Circuit and Electronics

LAB REPORT 1

EMMANUEL NIMO | 59112019 | LECTURER: DR. NATHAN AMANKWAH.

INTRODUCTION

This is a circuit lab report on the use of LT spice to design circuit upon a given instruction. The LT spice is a high performance software stimulator which allows easy evaluation of circuit designs. The content of this report was experimented, analyzed and studied in Ashesi University College.

AIM OF EXPERIMENT

The aim of the experiment is to make a circuit using a mesh of resistors and other element of a circuit using the LT spice and evaluate the circuit by using both LT spice and Matlab or calculator.

METHODOLOGY

The following steps were undertaken to achieve the aim of this experiment.

1. The LT spice software is launched and a new file is selected.
2. The components such as resistors, voltage source, current source are selected current controlled current source (from netlist) are then selected and pasted on the work space.
3. The pencil tool is used to draw a line to connect all the components as required by the instructions given.
4. The operating point (.op) is then used to evaluate the circuit drawn. Matlab or calculator is also used to evaluate the circuit as well.

CIRCUIT DIAGRAM

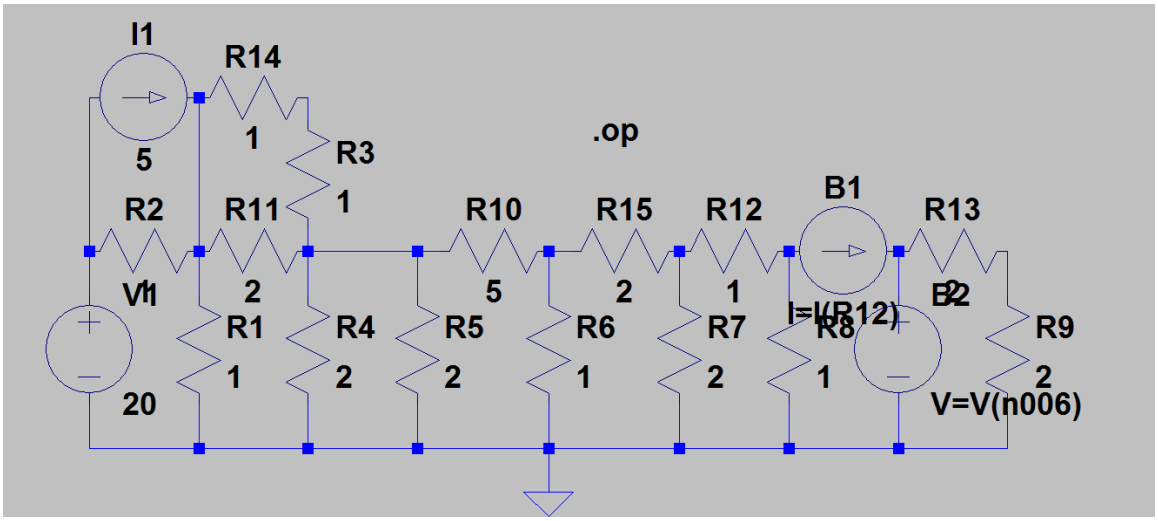

Figure 1. The diagram above shows the LT Spice circuit design of ten meshes consisting of electrical elements.

RESULT FORM OPERATING POINT (.op)

```
        --- Operating Point ---

V(n001):        7.18563         voltage
V(n004):        4.52844         voltage
V(n003):        9.84281         voltage
V(n005):        0.598805        voltage
V(n006):        0.224559        voltage
V(n007):        0.149716        voltage
V(n009):        0.11228         voltage
V(n008):        0.224559        voltage
V(n002):        20              voltage
I(B2):          -0.131013       device_current
I(B1):          -0.074843       device_current
I(I1):          5               device_current
I(R2):          -10.1572        device_current
I(R15):         -0.187123       device_current
I(R11):         -2.65719        device_current
I(R14):         -2.65719        device_current
I(R13):         -0.0561399      device_current
I(R12):         -0.074843       device_current
I(R10):         -0.785928       device_current
I(R9):          0.0561399       device_current
I(R8):          0.149716        device_current
I(R7):          0.11228         device_current
I(R6):          0.598805        device_current
I(R5):          2.26422         device_current
```

```
I(R4):          2.26422       device_current
I(R1):          9.84281       device_current
I(R3):          2.65719       device_current
I(V1):          -15.1572      device_current
```

