Electrical Circuits LAB 1 Online – Introduction to Computer Aided Analysis and Design

Objectives
Introduce circuit analysis and simulation using computer aided analysis tool, LTspice.

Preparation
Simulation Program with Integrated Circuit Emphasis (SPICE) is an open source mixed analog and digital behavioral simulator. First version of SPICE was introduced in 1973 and the open source version was released in 1993 which continues to be evolved.

SPICE is text based which means input and output is text-based. A large number of products have been developed to provide SPICE capability with a graphics user interface to make it easier to learn and use SPICE .

In this and future labs, we will be using LTspice XVII developed by Analog Devices' Linear Technology group. LTspice supports all three operating systems: MS Windows, Apple Mac and Linux (with Wine Windows compatibility layer).

LTspice provides schematic capture, enhanced SPICE analog circuit simulator, and waveform support. LTspice contains thousands of models for resistors, capacitors, inductors, transistors, diodes, operational amplifiers, etc.
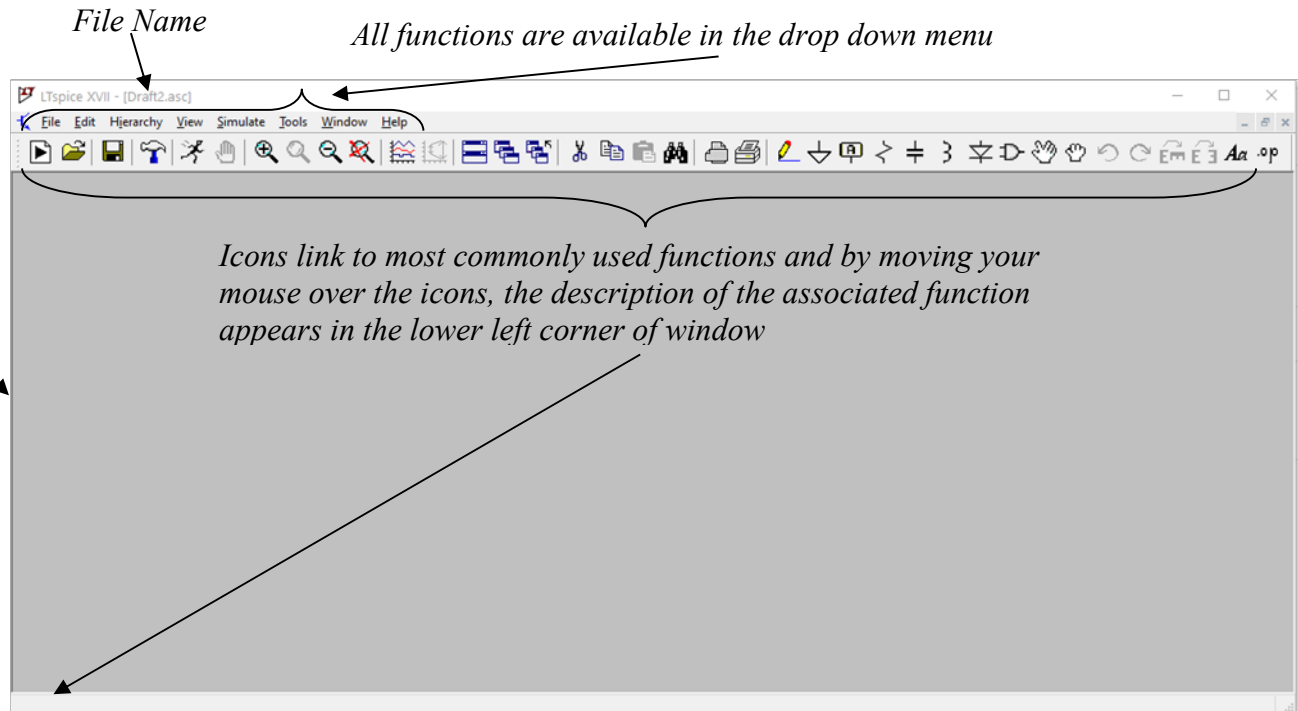
Complete the following steps before starting to work on the experiments in this lab:

