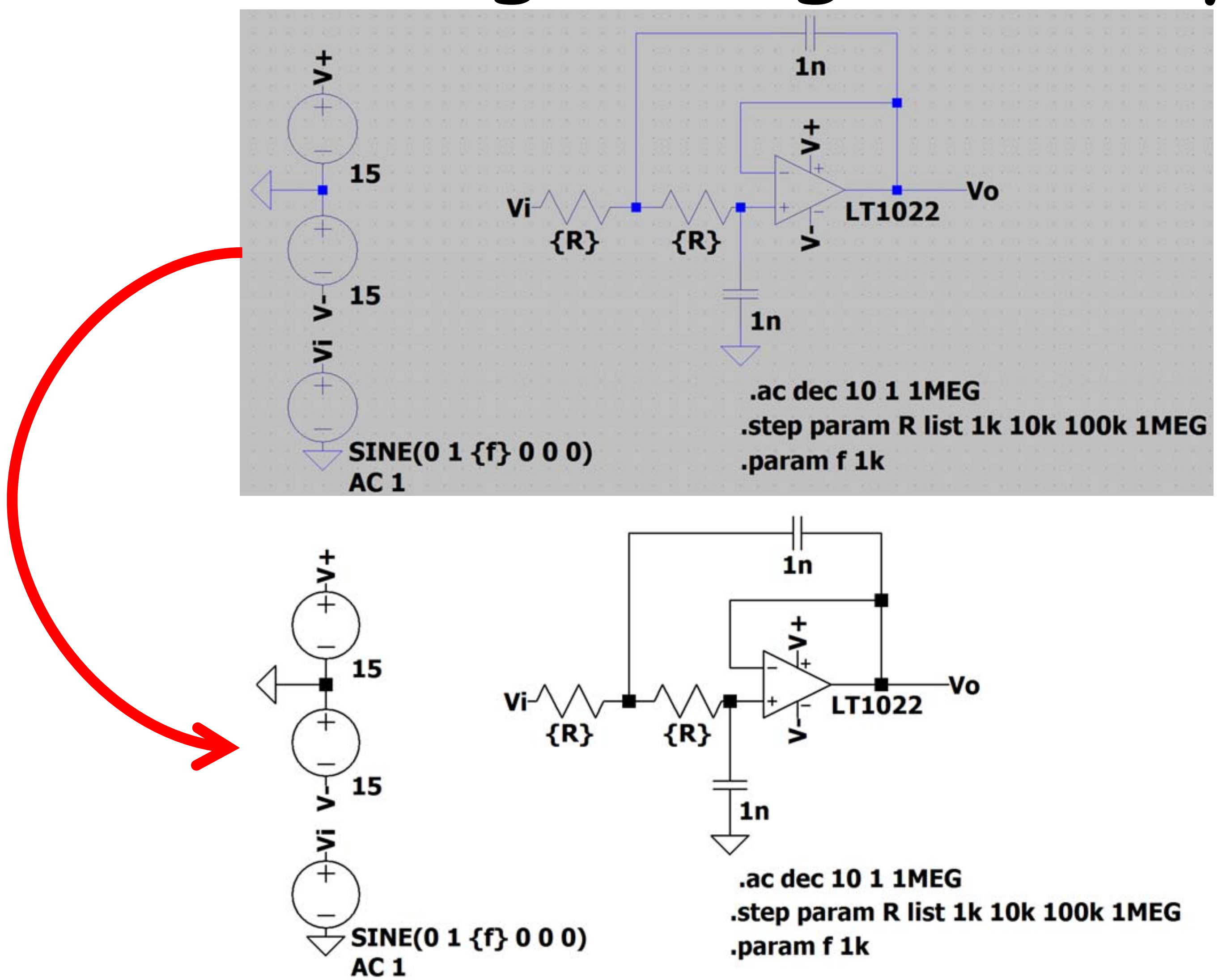


devices_dp.sub

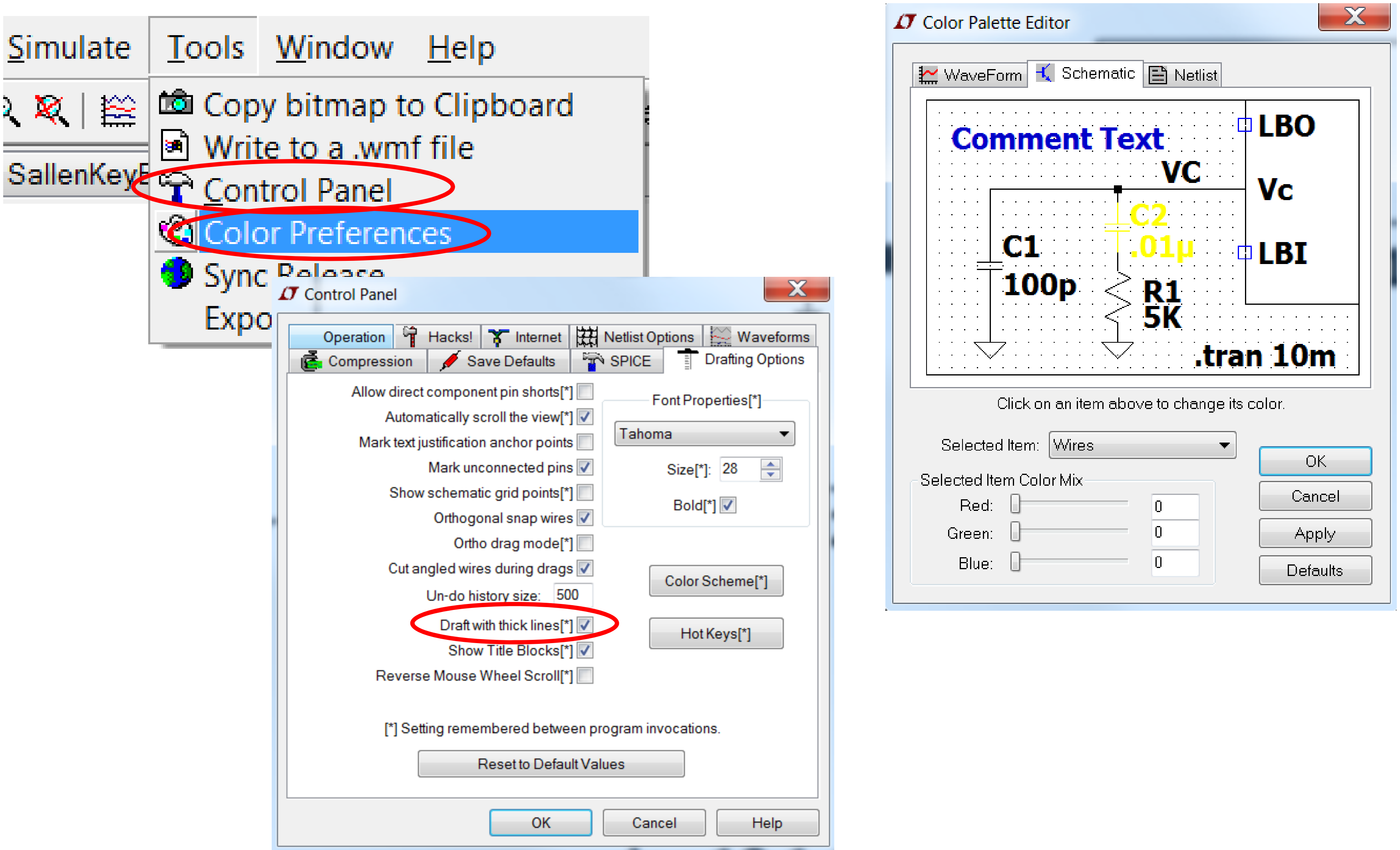
```
.subckt NFET D G S GND
.model NCH NMOS (                                      LEVEL  = 1
+ TOX     = 3.09E-8         GAMMA  = 0.80
+ PHI     = 0.96          VTO    = 0.63
+ UO      = 650
+ KP      = 7.27E-5
+ RSH     = 22.03           LAMBDA = 6.667e-2
+ LD      = 7.96E-15        WD     = 7.96E-15
+ CGDO    = 2.5E-10         CGSO   = 2.5E-10          CGBO   = 1E-10
