EEE 117L Laboratory – Network Analysis
Lab #1: Introduction to PSpice

Lab Day and Time: Wednesday 1:30pm - 4:10pm

Group Number: # 02

Group Members: (Last Name, First Name)

Member #1: Algador, Vigomar Kim

Member #2: Singh, Diljot

Member #3: ____________________

Total Score: /100

Work Breakdown Structure: It is important that every group member do their share of the work in these labs. Remember that you will receive no credit for the lab if you did not contribute. Write in the Table provided below, which group member(s) contributed to the solution of each problem in the lab. Also remember that only on lab worksheet per group will be turned in to me at the beginning of each lab class. If there was any group member that did not contribute, then write their name in the space provided below.

Problem Number	Group member(s) that worked on the problem.
Part IA	Vigomar Kim Algador
Part IB	Vigomar Kim Algador
Part IC	Vigomar Kim Algador
Part IIA	Diljot Singh
Part IIB	Diljot Singh
Part IIC	Diljot Singh
Part III	Diljot Singh
Part IV	Vigomar Kim Algador, Diljot Singh

Absent member(s):____

General Instructions:

1) If you need extra pages in order to finish up the worksheet, make sure you add them in the appropriate section. However, it will be important that the worksheet is formatted properly and is easy to read.

2) DO NOT change the format of the worksheet or delete any of the text provided. The circuit schematics and results of the simulations must be provided in the space indicated.

3) If you need help with the PSpice simulations, refer to the tutorial from Purdue University that was listed in the syllabus:

   https://engineering.purdue.edu/~ee255/lecturesupp_files/PSpice-Tutorial.pdf

4) This semester we will be using the new CSUS Sassafras servers, along with the remote desktop program, in order to connect to the computers on the third floor of Riverside Hall. You will be able to access and use the PSpice circuit simulation program from there. Remember to email your work to yourself since the computers are periodically wiped clean of any progress saved onto them. The link to the Sassafras servers is: https://ecs.csus.edu/news/articles/sassafras.html

5) Make sure to download, install, and run the CSUS VPN. Otherwise you will not be able to connect to the university computers. Use the link below and follow the instructions to do this: https://www.csus.edu/college/engineering-computer-science/computing-services/remote-tools.html#connecting-to-the-csus-vpn-via-the-global-protect-vpn-client

6) Remember that completeness and neatness count towards the grade on these worksheets. Make sure to show and justify all work. Points will be taken off for circuit diagrams that are sloppy or whose simulation results are difficult to see.

7) If you decide to type out equations to help with your explanations, use the built in equation editor in MS Word. Equations must be properly formatted and easy to understand at a glance.

8) Refer to sections 4.10, 4.11, and 4.13 for help with Thevenin and Norton equivalent circuits and the technique of superposition. Refer Chapter 7 for help with first-order circuits.

