EAST WEST UNIVERSITY

Department of Computer Science and Engineering

Course Title: Electrical Circuits

Course Number: 209

Semester: 4th

Experiment No.: 07

Experiment Title: DC Circuit Analysis in PSpice using source and Resistance Sweep.

Student ID: 2020-3-60-012

Student Name: Sadia Islam Prova

Date of Performance: 23-12-21

Date of Report Submission: 07-01-22

Objectives:

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

Circuit Diagram:

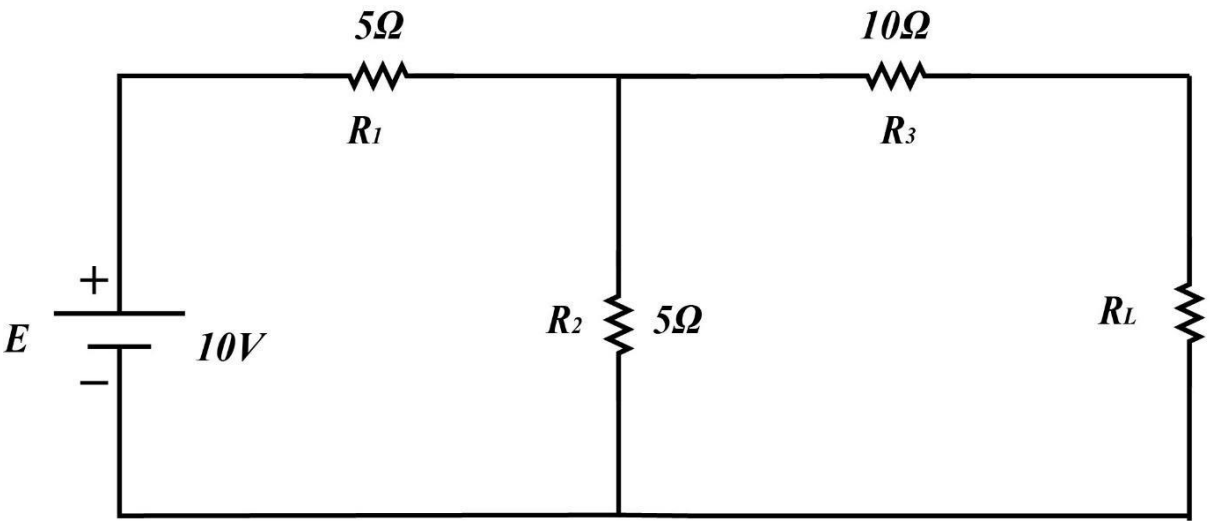

Figure 1: Circuit diagram for experiment

Answer to the post lab report question 01:

From PSpice generated graph we

identified,

V(1) = 4.44 V

V(2)=2.22V

I(R3)=222.22mA

These values are same for the step 4 and step 5(d).