+ CJ      = 2.91E-4         PB     = 0.88             MJ     = 0.5
+ CJSW    = 1.09E-10        MJSW   = 0.05             )
M1 D G S GND NCH l='lg' w='wg' ps='6u+wg'
+ pd='6u+wg' as='3u*wg' ad='3u*wg'

.ends NFET


.subckt PFET D G S VDD
.model PCH PMOS (                                      LEVEL  = 1
+ TOX     = 3.09E-8         GAMMA  = 0.481
+ PHI     = 0.7             VTO    = -0.87
+ UO      = 220
+ KP      = 2.48E-5
+ RSH     = 0.20            LAMBDA = 6.667e-2
+ LD      = 1E-14           WD     = 1E-14
+ CGDO    = 2.55E-10        CGSO   = 2.55E-10         CGBO   = 1E-10
+ CJ      = 2.89E-4         PB     = 0.8              MJ     = 0.44
+ CJSW    = 1.54E-10        MJSW   = 0.080        )
*
M1 D G S VDD PCH l='lg' w='wg' ps='6u+wg'
+ pd='6u+wg' as='3u*wg' ad='3u*wg'

.ends PFET

.subckt GANLED V1 V2
.model MXZ8-PW27 D(Is=2.3275e-19 Rs=2.4475 N=2.7088 mfg=Lumileds type=LED)
D1 V1 V2 MXZ8-PW27
.ends GANLED
```