Part I: Superposition Score: /30

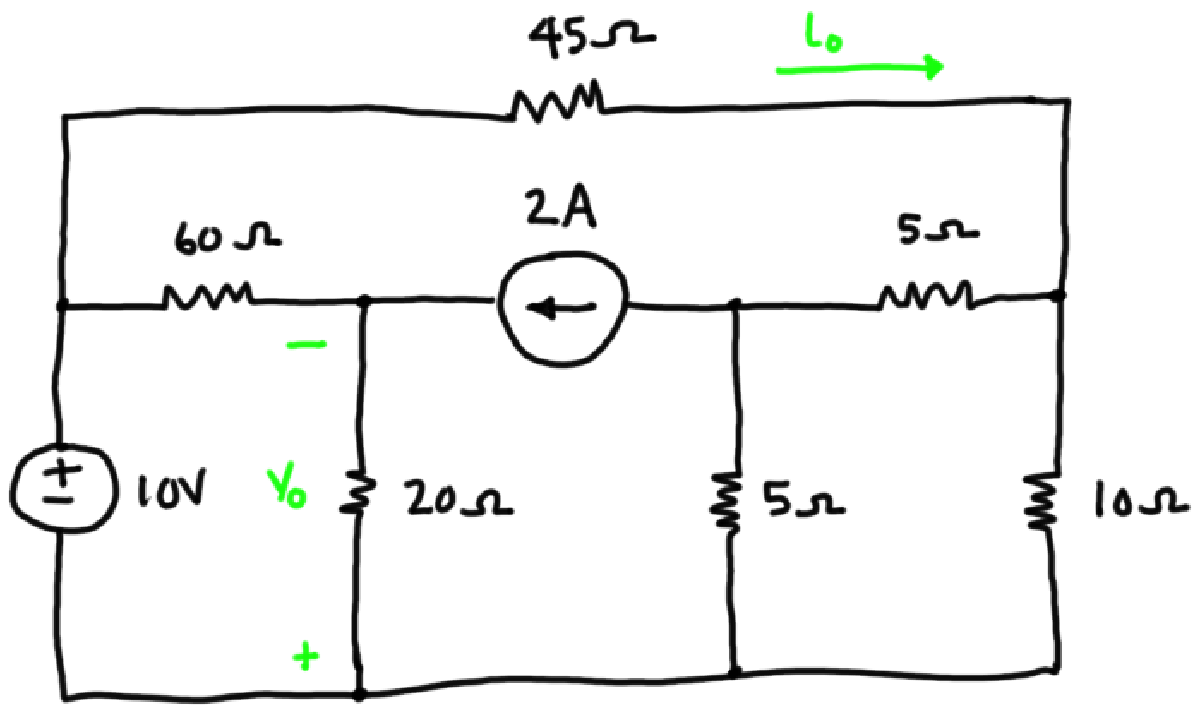


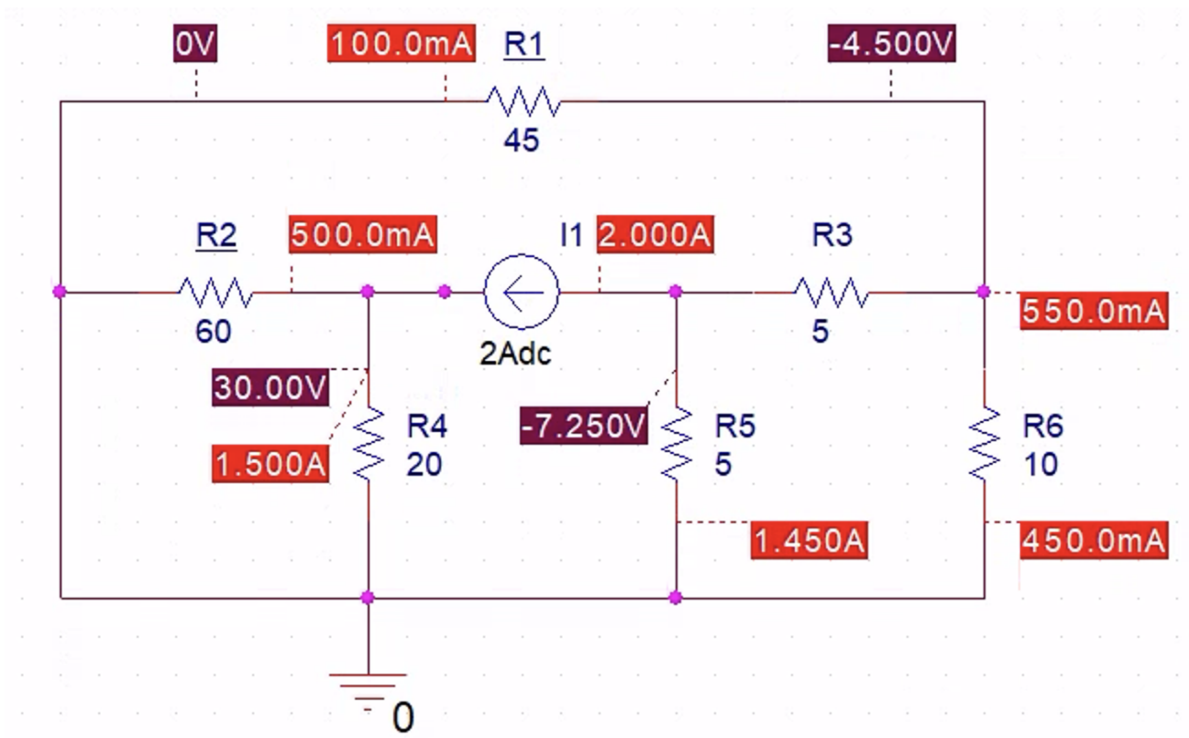
Figure 1. Superposition Circuit Diagram

In this section you will be using a "Bias Point" simulation in PSpice in order to determine the voltage v_0 and current i_0 by employing the principle of superposition. You will then verify your results by simulating the original circuit. Hint: Note the polarity of the voltage of interest v_0 and the direction of the current i_0

A. Turning off the voltage source Score: /10

Turn off the voltage source (replace it with a short circuit) while keeping the current source active. Simulate the resulting circuit and determine the voltage v_{0A} and current i_{0A}.

Simulation Result:



Answers:

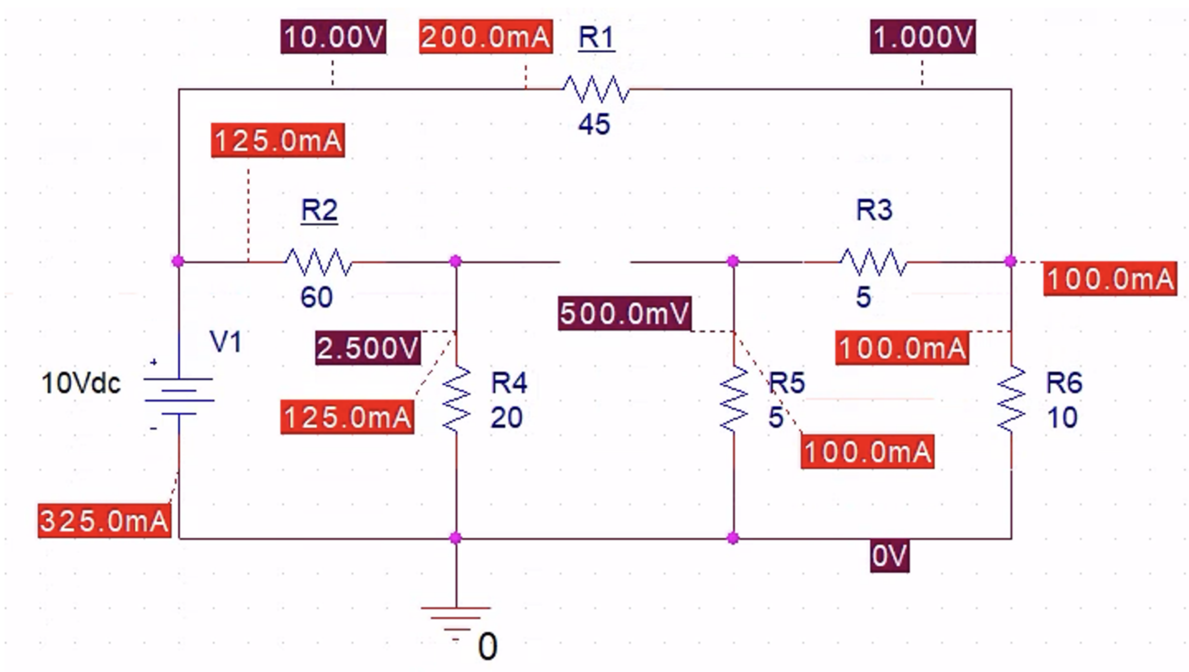
$v_{0A} =$ 30.0 V

$i_{0A} =$ 100.0 mA

B. Turning off the current source Score: /10

Turn off the current source (replace it with an open circuit) while keeping the voltage source active. Simulate the resulting circuit and determine the voltage v_{0B} and current i_{0B}.

Simulation Result:



Answers:

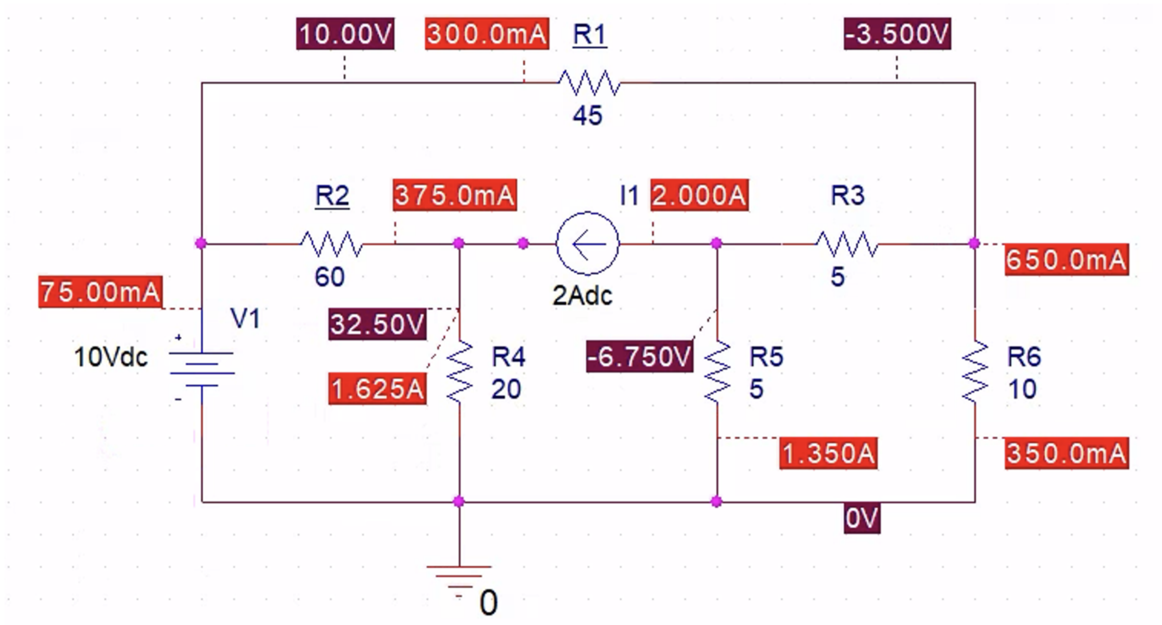
$v_{0B} = 2.50$ V

$i_{0B} = 200.0$ mA

C. Verifying superposition Score: /10

Simulate the circuit shown in Figure 1 and determine the voltage v_0 and current i_0.

Simulation Result:



Answers:

$v_0 = 32.50$ V

$i_0 = 300.0$ mA

Verify that superposition works. In other words, use the results from the simulation in parts A & B, along with the equations $v_0 = v_{0A} + v_{0B}$ and $i_0 = i_{0A} + i_{0B}$, and compare to the simulated results from part C. If they are not close to equal, then something has been done incorrectly and needs to be redone. Show your work below.

