

**EAST
WEST
UNIVERSITY**



Department of CSE

CSE209 Lab

Course Name: Electrical Circuits

Course Code: CSE209

Section No: 2

Experiment No: 07

Name of the Experiment: DC Circuit Analysis in PSpice using Source and Resistance Sweep.

Date of submission: 12/09/21

Student's Name: Md Abdul Ahad Rifat

Student's ID: 2020-1-60-215

Submitted to

Rashedul Amin Tuhin

Senior Lecturer

Department of Computer Science and Engineering

East West University

Objectives:

1. To analyze DC circuit in PSpice by sweeping source and resistance.
2. To verify maximum power transfer theorem.

Circuit Diagram(s):

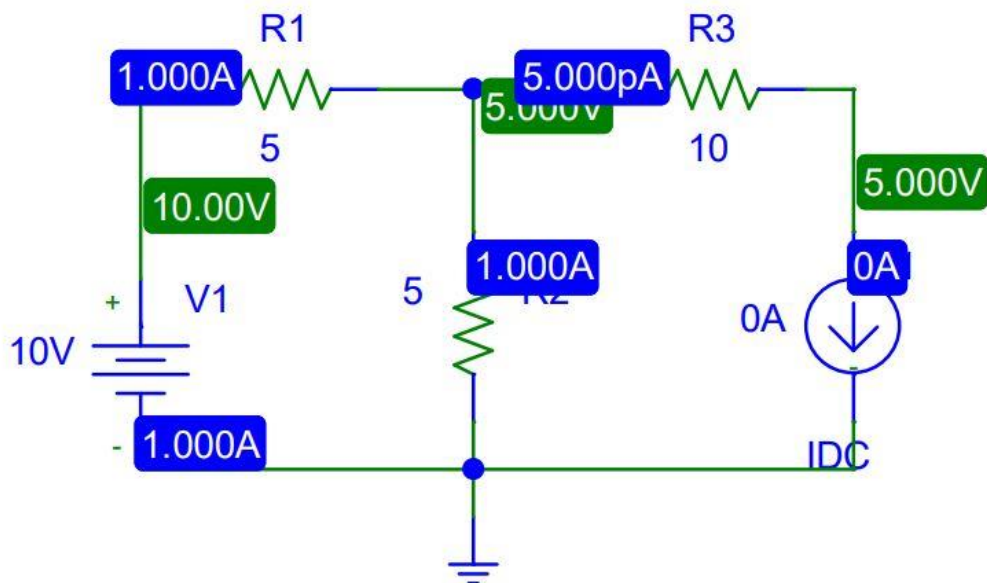


Figure 1.PSpice Schematic diagram for circuit 1

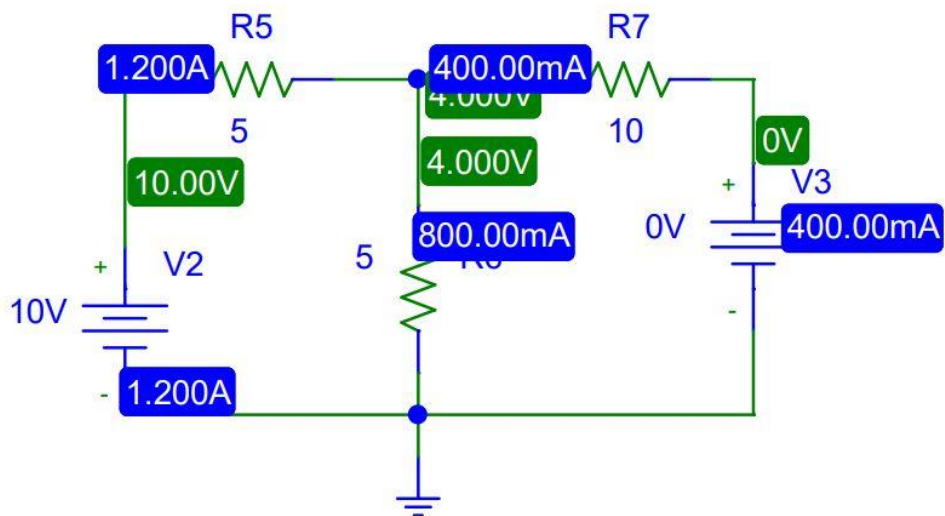


Figure 2.PSpice Schematic diagram for circuit 2

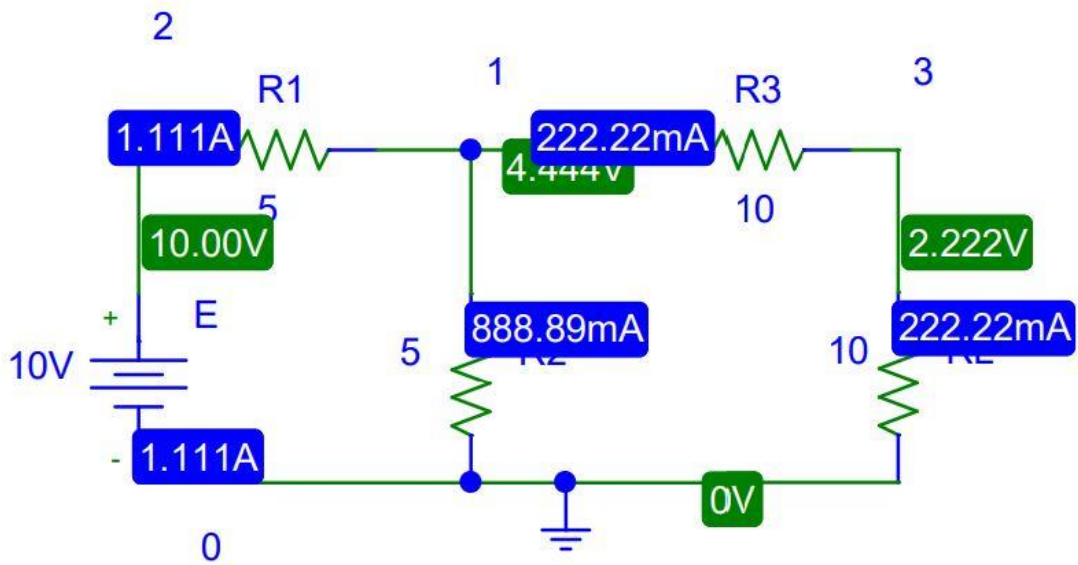


Figure 3.PSpice Schematic diagram for circuit 3

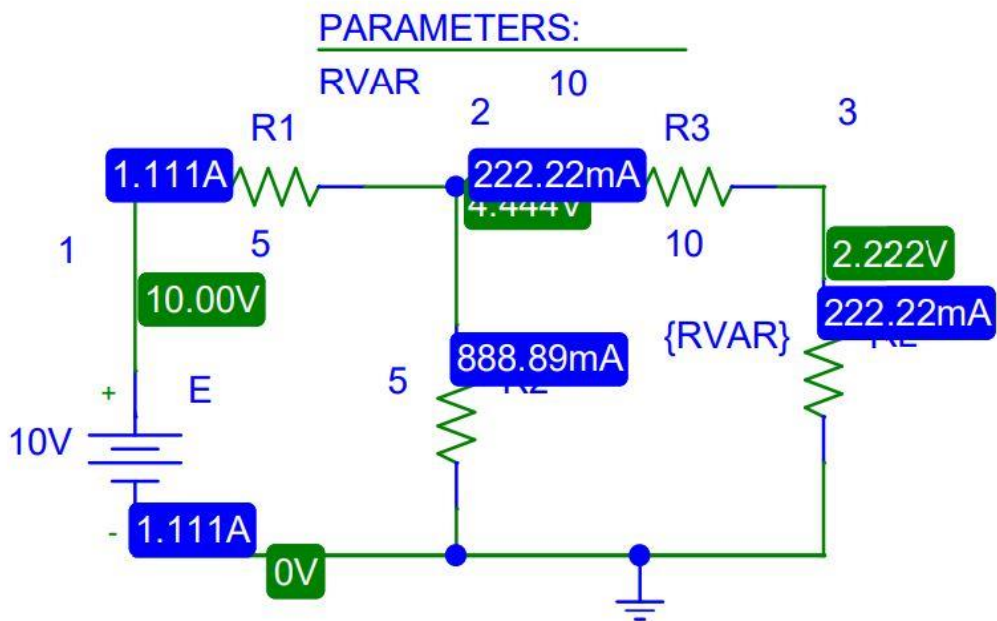


Figure 4.PSpice Schematic diagram for circuit 4

Graph:

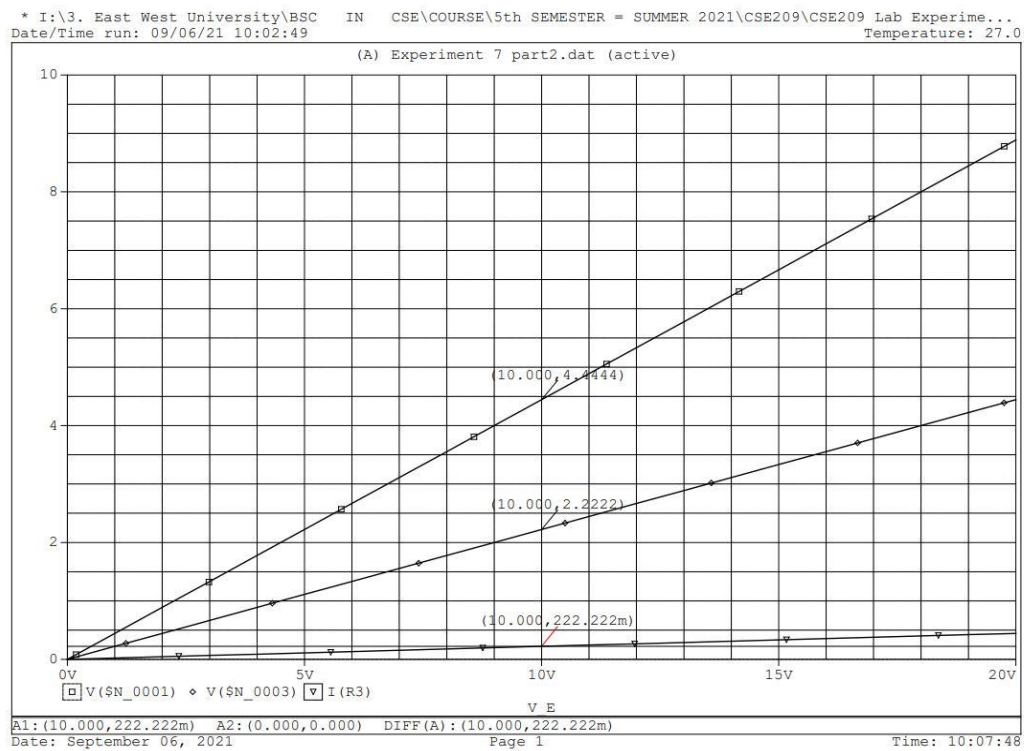


Figure 5. Voltage and Current characteristic graph using DC Sweep

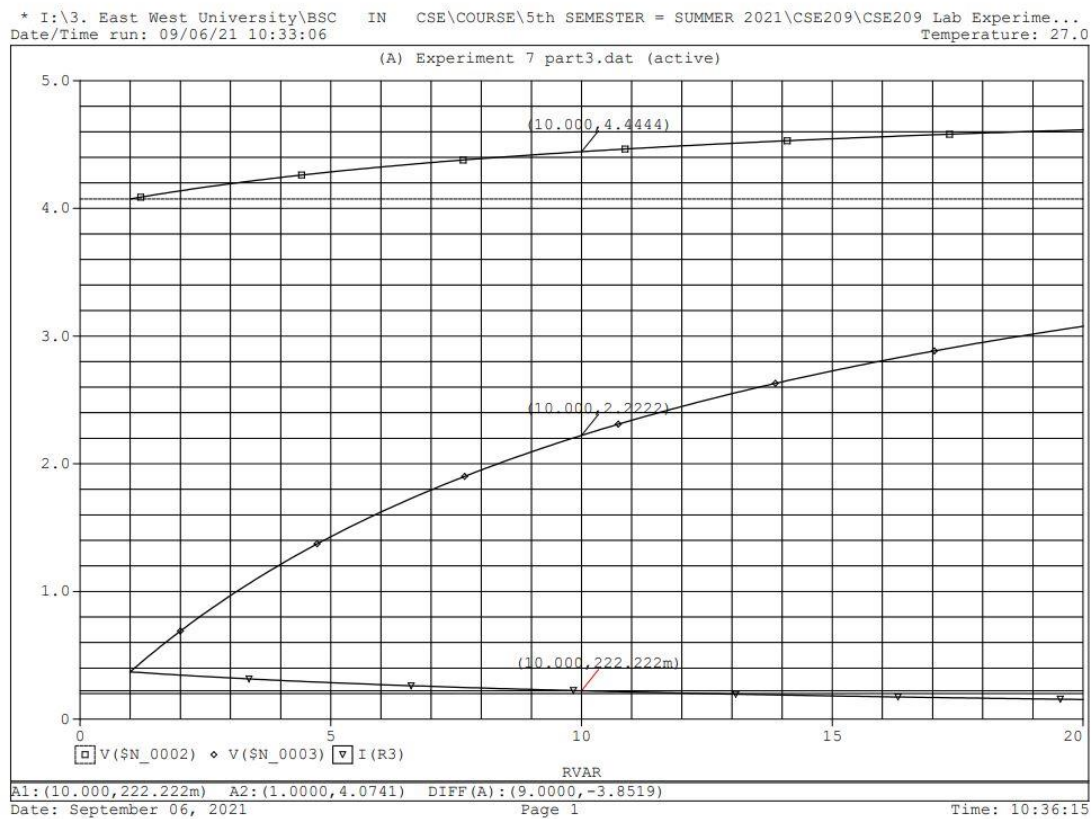


Figure 6. Voltage and Current characteristic graph using Resistance Sweep

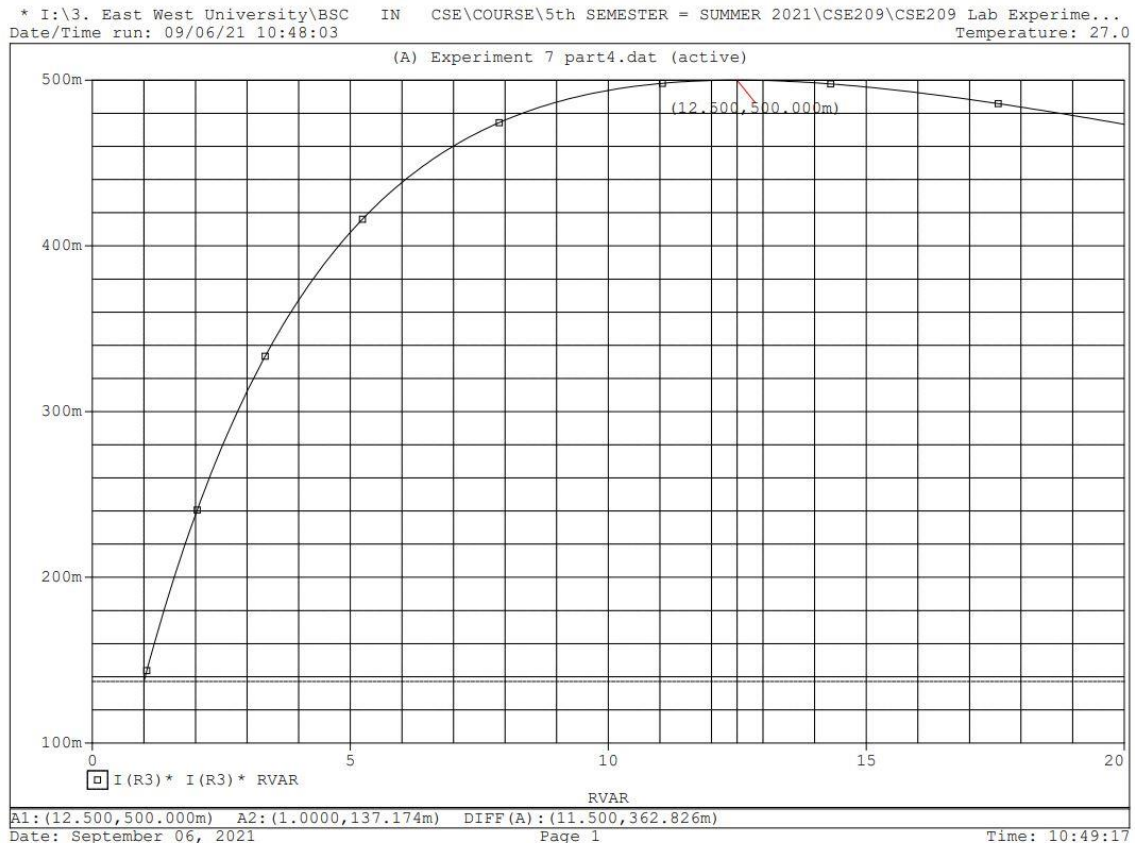


Figure 7. Graph 3 for the maximum power transfer

Post-Lab Report Questions and Answers:

1. Compare the values of V (1), V (2) and I (R3) obtained in steps 4 and 5(d).

Answer:

There has been no change between the values of V (1), V (2) and I (R3) obtained in steps 4 and 5(d). Both of the steps the value are the same. If we see figure 5 and 6 the graph shows us V (1), V (2) and I (R3) value are the same.

$$V(1) = 4.4444V$$

$$V(2) = 2.2222V$$

$$I(R3) = 222.22mA$$

2. Compare the load resistance R_L for maximum power transfer obtained in steps 3 and 5(e).

Answer:

We know, $R_{th} = R_L$ if the power is maximum

In the steps 3,

In figure 1,

$$E_{th} = V_{oc} = 5.000V$$

In figure 2,

$$I_{SC} = 400mA$$

$$\begin{aligned} \text{So, } R_{th} &= \frac{V_{oc}}{I_{SC}} \\ &= \frac{5V}{400mA} \end{aligned}$$

$$R_{th} = 12.5\Omega$$

In the steps 5(e),

We see in the figure 7 the power is maximum when $R_L = 12.5\Omega$

3. Compare the theoretical solutions with the solutions obtained from PSpice and comment on any observed discrepancy.

Answer:

Theoretical Calculation:

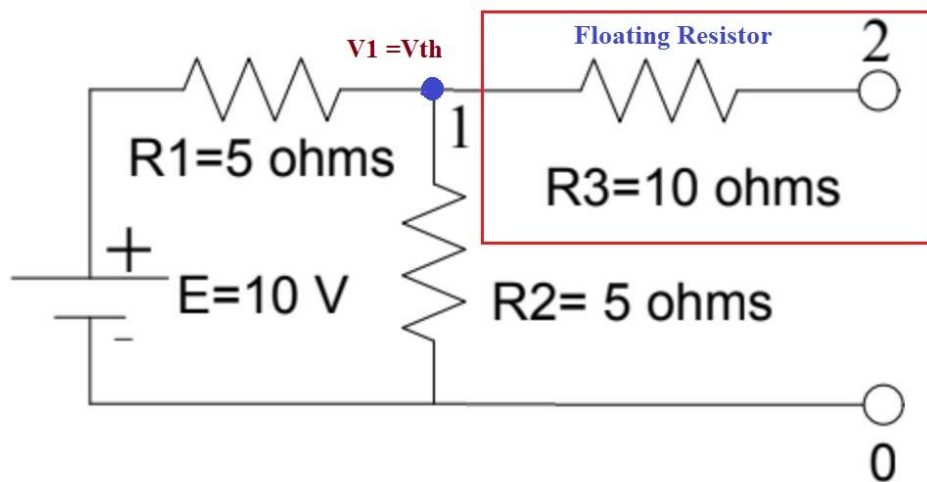


Figure 8. Calculation for V_{th}

Using VDR,

$$V_{th} = \frac{5 \times 10}{5 + 5}$$

$$V_{th} = 50V$$

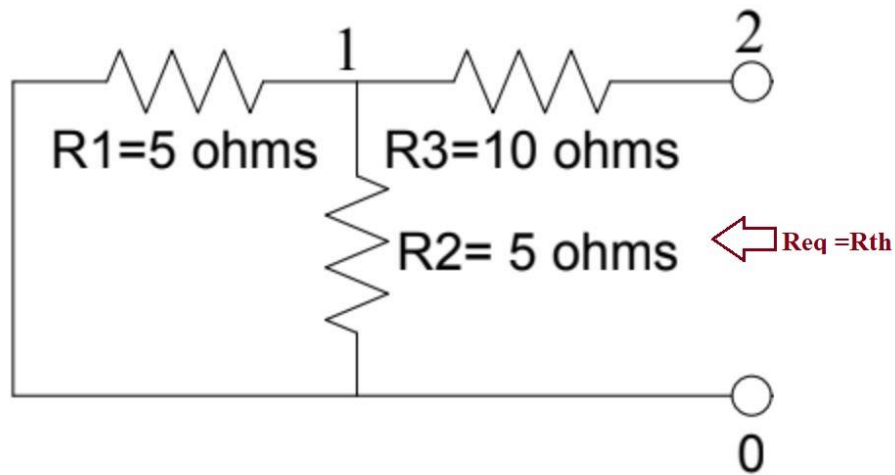


Figure 9. Calculation for Rth

$$R_{th} = [5||5]\Omega + 10\Omega$$

$$R_{th} = 2.5\Omega + 10\Omega = 12.5\Omega$$

$$P_{max} = \frac{5^2}{4 \times 12.5} \text{ watt} = 0.5 \text{ watt} = 500 \text{ mwatt}$$

Table 1. Comparing Theoretical Value and PSpice Simulation Software Value

Name	Theoretical Value	PSpice Simulation Value
R_L	12.5Ω	12.5Ω
P_{max}	500mWatt	500mWatt

There has been no discrepancy between theoretical value and PSpice simulation

Conclusion:

In experiment 7 we analysis DC circuit in PSpice using Source and Resistance Sweep. Also we use Thevenin's theorem to measure R_{th} and maximum power transfer into this circuit. In

this experiment we use one new element which is the parameter to measure load resistance.
Now we know how to measure maximum power in a circuit.