Making Things Pretty



Making Things Pretty

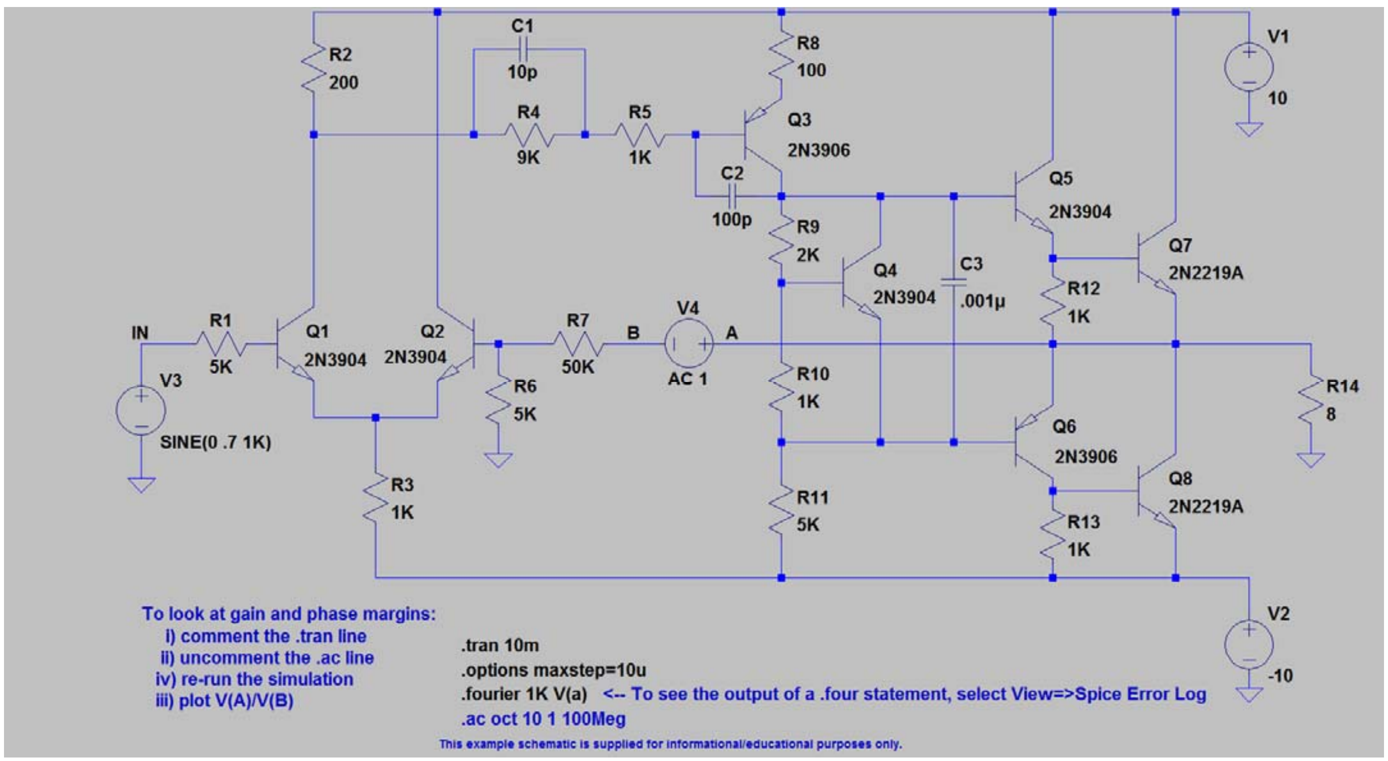


Making Things Pretty

Bob Reay of Linear Technology has provided a nifty tool on his website to give LTspice circuits an even better makeover:

http://reaylabs.com/tools/SchematicViewer/SchematicViewer.html

Before:

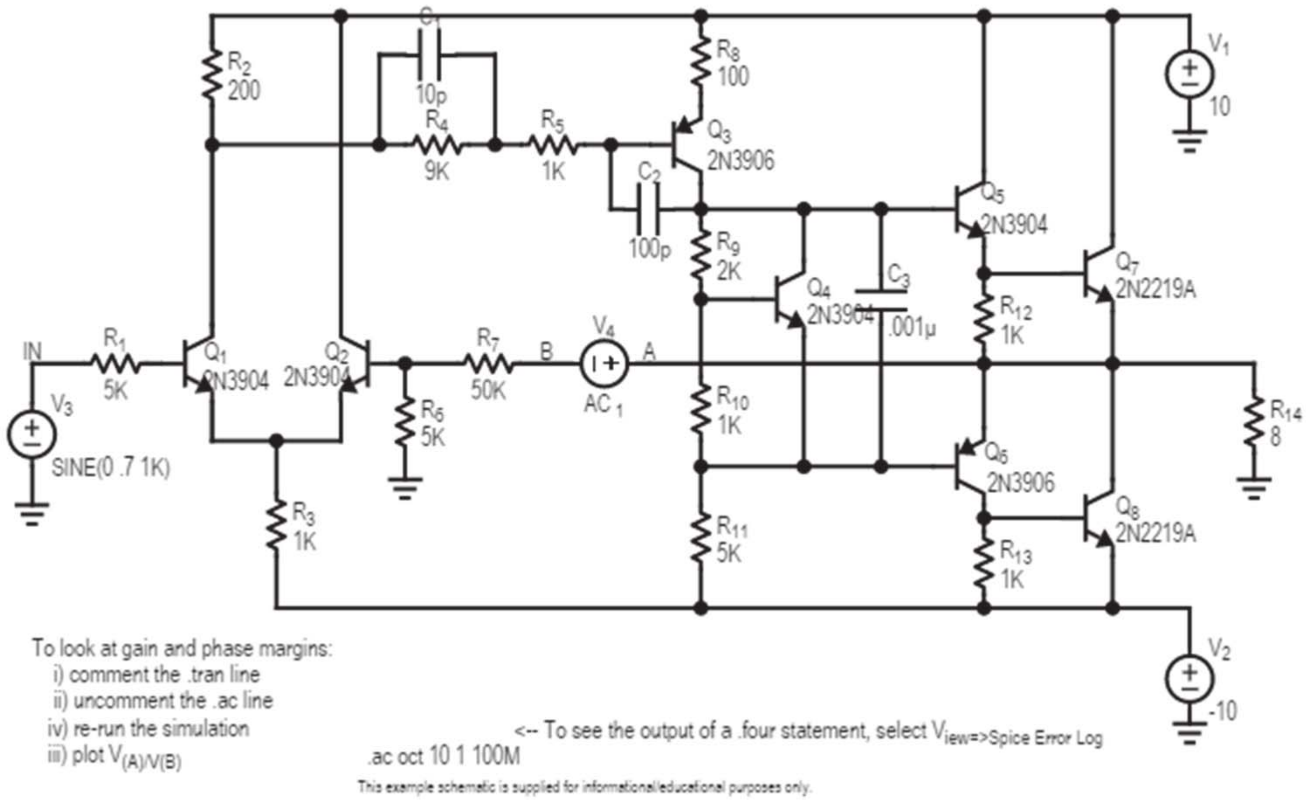


Making Things Pretty

Bob Reay of Linear Technology has provided a nifty tool on his website to give LTspice circuits an even better makeover:

http://reaylabs.com/tools/SchematicViewer/SchematicViewer.html

After:



LTspice *Secrets*

Many aspects and functions of LTspice are not documented. You can learn lots of interesting undocumented capabilities of LTspice from:

http://ltwiki.org/?title=Undocumented_LTspice

Of particular interest should be B-sources. These allow you to make devices such as non-linear resistors whose value is determined from a function of voltage, current, if statements, constants, etc. Though you cannot build these, they may be useful to model a part not available in LTspice, or to model a special function in your circuit you have not designed yet.

Questions??

LTspice HotKeys

	Schematic	Symbol	Waveform	Netlist
Modes	ESC – Exit Mode	ESC – Exit Mode		
	F3 – Draw Wire			
	F5 – Delete	F5 – Delete	F5 – Delete	
	F6 – Duplicate	F6 – Duplicate		
	F7 – Move	F7 – Move		
	F8 – Drag	F8 – Drag		
	F9 – Undo	F9 – Undo	F9 – Undo	F9 – Undo
	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo
View	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	
	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	
	Space – Zoom Fit		Ctrl+E – Zoom Extents	
	Ctrl+G – Toggle Grid		Ctrl+G – Toggle Grid	Ctrl+G – Goto Line #
	U – Mark Unncon. Pins	Ctrl+W – Attribute Window	‘0’ – Clear	
	A – Mark Text Anchors	Ctrl+A – Attribute Editor	Ctrl+A – Add Trace	
	Atl+Click – Power		Ctrl+Y – Vertical Autorange	Ctrl+R – Run Simulation
	Ctrl+Click – Attr. Edit		Ctrl+Click - Average	
	Ctrl+H – Halt Simulation		Ctrl+H – Halt Simulation	Ctrl+H – Halt Simulation
Place	**R – Resistor**	**R – Rectangle**		
	C – Capacitor	**C – Circle**		
	L – Inductor	**L – Line**		
	D – Diode	**A – Arc**		
	G – GND			
	S – Spice Directive			
	T – Text	**T – Text**		
	F2 – Component			
	F4 – Label Net			
	Ctrl+E – Mirror	**Ctrl+E – Mirror**		
	Ctrl+R – Rotate	**Ctrl+R – Rotate**		

