Microelectronic Circuits 8th Edition

A. Sedra, K.C. Smith T. Chan Carusone, V. Gaudet

LTSpice Simulation Guide

Prepared by: Nijwm Wary

2019

© Oxford University Press, 2020

LTSpice is a free SPICE simulator available from Analog Devices. It is currently available here, along with extensive documentation and examples to help you get started:

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

Furthermore, a wide array of tutorials and guides are readily available online.

Using the LTSpice simulation examples for Microelectronic Circuits

1. Copy any of the provided netlists into a text file and save it with the *.cir extension. In Windows, this can be done using the *Save As* option. Give it a filename <filenam>.cir and in *Save as type* chose the option *All files* as shown below.

Save As		×
← → • ↑ 📙	« Spice_Solutions > problems v 👌 Searc	h problems 🔎
Organize 🔻 Ne	w folder	≣≡ ▼ ?
RFP Spice_Solution OneDrive	ns Name No items match your se	Date modified Type
💻 This PC	v <	>
File name: Save as type:	P2_12.cir All Files	~
∧ Hide Folders	Encoding: ANSI ~	Save Cancel

- 2. Start *LTSpice* from the *Program Menu*.
- 3. Select *File>Open* and go to the folder where you have saved the *.cir file. Note that to see the file, the *File of type* should be set to *Circuit Files (*.cir)*.
- 4. To run the simulation (Transient, AC Sweep, DC Sweep, Operating Point) select *Simulate>Run*.
- 5. In most of the problems, upon running the simulation, you will see a blank simulation results window as shown below



© Oxford University Press, 2020

6. To plot a waveform (voltage or current), right click in the blank area of the above window and select *Add Trace*. The following window will pop up.

🗗 Add Traces to Plot		×		
Available data:	Only list traces matching	OK Cancel		
V(a) I(V_v1) V(b) I(V_vcc) V(c) I(V_vee) V(n1) time V(n2) k_u1:1) V(n3) k_u1:2) V(n4) k_u1:3) V(vcc) k_u1:4) V(vee) k_u2:1) I(R_r1) k_u2:1) I(R_r3) k_u2:3) I(R_r5) k_u2:4) I(R_r6) tx_u2:5)				
Expression(s) to add:				
✓ Auto Range				

7. Select the waveform you want to plot. Or type the expression in the text box *Trace Expression*. You can also perform various mathematical functions on the simulation results.

© Oxford University Press, 2020