

EE 221 L CIRCUIT II

LABORATORY 3: LTSPICE

DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING
UNIVERSITY OF NEVADA, LAS VEGAS

OBJECTIVE

Learn to use LTspice to run circuit simulations for voltage, current, etc.

COMPONENTS & EQUIPMENT

Computer

LTspice software

BACKGROUND

SPICE (*Simulation Program with Integrated Circuit Emphasis*) is a general-purpose, open source analog electronic circuit simulator. It is a program used in integrated circuit and board-level design to check the integrity of circuit designs and to predict circuit behavior.

LTspice is a high-performance SPICE simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulators. The enhancements to SPICE have made simulating switching regulators extremely fast compared to normal SPICE simulators, allowing the user to view waveforms for most switching regulators in just a few minutes. Included in this download are LTspice, Macro Models for majority of Linear Technology's switching regulators, over 200 op amp models, as well as resistors, transistors and MOSFET models.

LAB DELIVERIES

PRELAB:

1. Read:

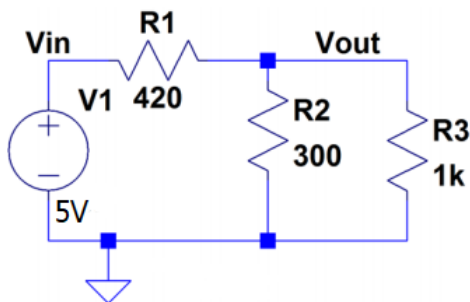
https://en.wikipedia.org/wiki/Electronic_circuit_simulation
<https://en.wikipedia.org/wiki/SPICE>

LAB EXPERIMENTS:

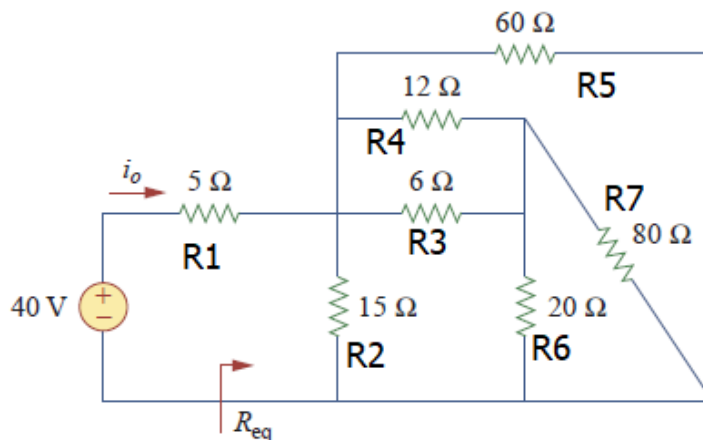
1. Go through “Simulations Step-by-Step with Schematic Files” starting from Page 4 of this manual.

- Demonstrate the final results to TA.

2. Simulate the following circuit for 1ms, and compare the output voltage and current through each resistor with the results in Lab 1.



3. Simulate the following circuit for 1ms, and write down the voltage at each node and the current through each resistor.



POSTLAB REPORT:

Include the following elements in the report document:

Section	Element	
1	Theory of operation <i>Include a brief description of every element and phenomenon that appear during the experiments.</i>	
2	Prelab report Why do we need to run simulations before implement real circuits?	
3	Results of the experiments	
	Experiments	Experiment Results
	1	Screenshots of simulation results
	2	Screenshots of simulation results
	3	Screenshots of simulation results.
4	Answer the questions	
	Questions	Questions
	1	Why do we need to use SPICE for simulations? What are the advantages?
	2	What's the value of Req in Experiment 3?
5	Conclusions <i>Write down your conclusions, things learned, problems encountered during the lab and how they were solved, etc.</i>	
6	Images <i>Paste images (e.g. scratches, drafts, screenshots, photos, etc.) in Postlab report document (only .docx, .doc or .pdf format is accepted). If the sizes of images are too large, convert them to jpg/jpeg format first, and then paste them in the document.</i> Attachments (If needed) <i>Zip your projects. Send through WebCampus as attachments, or provide link to the zip file on Google Drive / Dropbox, etc.</i>	