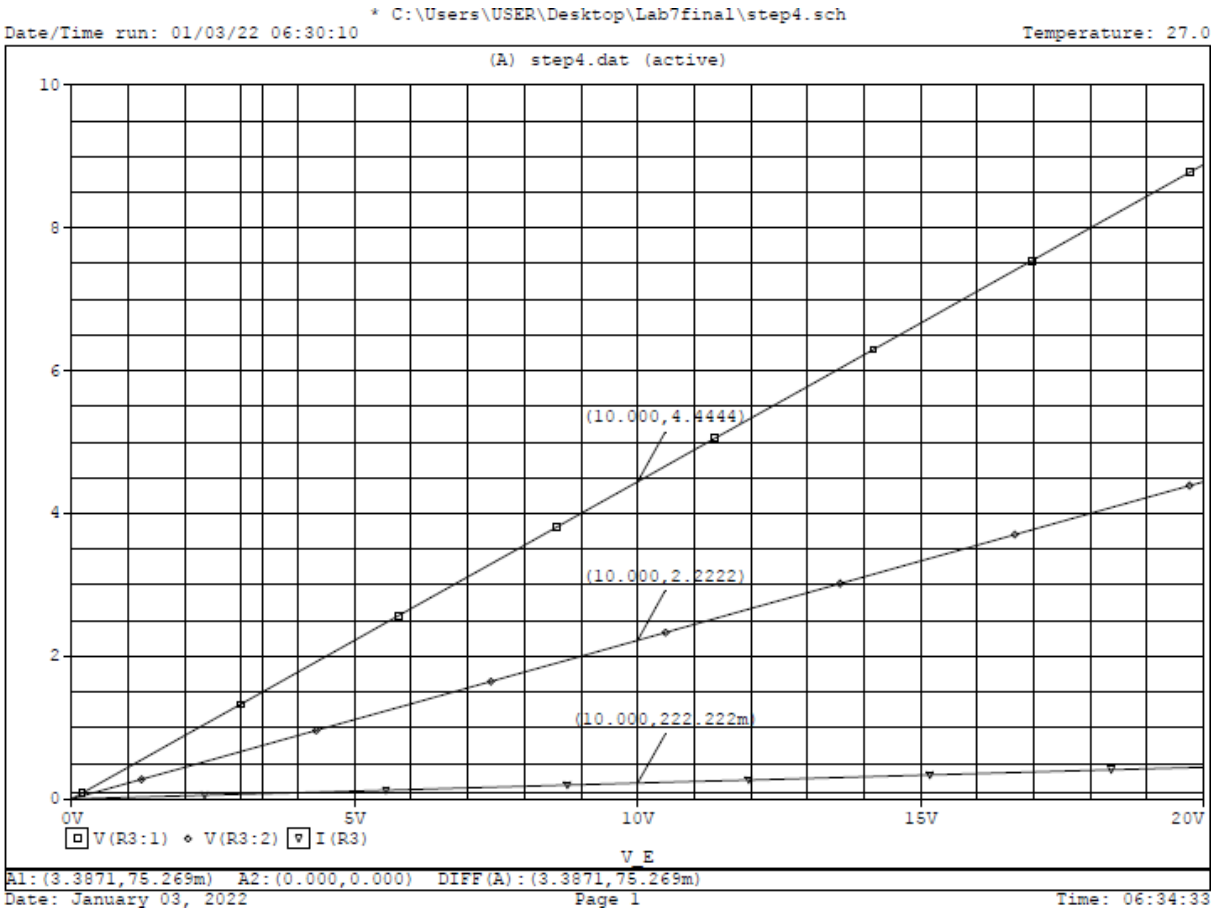

Figure 2: graph for step 4

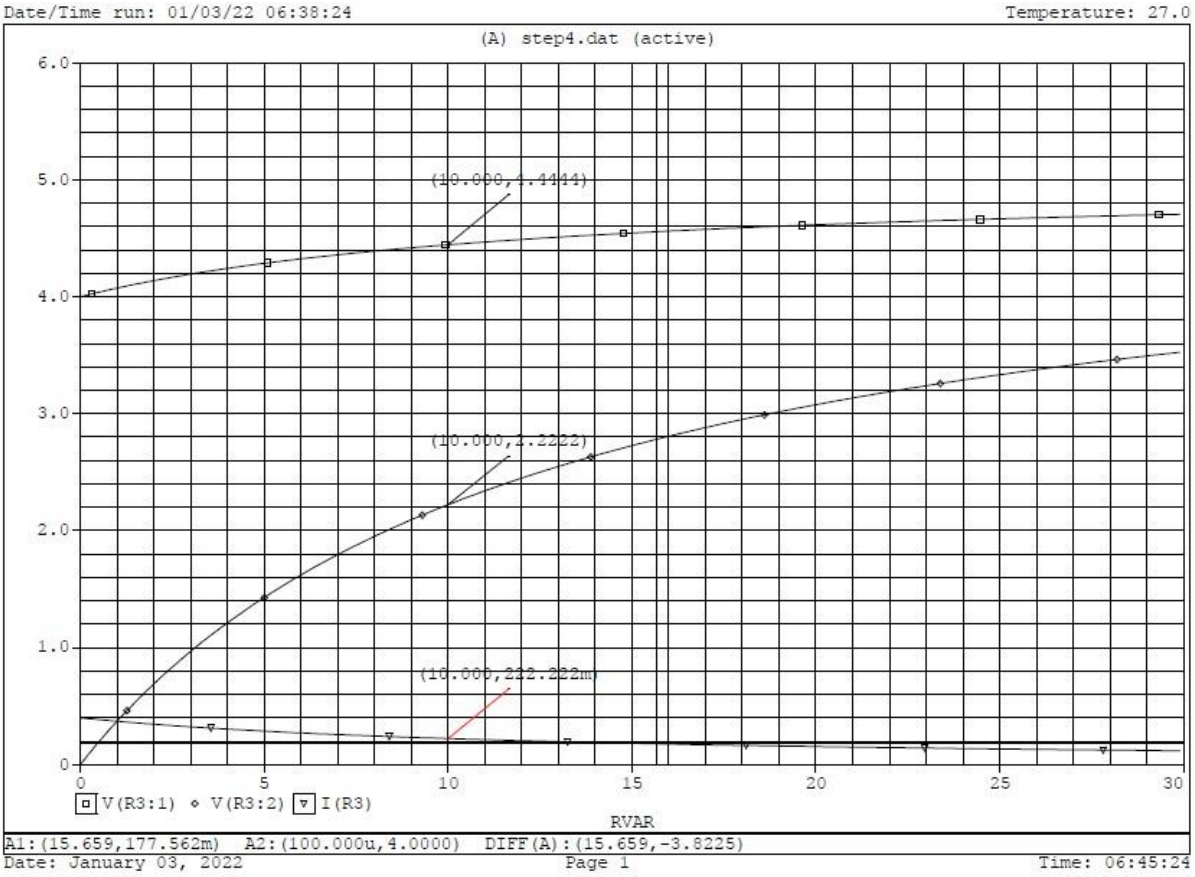

Figure 3: graph for step 5.d

Answer to the post lab report question 02:

From PSpice step 3 simulation we have

(step 3) VOC = 5 V

ISC = 0.4 A

∴ Rth = (5/0.4 Ω)= 12.5 Ω

From step 5(e) PSpice generated graph we have

Maximum power at load resistant 12.5Ω

∴ Rth = RL = 12.5 Ω

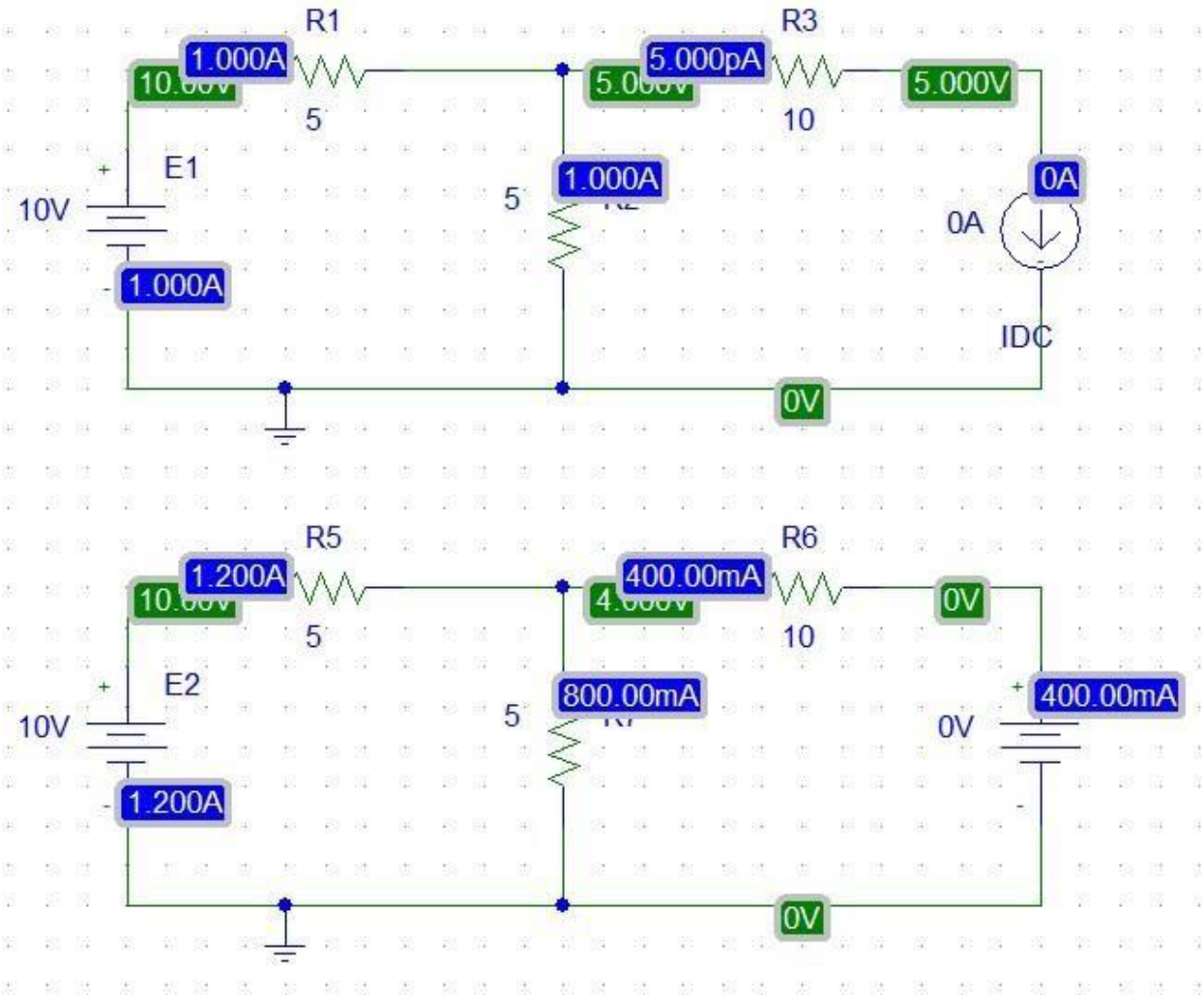

Figure 4: Circuit simulation in PSpice

After comparing the values there is no discrepancy in PSpice.

Conclusion:

In this experiment we verified the maximum power theorem and analyzed DC circuit in PSpice and found no discrepancy with our actual value in PSpice.