CALCULATIONS FROM A T1-84

The node voltage method was applied in solving for the voltages. This is because after using Kirchhoff's method, the voltages and currents obtained were far more different from the one obtained in the LT Spice software.

$$2.5V_1 - V_2 - 0.5V_3 = 20 \ldots\ldots\ldots (1)$$

$$-V_1 + 3V_2 - 0.5V_3 = 10 \ldots\ldots (2)$$

$$-0.5V_1 - V_2 + 2V_3 = 0 \ldots\ldots (3)$$

$$-V_3 + 1.7V_4 - 0.2V_5 = 0 \ldots\ldots (4)$$

$$-0.2V_4 + 1.7V_5 - 0.5V_6 = 0 \ldots\ldots (5)$$

$$+0.5V_5 + 2V_6 - V_7 = 0 \ldots\ldots (6)$$

$$-V_6 + 3V_7 - V_8 = 0 \ldots\ldots (7)$$

$$-V_1 + 1.25V_8 = 20 \ldots\ldots (8)$$

Solutions to the equations above

$V_1 = 13.17V$	$V_5 = 0.95V$
$V_2 = 9.024V$	$V_6 = 1.380V$
$V_3 = 7.80V$	$V_7 = 2.29V$
$V_4 = 4.70V$	$V_8 = 5.46V$

With regard to the table above, voltages such as 9.024V, 7.08Vand 4.70V are precise with the voltages obtained for the V(n003), V(n001) and V(n004) respectively.

CONCLUSION AND LESSON LEARNED.

The LT Spice is actually a very good tool for easier and faster evaluation of circuit. Solving the circuit manually using Matlab or calculator requires a lot of time and hard work and as there can be errors without you knowing. Again, it is always advisable that one uses the best possible method in soling for the voltages and currents of a circuit. It is also important and very insightful when you manually arrive at the same answers that the LT spice provides. It increases one's confidence level and hone your skills in circuit design calculations.

PART 2

AIM OF EXPERIMENT

The aim of the experiment is to construct an electronic circuit on a breadboard with at least five meshes and ten resistors and find the voltages and current across and through some of the resistors respectively.

METHODOLOGY

To achieve this task, the following procedure was considered;

1. The diagram of the circuit design was drawn in the LT Spice software and ran on a laptop and the reading for the current and voltages were taken.
2. Ten resistors with known resistors were taken from the analog kit and were connected in series and parallel to form seven meshes.
3. A side of the first mesh was connected to a voltage source and that was source provided by the waveform software.
4. The circuit was also grounded to serve as a reference point in the electrical **circuit** from which voltages are measured.
5. The voltmeter was then used to measure the voltages and current across and through the resistors respectively.
6. The result obtained were then analyzed and compared to the data produced by the LT Spice software.

APPARATUS USED

Resistors, breadboard, conducting wires, analog discovery waveform, LT Spice software, laptop and a voltmeter.