Calculations

$v_0 = v_{0A} + v_{0B}$

32.50 V = 30.0 V + 2.50 V

$i_0 = i_{0A} + i_{0B}$

300.0 mA = 100.0 mA + 200.0 mA

32.50 V = 32.50 V 300.0 mA = 300.0 mA

Part II: Thevenin Equivalent Circuit Score: /30

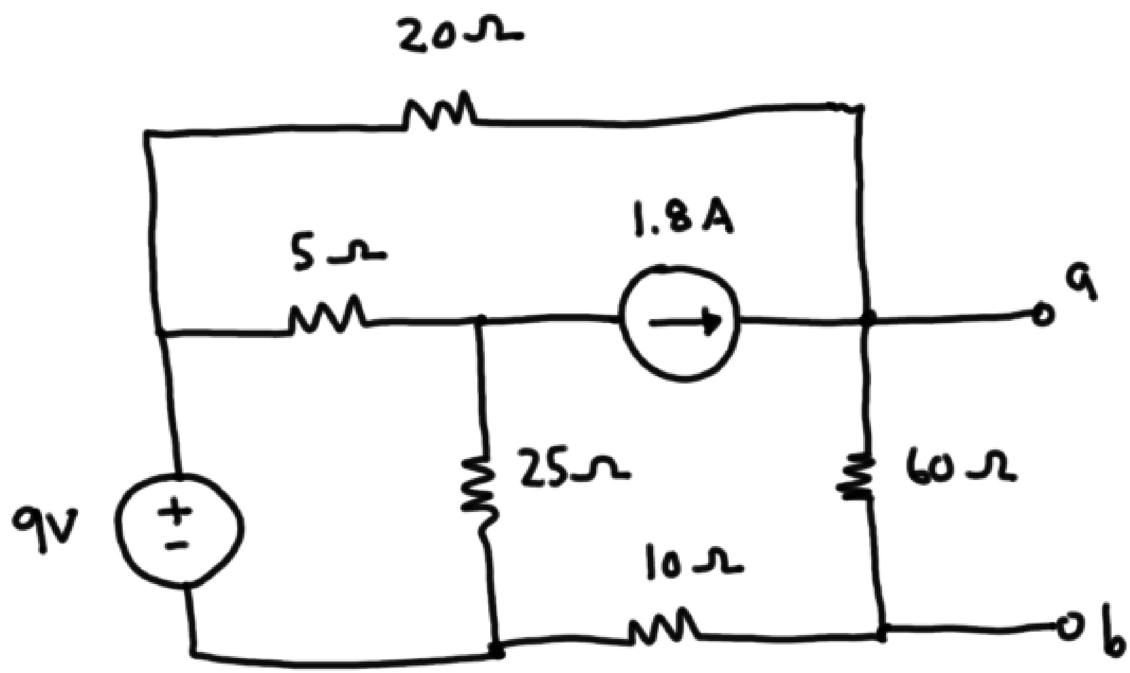


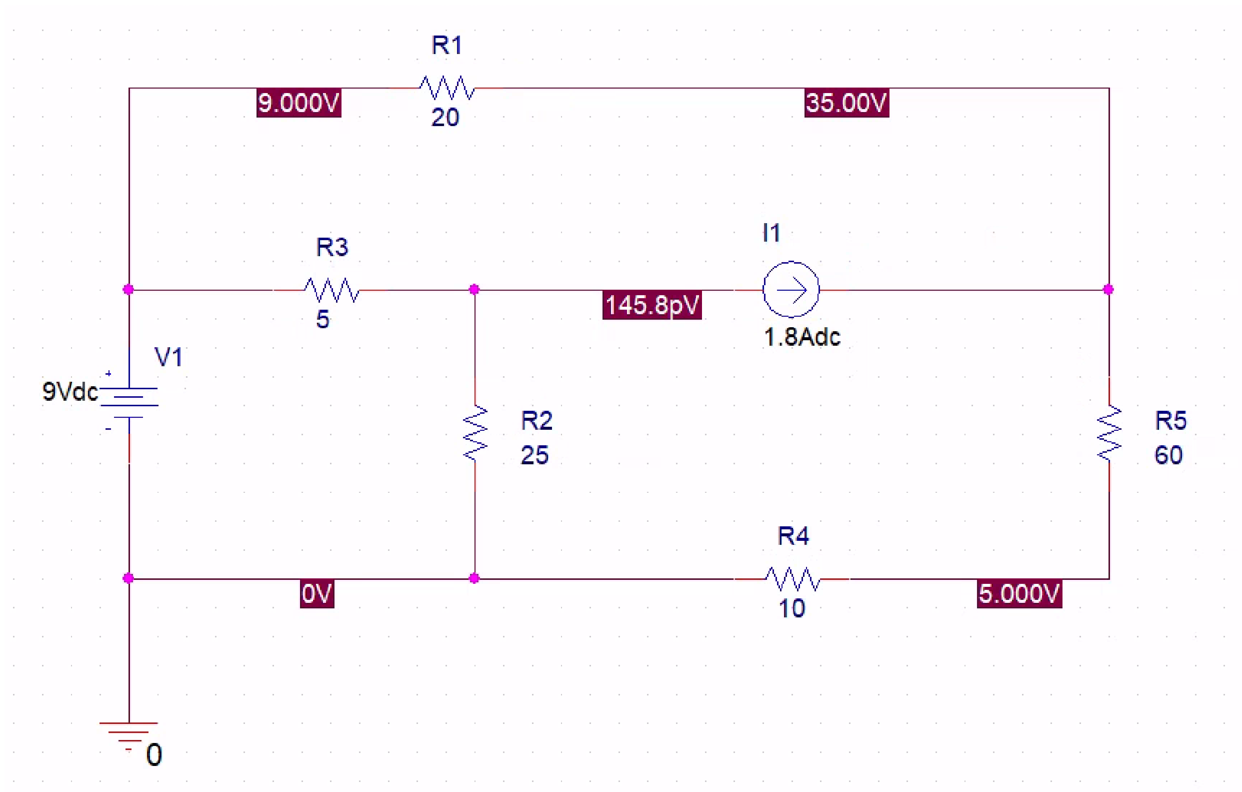
Figure 2. Thevenin Equivalent Circuit Diagram