REFERENCES & ACKNOWLEDGEMENT

1. C. K. Alexander and M. Sadiku, "Fundamentals of Electric Circuits", 4th Ed.
2. <http://www.linear.com/designtools/software/>

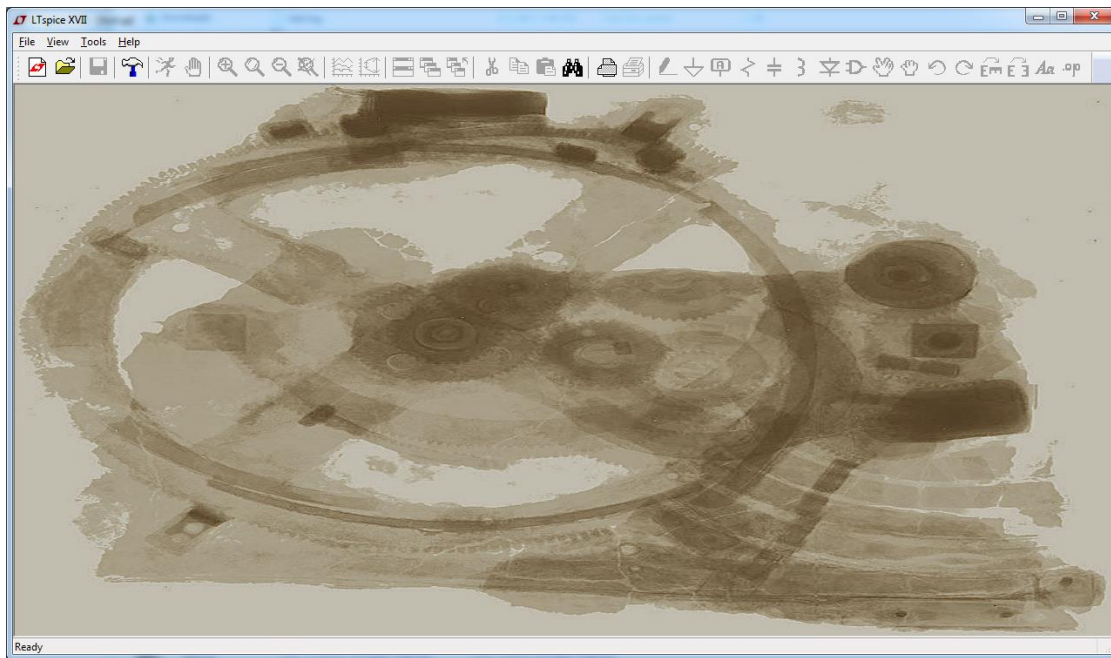
I appreciate the help from faculty members and TAs during the composing of this instruction manual. I would also thank students who provide valuable feedback so that we can offer better high education to the students.

Simulation Step-by-Step with Schematic Files

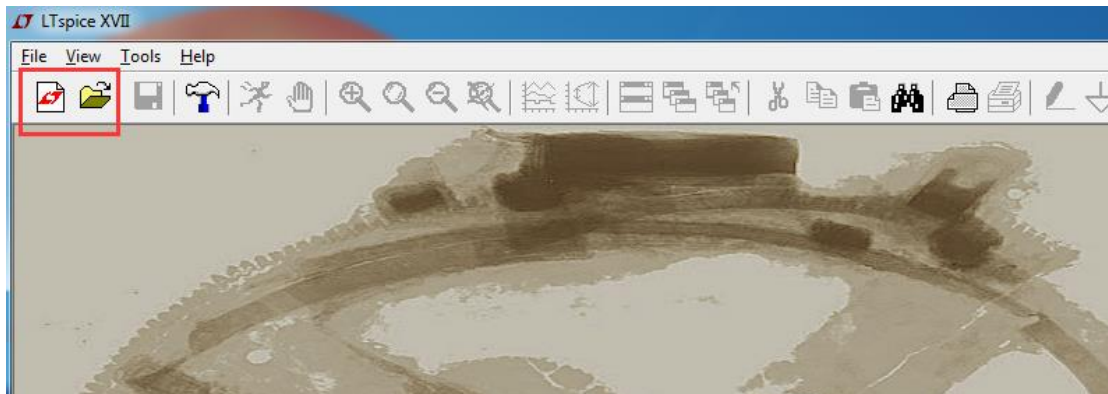
LTspice can run simulations on either schematic files or coded netlist files. Both methods are quite similar, and only simulations with schematic files are introduced in detail here.

