

Santa Clara University	ELEN 115	Dr. Shoba Krishnan
LTSpice Startup Guide		

Part 1: Getting Started with LTSpice

LTSpice is a free SPICE (Simulation Program with Integrated Circuit Emphasis) simulator available from Analog Devices. SPICE simulators are valuable software for a designer, allowing them to verify their designs before sending them out for production.

LTSpice is currently available at the below link, along with extensive documentation and examples to help you get started. Download the software and get it installed on your computer.

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

Please use the **LTSpice** tutorial in Camino as a guide for completing this part of the lab.

Please read sections:

I. Opening LTspice

II. Drawing the circuit

A. Making Sure You Have a GND

B. Getting the Parts

C. Placing the Parts

D. Connecting the Circuit

E. Changing the Name of the Part

F. Changing the Value of the Part

G. Using Net Labels

H. Adding your own SPICE Models or Subcircuits (Skim briefly this week. It will be useful for the opamp lab)

I. Saving

J. Printing

III. Simulation

A. Before you do the simulation

B. Choosing a simulation

C. Graphing

D. Adding/Deleting Traces

E. Doing Math

F. Labelling

G. Finding Points (aka Using Cursors)

H. Saving

I. Printing

IV. Simulation Commands

A. DC Sweep

B. DC Operating point

C. Transient

D. AC Analysis

V. Types of Sources

A. Voltage Sources

1. DC

2. PULSE

3. SINE