In this section you will be using "Bias Point" simulations to find the Thevenin equivalent of the circuit shown in Figure 2 with respect to the terminals "a" and "b".

A. Finding the open circuit voltage Score: /10

Simulate the circuit shown in Figure 2 and determine the open circuit voltage v_{oc}. Hint: You do not need to make any changes to the circuit shown. You should be able to simulate the circuit directly and obtain the open circuit voltage.

Simulation Result:



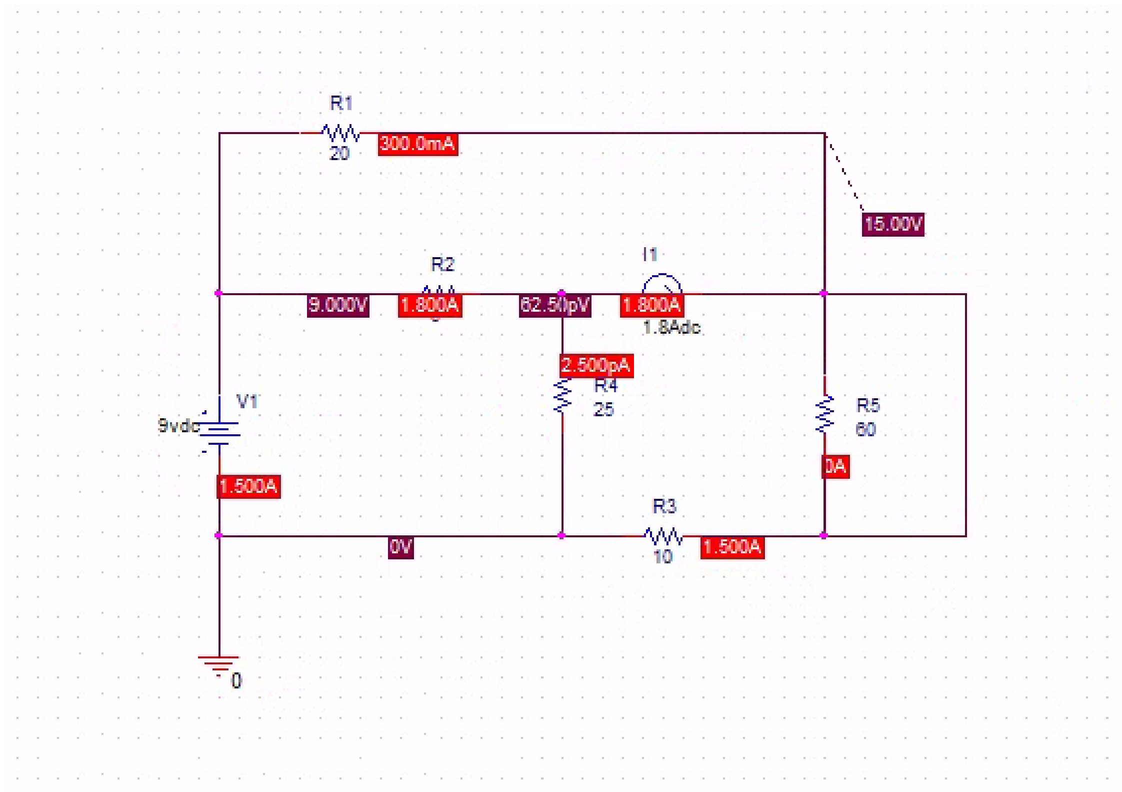
Answers:

$v_{oc} = 30V$

B. Finding the short circuit current Score: /10

Place a short circuit along the terminals “a” and “b”. Simulate the resulting circuit and determine the short circuit current i_{sc}. Hint: You cannot use the circuit shown in Figure 2 directly. You must modify it in order to obtain the correct short circuit current.