Step 1: Install LTspice and open the software (Click No if requested update).

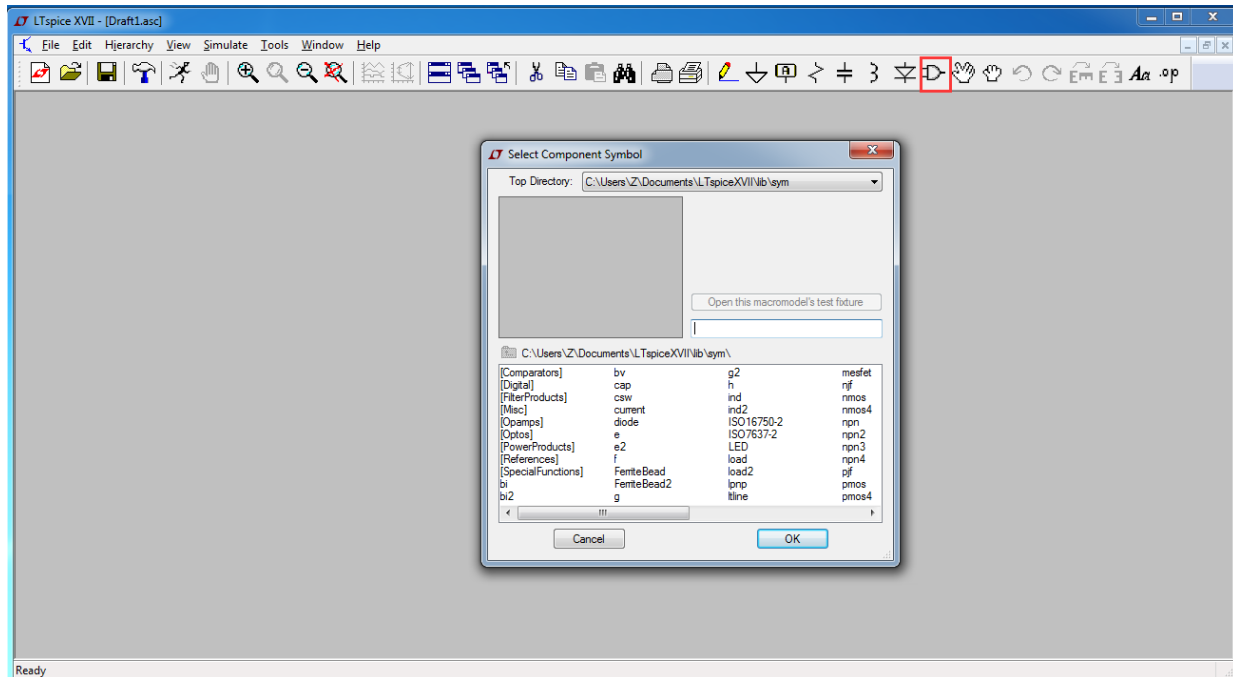
Current version: LTspice XVII.



Step 2: Create a new schematic, or open an existing schematic file.