1) Read textbook, watch instructional videos and complete assignment in Chapter 1 & 2
2) Introduction to LTspice video:
    a. Microsoft Windows users watch https://youtu.be/L1MONxRj9g4
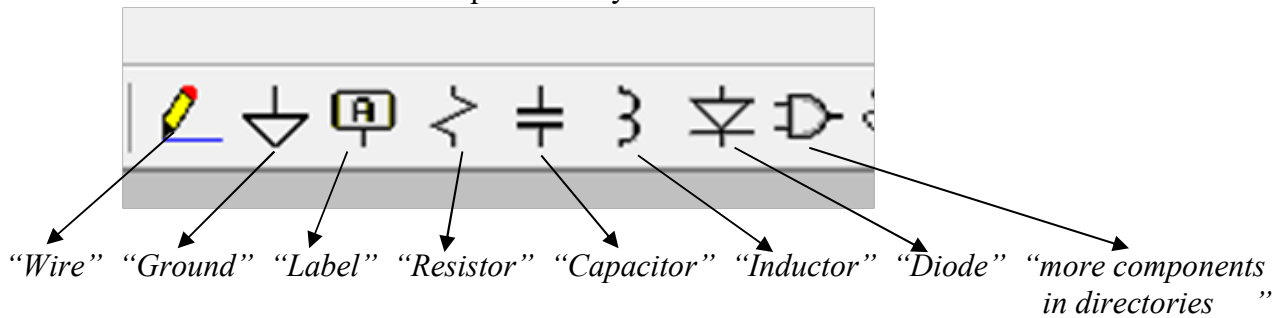   b. Apple Mac users watch the first 10 minutes of https://youtu.be/6AA4YBtqhwE

Experiment 1
This experiment outlines the steps to download LTspice and analyze a simple circuit on MS Windows. You should be able to follow the same instructions for other operating systems, but you may have to make adjustments based on the platform used.

1) Download LTspice from the link http://www.analog.com/LTspice.
   Use default setting for all options.

2) Start LTspice

3) Use menu "File > New Schematic" to open a new schematics page. At this point, the following window should be open:

LTspice XVII - [Draft2.asc]

File   Edit   Hierarchy   View   Simulate   Tools   Window   Help

Icons link to most commonly used functions and by moving your mouse over the icons, the description of the associated function appears in the lower left corner of window

You can use icons to add wire and components to your circuit as shown below:

"Wire"   "Ground"   "Label"   "Resistor"   "Capacitor"   "Inductor"   "Diode"   "more components in directories   "

To place a component, left click on the component icon, drag to desired location and left click to place the component. When you are done, press the <Esc> key or right click to terminate placement.

Once you place a component on the schematics, you can right click on the component to change its designator, value and other attributes in property window. Here is example of specifications available for a 10kΩ resistor symbol and its property window:
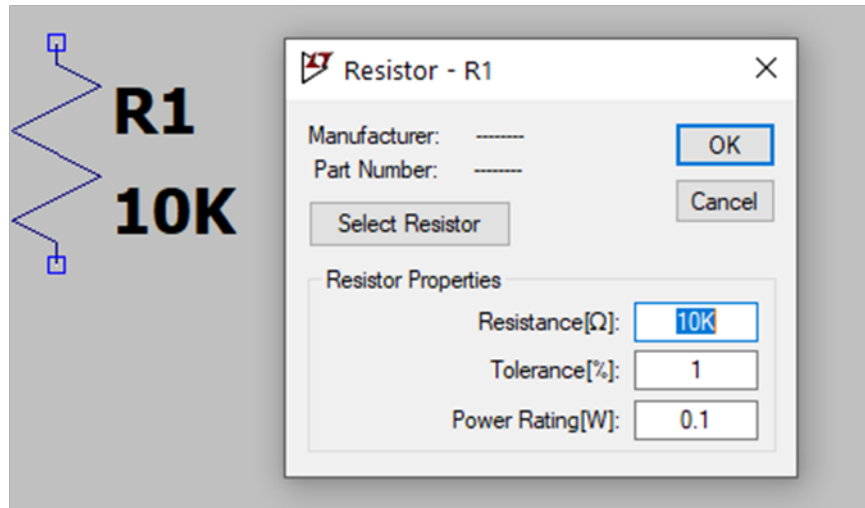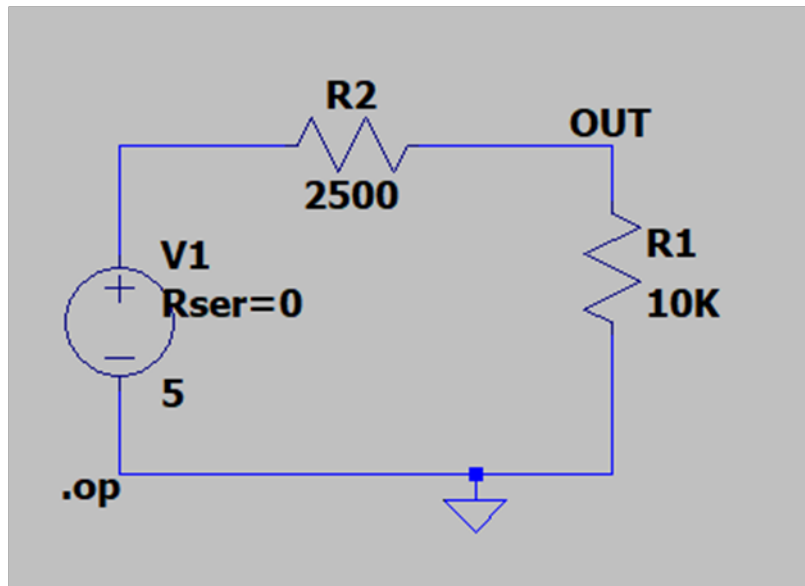
You can use powers of 10 designators in setting the value as list below:

K= k = kilo = 10^3	m= M = milli = 10^{-3}
MEG= meg = 10^6	u= U = micro = 10^{-6}
G= g = giga = 10^9	n= N = nano = 10^{-9}
T= t = terra = 10^{12}	p= P = pico = 10^{-12}
	f= F= femto=10-15

Note, you cannot enter Ω, F or H symbol in the value field. Also need to add **ground connection** to your circuit in order to provide reference voltage (V=0).

4) Enter the following circuit:

The functionality associated with icons in the upper right side may be useful during drawing of the schematics.

Move the
selection

Drag the selection but
keep the connections

Undo last edit

Redo last edit

Mirror components
during insertion

Adding text
comments

Rotates components
during insertion
Short Cut: <Ctl> R
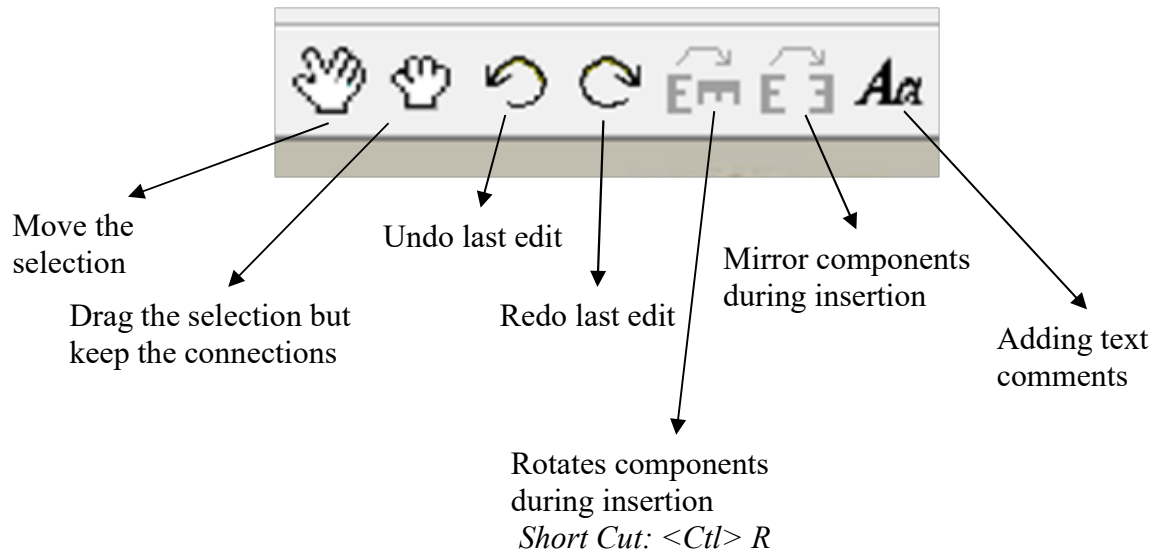
Scissor Icon is helpful in deleting wires and components.