Simulation Result:



Answers:

i_{sc} =1.500A

C. Sketching the Thevenin Equivalent Circuit Score: /10

Using the results from parts A and B, along with the theory from section 4.10 in the textbook, sketch the Thevenin equivalent of the circuit shown in Figure 2 with respect to terminals "a" and "b" below. Make sure to show all calculations in full detail and references for any equations used.

Calculations:

$\frac{9-V_a}{5} + 1.8A = \frac{V_a}{60+10}$

$R_{th} = (20 + 10) \,||\, 60 = \frac{30 * 60}{30 + 60} = 20\Omega$

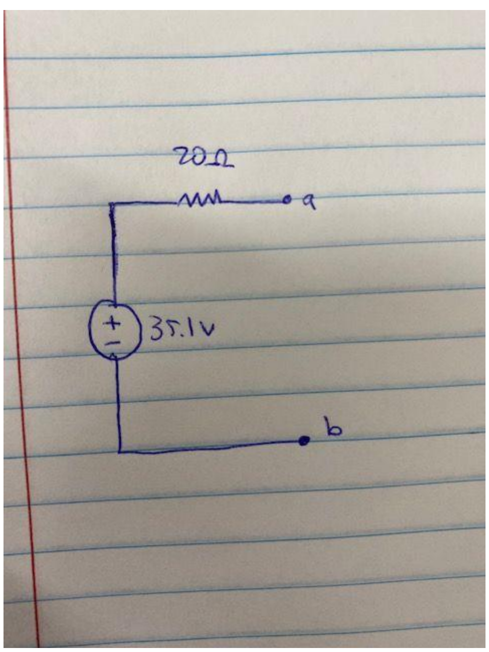
$70(9\text{-}V_a)\text{+}2520 = 20V_a$

$640\text{-}70V_a + 2520 = 20V_a$

$3160 - 70V_a = 20V_a$

$V_a = 35.1V$

Sketch of the Thevenin equivalent: Include the values of the parameters obtained in calculations portion above in your sketch.



Part III: Charging and Discharging Capacitor Score: /30

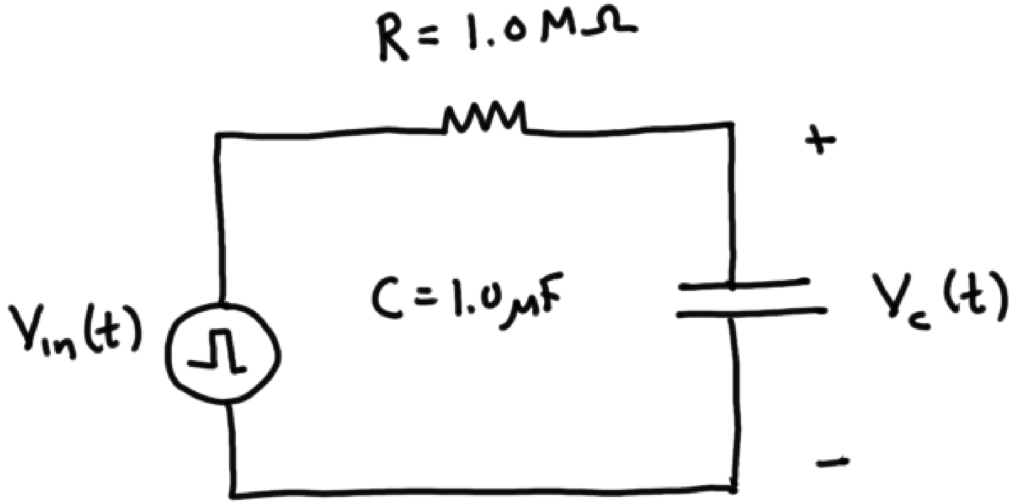


Figure 3. First-order Circuit Diagram

In this exercise you will be exploring a "Time Domain (Transient)" simulation in PSpice. You will be using the pulse voltage source "VPULSE" in order to approximate the desired square wave voltage input $V_{in}(t)$ shown in Figure 4 below. The voltage output will be the voltage across the capacitor, $V_C(t)$.