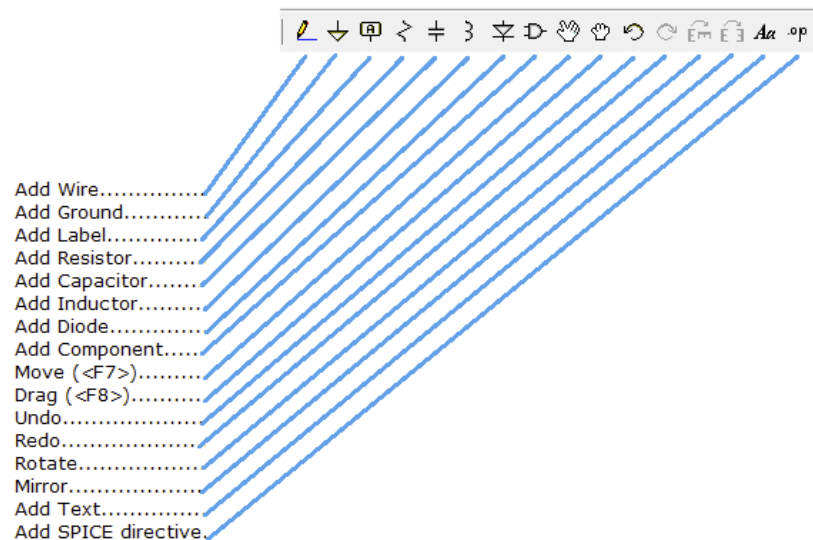


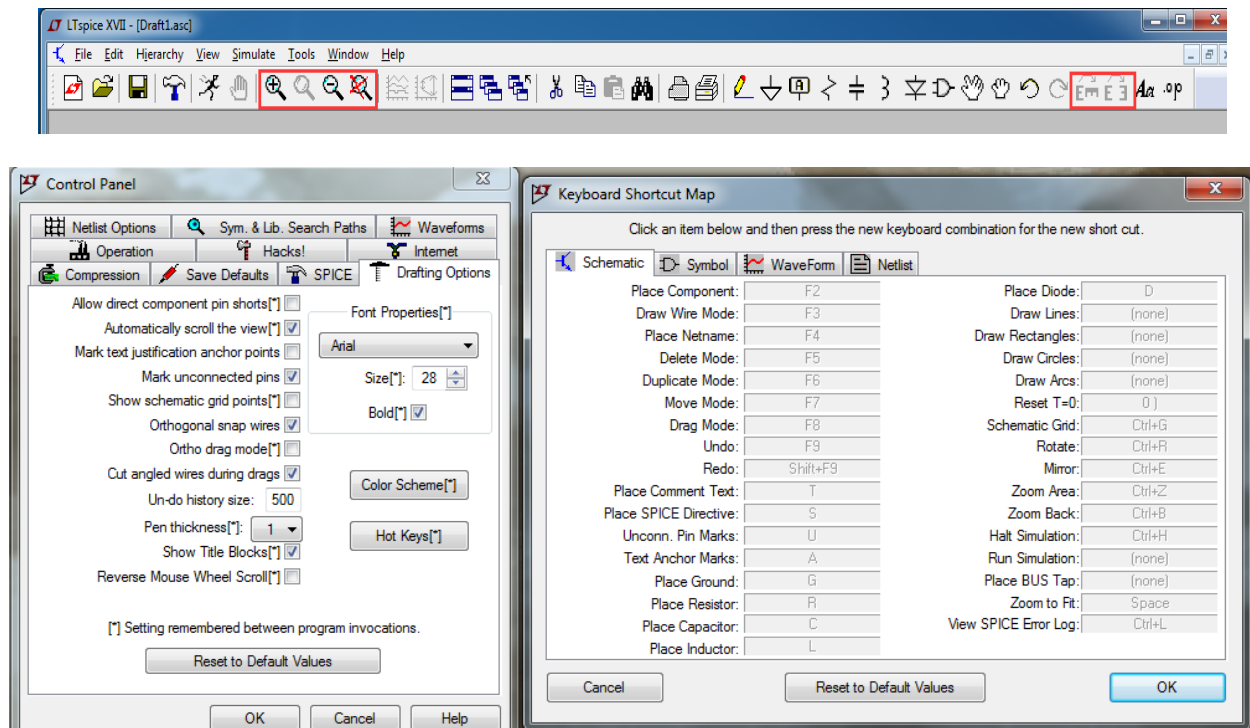
Step 3: Add/Modify/Delete electrical components into the schematic.



- Shortcut keys and buttons; zoom in/out or rotate components if needed. Also available to view in “Control Panel”.