5) Simulate the circuit to find the current of voltage at various points in the circuit. Click on simulation icon to start the simulation and then select "DC Operating Point" simulation tab to complete the simulation. DC OP simulation is used for DC single value electrical circuit analysis.

1) Click to start simulation   *2) Select DC Operating Point*
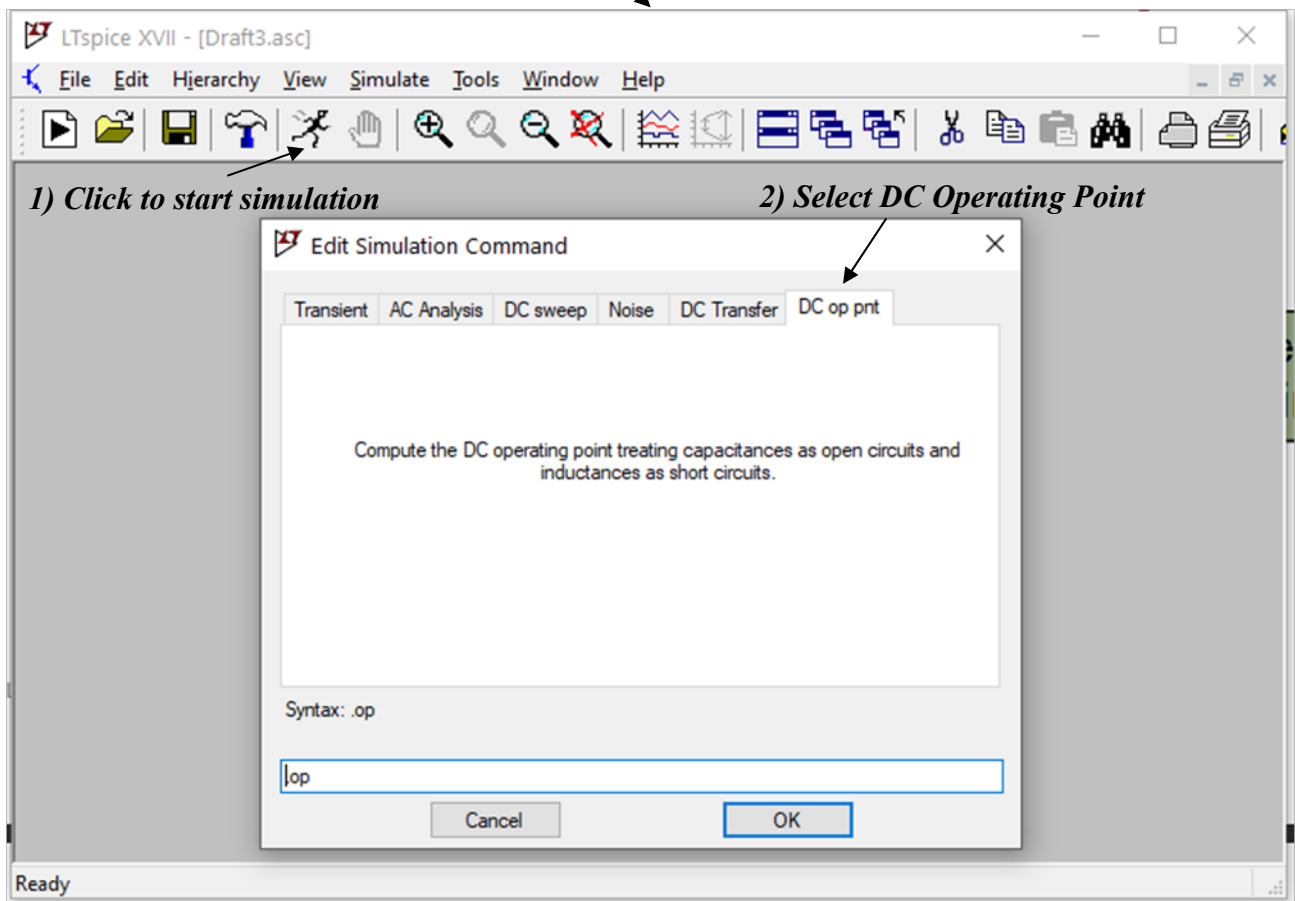
Upon simulation completion, the results of simulation will be displayed in a new window.

```
*  C:\Users\izad\Documents\LTspiceXVII\Draft3.asc                                    ×
            --- Operating Point ---

V(out):          4               voltage
V(n001):         5               voltage
I(R2):           -0.0004         device_current
I(R1):           0.0004          device_current
I(V1):           -0.0004         device_current
```

Simulation output list voltages for every node with respect to ground. You can add label to any node your want to show up on the list with a specific label.