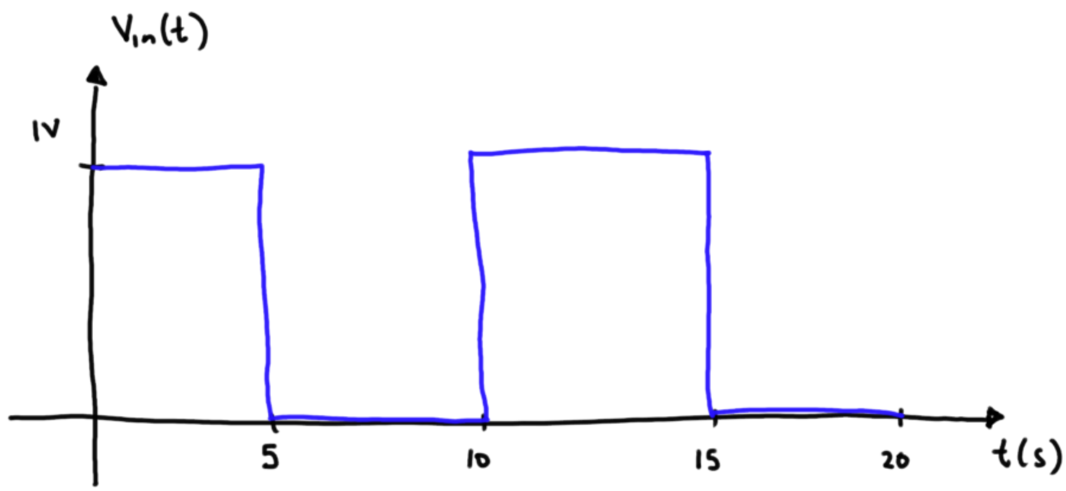


Figure 4. Square Wave Input Voltage

VPULSE has seven parameters that describe its shape and are shown below in Figure 5.

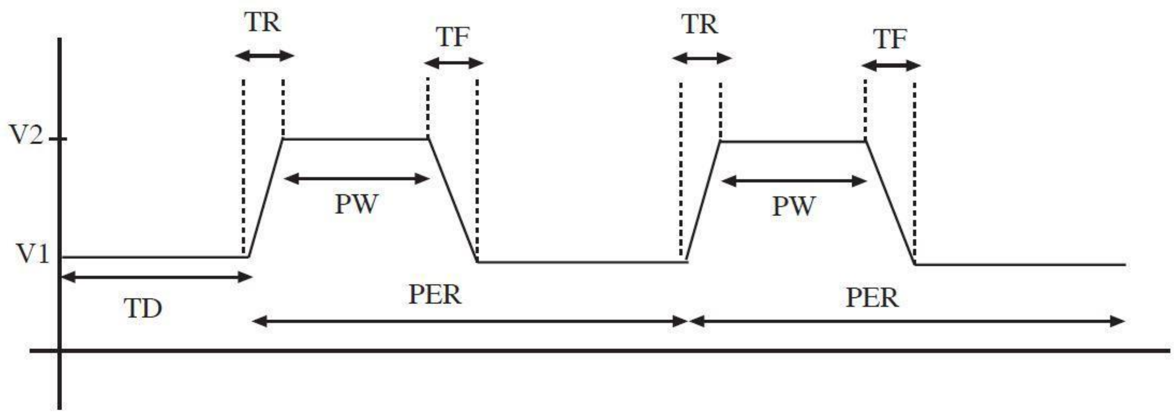
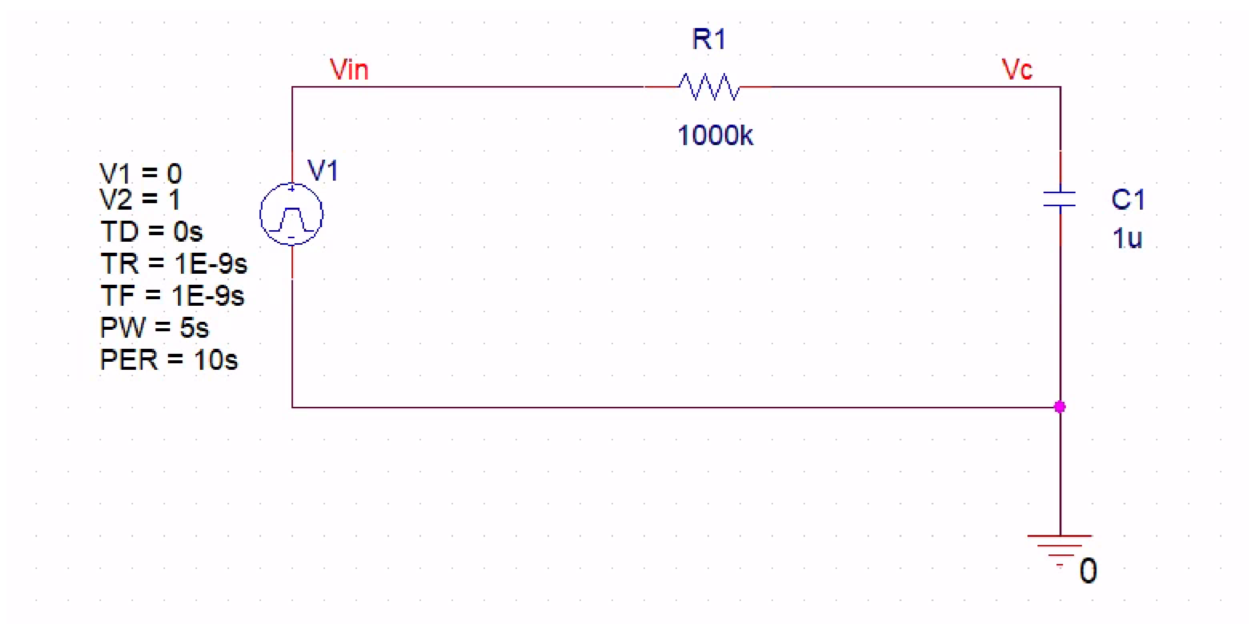


Figure 5. VPULSE Parameters in PSpice

In order to simulate the voltage waveform shown in Figure 4, the appropriate parameter values must be chose. If you have any questions about how to do this, ask your instructor for guidance.