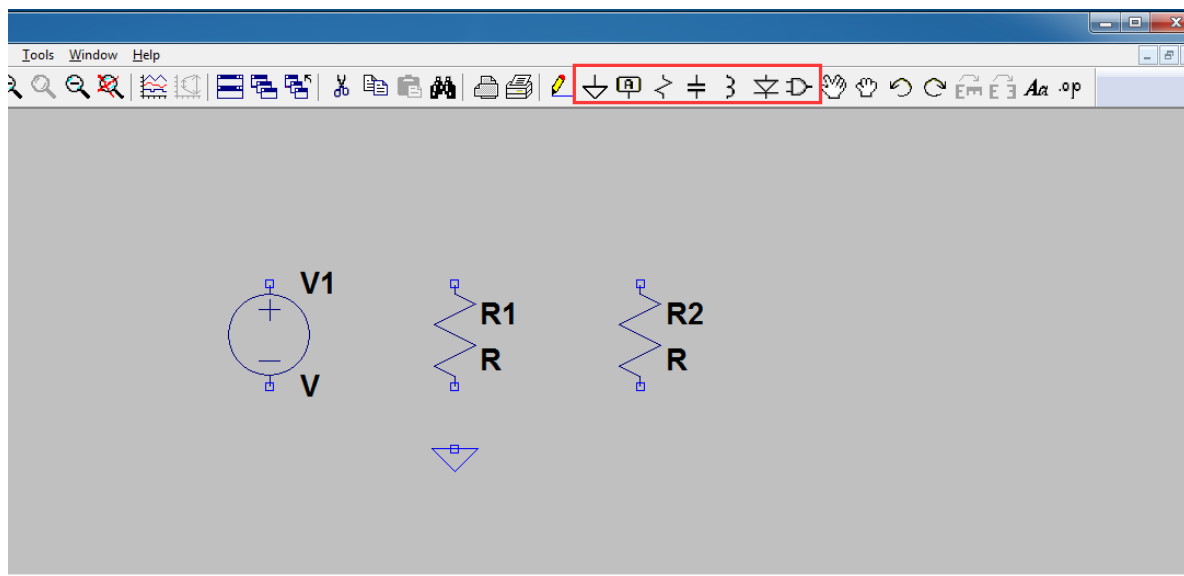
<F5>	delete
<F6>	copy
<F7>	move component without wires attached
<F8>	move components with wires attached
<F9>	undo
shift<F9>	redo
<CTRL R>	rotates component (once it has been selected using <F7>)
<CTRL E>	mirrors component (once it has been selected using <F7>)



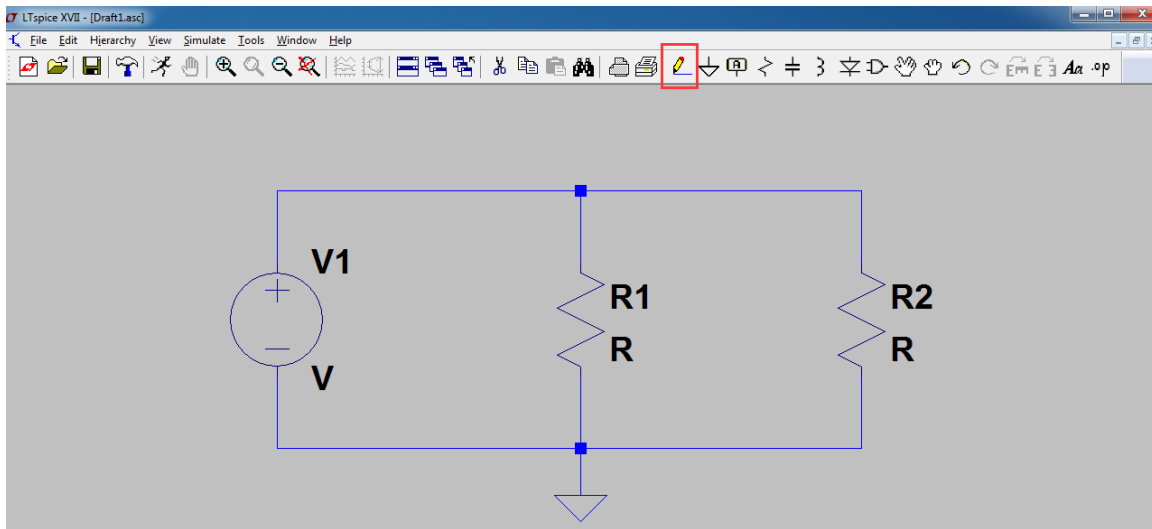


Step 4: Add components from the library, and place them at proper positions.