Command Line Switches

Flag	Short Description
-ascii	Use ASCII .raw files. (Degrades performance!)
-b	Run in batch mode.
-big or -max	Start as a maximized window
-encrypt	Encrypt a model library
-FastAccess	Convert a binary .raw file to Fast Access Format
-netlist	Convert a schematic to a netlist
-nowine	Prevent use of WINE(Linux) workarounds
-PCBnetlist	Convert a schematic to a PCB netlist
-registry	Store user preferences in the registry
-Run	Start simulating the schematic on open
-SOI	Allow MOSFET’s to have up to 7 nodes in subcircuit
-uninstall	Executes one step of the uninstallation process
-wine	Force use of WINE(Linux) workarounds

Simulator Directives – Dot Commands

Command	Short Description
.AC	Perform a Small Signal AC Analysis
.BACKANNO	Annotate Subcircuit Pin Names on Port Currents
.DC	Perform a DC Source Sweep Analysis
.END	End of Netlist
.ENDS	End of Subcircuit Definition
.FOUR	Compute a Fourier Component
.FUNC	User Defined Functions
.FERRET	Download a File Given the URL
.GLOBAL	Declare Global Nodes
.IC	Set Initial Conditions
.INCLUDE	Include another File
.LIB	Include a Library
.LOADBIAS	Load a Previously Solved DC Solution
.MEASURE	Evaluate User-Defined Electrical Quantities
.MODEL	Define a SPICE Model
.NET	Compute Network Parameters in a .AC Analysis
.NODESET	Supply Hints for Initial DC Solution
.NOISE	Perform a Noise Analysis
.OP	Find the DC Operating Point
.OPTIONS	Set Simulator Options
.PARAM	User-Defined Parameters
.SAVE	Limit the Quantity of Saved Data
.SAVEBIAS	Save Operating Point to Disk
.STEP	Parameter Sweeps
.SUBCKT	Define a Subcircuit
.TEMP	Temperature Sweeps
.TF	Find the DC Small-Signal Transfer Function
.TRAN	Do a Nonlinear Transient Analysis
.WAVE	Write Selected Nodes to a .WAV file

Suffix		Suffix	
		f	1e-15
T	1e12	p	1e-12
G	1e9	n	1e-9
Meg	1e6	u	1e-6
K	1e3	M	1e-3
		Mil	25.4e-6

Constants	
E	2.7182818284590452354
Pi	3.14159265358979323846
K	1.3806503e-23
Q	1.602176462e-19
TRUE	1
FALSE	0

LTspice-6/18(E)

analog.com

ANALOG DEVICES
AHEAD OF WHAT’S POSSIBLE™

LTspice

LTSPICE SHORTCUTS ON A MAC

11/5/2013 REV 3

a		DRAW CIRCLE
b		BUS TERMINATION
g		GROUND
l		DRAW LINE
s		ADD SPICE DIRECTIVE (right click for HELP ME EDIT)
t		ADD TEXT COMMENT
w		DRAW BOX
⌘	H	HIDE LTSPICE
⌘	L	SPICE LOG
⌘	N	NEW SCHEMATIC
⌘	O	OPEN
⌘	Q	QUIT LTSPICE
⌘	S	SAVE
⌘	Z	UNDO
⇧⌘	Z	REDO
⌘	M	MINIMIZE
⌥⌘	M	MINIMIZE ALL
⌘	W	CLOSE
⌥⌘	W	CLOSE ALL
⌘	P	PRINT
⇧⌘	P	page seupt

F2	COMPONENT
F3	WIRE
F4	NET NAME
F5	DELETE
F6	DUPLICATE
F7	MOVE (CNTRL-R to rotate, CNTRL-E to mirror)
F8	DRAG (CNTRL-R to rotate, CNTRL-E to mirror)
F9	UNDO
⇧F9	REDO

SPACE BAR	ZOOM TO FIT
2 FINGER PINCH	ZOOM IN
2 FINGER SPREAD	ZOOM OUT

Here are the modifier key symbols you may see in OS X menus:

⌘	COMMAND
⌥	ALT OR OPTION
⇧	SHIFT