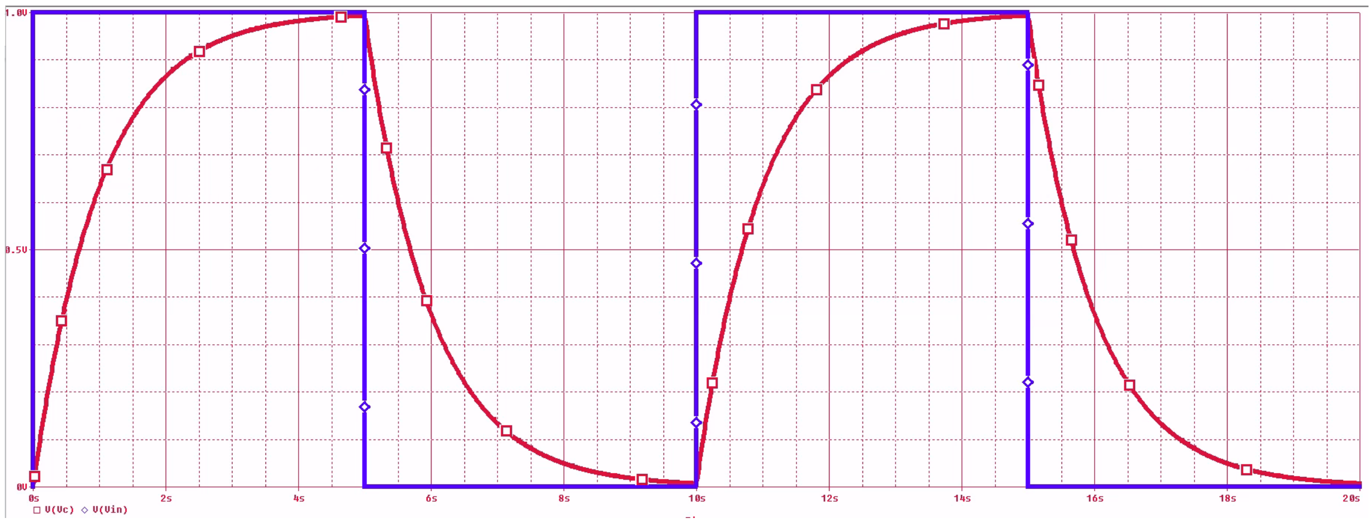
In the "Time Domain (Transient)" simulation we wish to choose the run time so that two full periods of the voltage input are displayed in the output results. Also choose the time step to be 0.01 s. Also, to simulate properly the initial voltage across the capacitor must be manually set to 0 V. In other words, choose IC = 0. Include the circuit schematic after the simulation has run successfully below. Also include your output results on the next page.

<u>Circuit Schematic:</u> (this is after the simulation)



Include a graph that shows both $V_{in}(t)$ and $V_C(t)$ as functions of time for the entire run time of 20 s. The background must be in white. Choose the appropriate colors and thicknesses of the traces so that they can be seen clearly after they have been printed out and turned in. Include a legend below the graph that clearly states which trace belongs to which voltage waveform.

Simulation Results:



Part IV. Conclusion Total Score: /10

Explain in a few paragraphs the purpose of the lab, the experimental set up and methodology, and central results of the lab and these experiments. **You should be quantitative** in this summary. Include any important equations used and explain their significance. Write the conclusion as if you were writing an English essay. This is an important portion of the lab, so make sure to do a good and thorough job.

This laboratory shows different types of simulations to students using the PSpice simulator. The first part of the laboratory shows how to use “Bias Point” simulation on PSpice to work on the principles of superposition by deactivating the voltage source and replacing it with a short circuit, and deactivating the current source and replacing it with an open circuit. In the second part of the lab we were given a circuit in which we had to find the Thevenin equivalent circuit of said circuit. After following the provided steps we found out the open circuit voltage of the figure 2. Then after adjusting the circuit we found out the short circuit current of the given circuit. After deducing both the open circuit voltage and the short circuit current we then computed the Thevenin equivalent voltage and resistance using formulas from the book and finally we created the Thevenin equivalent circuit of figure 2. In the third part of the lab we were given a pulse voltage and told to create a graph encompassing the information provided through the graph in figure 4. After analyzing the graph in figure 4 we discovered the needed information in order to create the graph in a Pspice simulation. Following the instructions we were able to successfully create the circuit and graph with little to no errors. Overall the lab required us to use out critical thinking abilities and compute various voltages and currents. This lab helped us grasp the idea of how circuits will function and the duties that’ll be entrusted to us throughout this course.