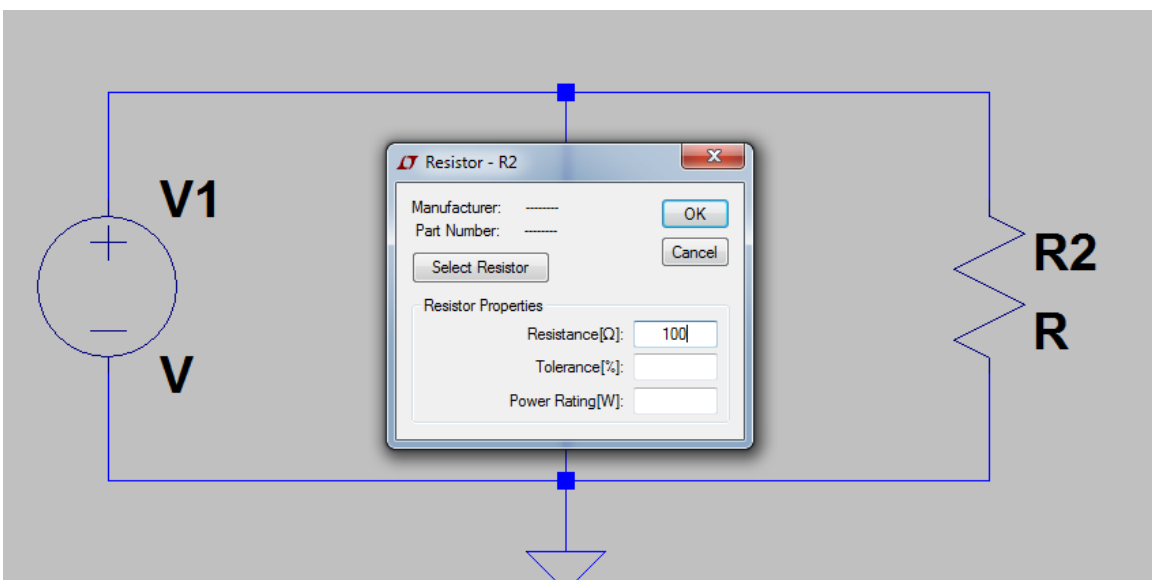
- Components will be placed where you left click the mouse, until you press “ESC” to stop placing that component.



Step 5: Add wires (click and release) to connect all components; fulfill the circuit/schematic.