DIAGRAM OF THE LT SPICE CIRCUIT DESIGN

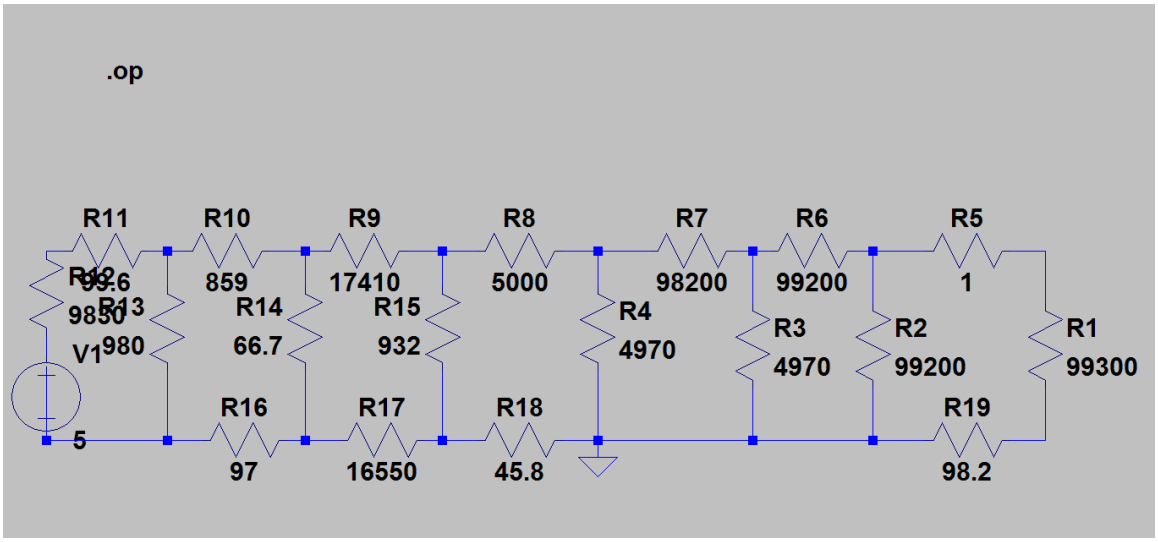

Figure 2. The diagram above shows the circuit designed using LT spice software

RESULT AND ANALYSIS.

The following are the results obtained from the LT spice run command.

```
        --- Operating Point ---

V(n008):        2.88027e-006   voltage
V(n013):        2.84555e-009   voltage
V(n007):        2.8803e-006    voltage
V(n006):        8.63513e-006   voltage
V(n005):        0.00018495     voltage
V(n004):        0.000379993    voltage
V(n003):        0.00819087     voltage
V(n002):        0.20971        voltage
V(n001):        0.257457       voltage
V(n010):        -0.0301828     voltage
V(n011):        -0.00742683    voltage
V(n012):        -1.7866        voltage
V(n009):        4.96982        voltage
I(R12):         -0.000479386   device current
I(R19):         -2.89771e-011  device current
I(R18):         -3.90087e-008  device_current
I(R17):         -4.48643e-007  device_current
```

```
I(R16):         -0.000234597  device_current
I(R15):         -4.09635e-007 device_current
I(R14):         -0.000234149  device_current
I(R13):         -0.000244788  device_current
I(R11):         -0.000479386  device_current
I(R10):         -0.000234597  device_current
I(R9):          -4.48643e-007 device_current
I(R8):          -3.90087e-008 device_current
I(R7):          -1.79546e-009 device_current
I(R6):          -5.80124e-011 device_current
I(R5):          -2.89771e-011 device_current
I(R4):          3.72132e-008  device_current
I(R3):          1.73745e-009  device_current
I(R2):          2.90353e-011  device_current
I(R1):          2.89771e-011  device_current
I(V1):          -0.000479386  device_current
```

COMPARISON WITH VOLTMETER

The voltage across R14 (R14 N011 N003), R15(R15 N012 N004) AND R10 (R10 N003 N002) were 0.008V, -1.78 and -0.007V respectively. The other voltages that had -0.000 and +0.000 were assumed to have had a lot of decimal places, hence the voltmeter could not display.

Finding the current with the voltmeter was very tedious because we encountered a lot of -0.000 and +0.000 so we had to use the Ohms' Law. The voltages obtained were divided by the value of the resistors.

Calculation:

$$I_{R14} = \frac{0.008V}{4970\Omega} = 1.61\ x\ 10^{-6}A = 1.6\mu A$$

$$I_{R15} = \frac{-1.7866V}{99200\Omega} = -1.80\ x\ 10^{-5}A = -18\mu A$$

$$I_{R10} = \frac{-0.007V}{5000\Omega} = 1.4\ x\ 10^{-6}A = 1.4\mu A$$

CONCLUSION AND LESSON LEARNED

With regard to the information gathered and compared, the LT spice provides a convenient way to evaluate a circuit, but can also be detrimental if not experimented physically with the circuit component. I obtained similar values for the voltages and currents of some resistors. I also learned that one needs to be careful and watchful when using the voltmeter to measure current and voltage of the resistors The tip of the voltmeter pins should touch either the holes in which the resistor enter the breadboard or touch the metal pin of the resistor connected. One should connect the red and black pin of the voltmeter to the Voltage(V) and COM respectively. For current, the Current(A) and COM are required. I think the tests did go smoothly and I had no problems, except for the fact that the voltmeter could not display all the values for the voltages and current due to its four-digit display. It was a very insightful experiment done by me and much has been learned.