Frequency response of a circuit via SPICE simulation A tutorial (in 20 steps or so) using LTspice by cctsim

The aim is to simulate the frequency response of the 1st-stage tone stack of the Boss BD-2 and optimize individual components, however, the method can be applied to any circuit. Here is the schematic of the circuit to be simulated:



Step 0: Download and install LTspice from

http://ltspice.linear.com/software/LTspiceIV.exe

Step 1: Start LTspice and create a new schematic and save it. The name is arbitrary, I've used "boss BD2 1"



Step 2: To place a component, such a resistor, capacitor etc, we can use the available toolbar:



Step 3: Click on the resistor icon and press Ctrl+R to rotate the component (if necessary). To delete a component use the DEL button on your keyboard.

Ø	LTspice IV • [boss_BD2_1.asc]	- ^ ×
/ Elle Edit Hjerarchy Yiew Simulate Iools Window 교 교 대 약 가 가 이 옷 이 있 것	500 日間線 16 10 10 10 10 10 10 10 10 10 10 10 10 10	n ser La ser
Type Ctrl+R to rotate or Ctrl+E to mirror.		

Step 4: To change the name of the resistor, right click above R1 and type R36 to conform with the BD-2 schematic:

R1 ₽∕\/_₽	
R	
Justification	OK
Bottom	Cancel
R36	

Step 5: To change the value of the resistor, right click above R and type the required value. You can use k for kilo, and Meg for Mega.

D LTspice IV - [boss_BD2_1.asc]	- ^ ×
🐔 Eile Edit Hierarchy View Simulate Iools Window Help	_ 코 ×
፼፼፼₽♀⊀ଶ €<<<<<><><> < <	₽<≠3\$D00000
R36 ↓ ↓ ↓ 100k	Move
Move a section of the schematic.	

Step 6: To move the component we can use the corresponding icon:



Step 7-9: Repeat a similar procedure for the capacitor C26. For capacitor values, u stands for micro, n for nano and p for pico.

C	🗊 Enter new Valu	ie for C26 🛛 🗙
	Justification	OK
	Vertical Text	Cancel
$^{*} \vee \vee^{\bullet}$	220p	

Step 10: Repeat similar procedure for the rest of the circuit. Finally, we add two ground symbols too.

Step 11-12: Now we need to add a voltage source by clicking on the component icon and selecting voltage from the dialog window:

Ø	Līspice IV - [boss_BD2_1.asc]	- ^ ×
『Ele Edit Hierarchy Yew Simulate Iools Window @ @ ■ ■ 帝 孝 』 电 へ Q 裂 論	₩0 [] □ ☜ ☜ ४ ☜ ा ⋈	≭ছ⊾ ি ৫ ৫ ৫ ⊈
	C26 R37 220p 330k	Kg Component
	R36 100k C1 R50 0.047µ C1 1Meg	
	R51 ~ 15k ~	
Place a new component on the schematic.		

Select Component Symbol					
Top Directory	Top Directory: C:\Program Files\LTC\LTspicelV\lib\sym				
	+	Voltage Source, either DC, AC, PULSE, SINE, PWL, EXP, or SFFM			
	Ē	voltage			
C:\Program	C:\Program Files\LTC\LTspiceIV\lib\sym\				
nmos nmos4 npn npn2 npn3 npn4 pjf pmos pmos4 pnp pnp2	pnp4 polcap res res2 schottky sw tline varactor voltage zener				
	Cancel				

Step 13: Right click on the voltage symbol to enter the required parameters and click "Advanced":

	Voltage S	ource - V1	×
	DC value[V]: Series Resistance[oh		OK Cancel Advanced
V1	0.047µ	100 <	2

Step 14: For the frequency analysis, we enter DC 0V, AC 1V and phase 0 (degrees) on the corresponding entries.

🗖 📃 Independent Voltage	Source - V1 X
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: 1V AC Phase: Make this information visible on schematic: Parasitic Properties Series Resistance[oh Parallel Capacitance[F]: Make this information visible on schematic: ✓
Additional PWL Points Make this information visible on schematic: 🔽	Cancel OK

Step 15: Once finished, we can use the "Wire" icon to connect the components:

Step 16: The complete circuit should look as below. To run the simulation we press the corresponding icon:

Step 17: From the simulation dialog we select the AC analysis TAB and enter the following parameters and press OK:

Edit Simulation Command 🛛 🕹
Transient ACAnalysis DC sweep Noise DC Transfer DC op pnt
Compute the small signal AC behavior of the circuit linearized about its DC operating point.
Type of Sweep: Decade 💌
Number of points per decade: 1001
Start Frequency: 10
Stop Frequency: 10k
Syntax: .ac <oct, dec,="" lin=""> <npoints> <startfreq> <endfreq></endfreq></startfreq></npoints></oct,>
.ac dec 1001 10 10k
Cancel OK

Step 18: Once the simulation is complete we can click on a specific node to see either the voltage of current response. Since we have used the 1V as input the output voltage (at right side of C34) represents the voltage transfer function of this circuit.

Step 19: To add grid on the plot, you can right click on the plot and select grid.

U LTspice IV - boss_BD2_1.raw	<i>i</i>	- ^ ×
File View Plot Settings Simulation Iools Window Help	白昏 / - + 明 く = 3 文 D	80 0 06
weboss_BD2_1.raw		
V(n003)		
0dB 4dB -8dB -12dB -16dB	Zoom Area Ctrl+Z Zoom Back Ctrl+B Zoom to Et Ctrl+E Pan Autorange Y-axis Ctrl+Y	
-20dB	rm (Jahoa Links	-50* 10KHz
Cite 107 200 200	Model Inace Ctrl+A Model Delete Traces F5 Model State F5	لفالله
	Add Blot Pane	
ac dec 1001 to 100 x = 608.262Hz y = 2.698dB, 6.746*	Marching <u>w</u> avetorms ► Grid ≈ Reset Colors	

Step 20: To get rid of the phase plot, left click on the right axis of the plot and select "Don't plot phase"

D	Right Vertical Axis	×
Range Top: 5° Tick: 5° Bottom: -50°	Representation Phase Unravel Branch Wrap Group Delay	OK Cancel
	Don't plot phase.	

The final result:

Now we want to change the values of a specific component, lets say R36, and see how the frequency response is affected. In this example, we change R36 to take the following values: 10k, 22k, 47k, 100k and 220k.

To do that, we need to use a parametric value for R36. From the edit menu we select "Spice Directive" or press S on the keyboard and enter the following:

.step param var list 10k 22k 47k 100k 220k

We need also to modify the value of R36 to {var}. The variable name var here is arbitrary.

If we run the simulation again we obtain the parametric frequency plot with respect to varying R36.

Similarly, we can use the same principle to vary any other components in the circuit.