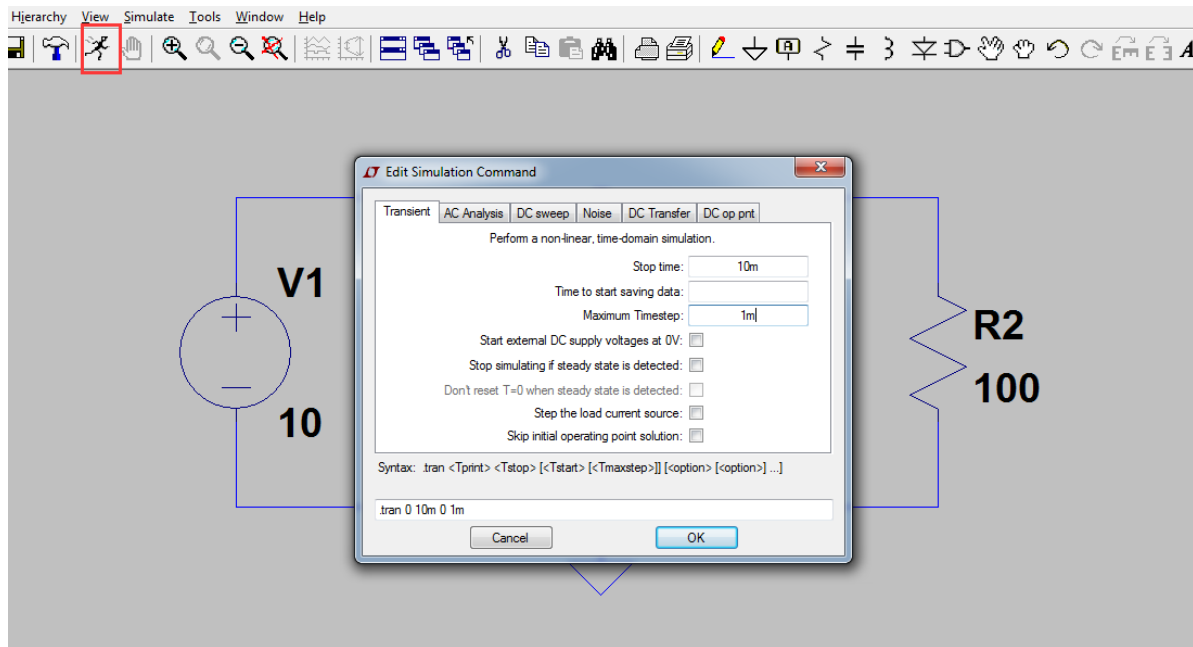


Step 6: Right click on the components, and change their values.

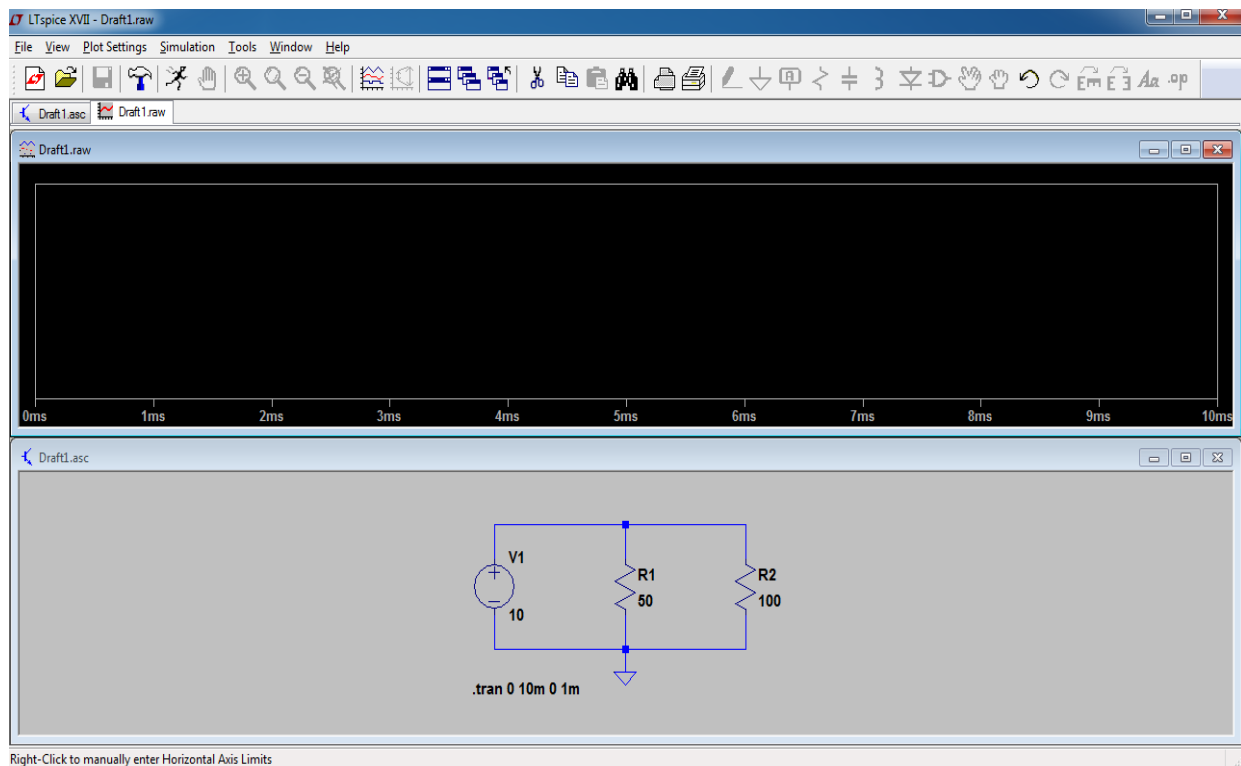


Step 7: “Run” the simulation.