Simulation output lists current through every component. Negative current is in reference to assumed + terminal of the component.

Additionally, you can close the above window and view current, voltage and power for specific node or component by one the following methods:

1) *Current and Power - Moving the cursor over a component causes the current and power to be displayed at the bottom left corner of the LTspice window.*

2) *Voltage with respect to ground- Moving the cursor over a node displays the voltage at that node with respect to ground.*

3) *Voltage Across a Component - left click on one terminal of a component and drag the mouse to the other terminal to display the voltage across the component. The first terminal will be designated as positive terminal.*

Experiment 2

Use LTspice to analyze the following circuit and determine the current flowing through R4 and voltage across R1:
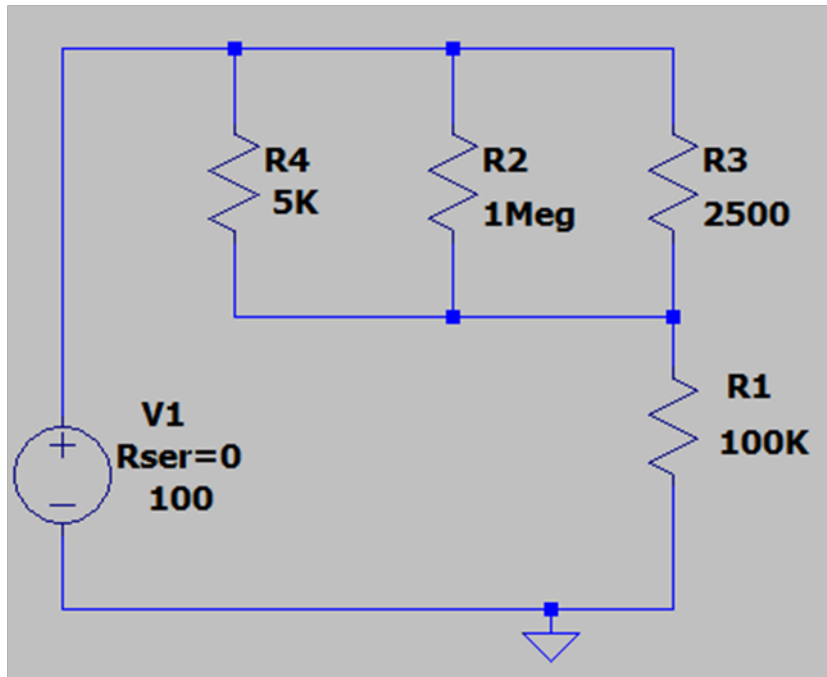
Experiment 3

Use LTspice to determine the R4 value that results in maximum voltage magnitude across R1 for the circuit in experiment 2. You are expected to simulate the circuit with three or more R4 values and use the simulation results to support your answer.

Experiment 4

Use LTspice to determine the R4 value that results in maximum current magnitude through R1 for the circuit in experiment 2. You are expected to simulate the circuit with three or more R4 values and use the simulation results to support your answer.

Report Requirements

This lab and associated report must be completed individually. All reports must be computer printed (Formulas and Diagrams may be hand drawn) and at minimum:

For each experiment include:
- Clear problem statement in your words.
- Answer to any specific experiment questions (if any)
- Identify the theory or process and associated calculations
- Documents resulting circuit schematics from LTspice, simulation output and additional tables, timing diagram or chart required by the experiment.

For the whole report include:
- A Cover page with your name, class, lab and completion date.
- A Lessons Learned section which summarizes your learning from this lab in 5 sentences or more.
- A New Experiment section that has description of a new experiment and the experiment's results. Experiment should be related to material covered in class but not simply variation of the existing lab experiments.