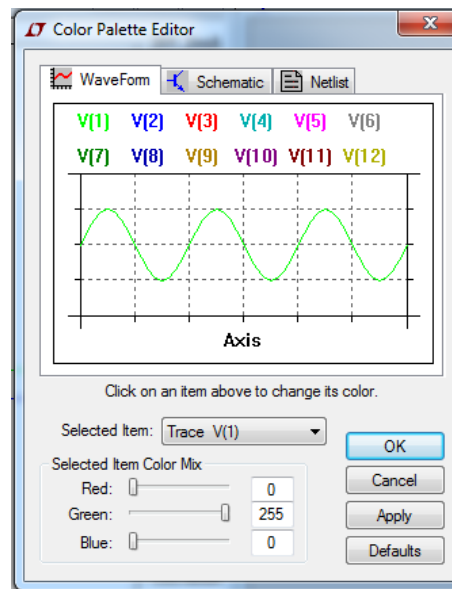
- Fill in the simulation duration (stop time) and precision (timestep). Simulation command will be generated automatically. You can modify the command after you learn more about LTspice or other SPICE.



- Click “OK” to run. The window arrangement of the trace panel (Draft1.raw) and the schematic panel (Draft1.asc) can be adjusted.

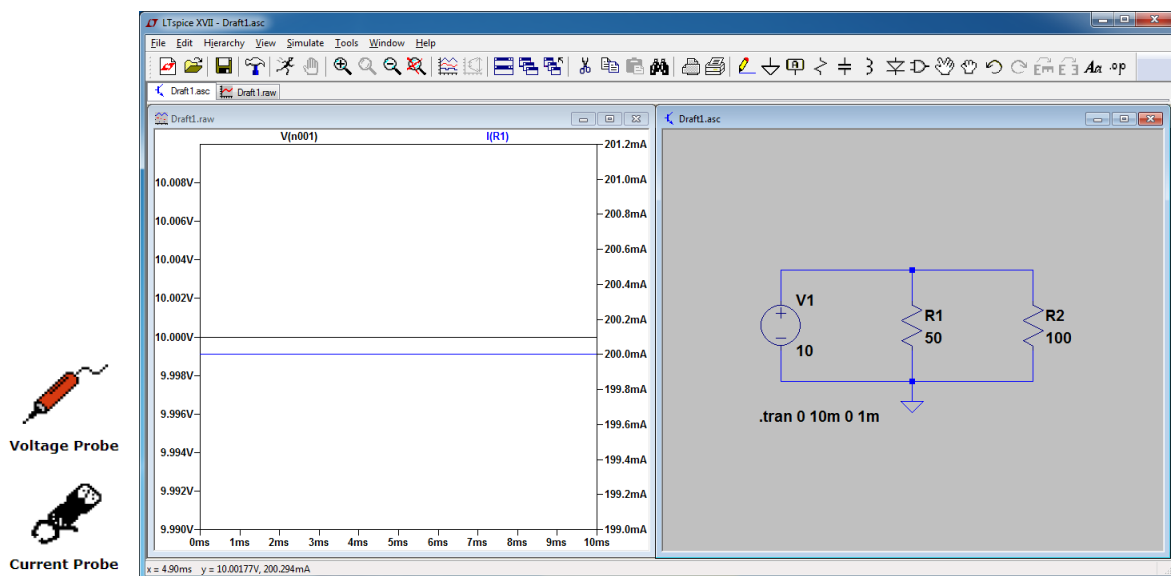


- Color of trace panel can be adjusted in “Tools”, “Color Panels”, “WaveForm” tab, “background” in pulldown menu.



Step 8: View the output

- Left click to activate (bring front) the schematic panel. Place the cursor on the node/wire (voltage) or the element (current), and left click to probe and display the measured values.



- 2) Right click on the trace panel, select “Add Traces”, and left click on the signals you want to see. Click “OK” to display.

