University of engineering & technology Peshawar

Circuit & system-1

Lab report # 9

Fall 2020

Submitted by: Ashfaq Ahmad

Section: B

Reg No: 19PWCSE1795

Semester: 2nd

Submitted to:

Eng: faizullah

Department Of Computer System Engineering

ASSESSMENT RUBRICS LAB # 9

Node Voltage Analysis using PSPICE

LAB REPORT ASSESSMENT				
Criteria	**Excellent**	**Average**	**Nill**	**Marks Obtained**
1. Objectives of Lab	All objectives of lab are properly covered [Marks 0.5]	Objectives of lab are partially covered [Marks 0.25]	Objectives of lab are not shown [Marks 0]	
2. Node Voltage Analysis (Theory, Circuit Diagram)	Brief introduction about Node Voltage Analysis (what is Node voltage analysis, What are nodes, How to apply KCL equations at each node) is shown along with properly labeled circuit diagram [Marks 1]	Some of the points about Node Voltage Analysis are missing and circuit diagram is not properly labeled [Marks 0.5]	Introduction about Node Voltage Analysis and circuit diagram is not shown [Marks 0]	
3. PSPICE Simulator	Brief introduction of PSPICE simulator [Marks 1]	Brief introduction of PSPICE simulator Is not shown [Marks 0]		
4. Procedure	All experimental steps are shown in detail along with how to verify Node Voltage Analysis. [Marks 1.5]	Some of the experimental steps are missing [Marks 1]	Experimental steps are missing [Marks 0]	
5. Observations & Calculations	Mathematical calculations are shown and comparison with PSPICE results. [Marks 5]	Mathematical calculations are shown but no comparison with PSPICE results [Marks 2.5]	No mathematical calculations are shown [Marks 0]	
6. Conclusion	Conclusion about experiment is shown [Marks 1]	Conclusion about experiment is partially shown [Marks 0.5]	Conclusion about experiment is not shown [Marks 0]	
Total Marks Obtained:_____ Instructor Signature: _____				

NODE VOLTAGE ANALYSIS USING PSPICE

1. OBJECTIVES OF LAB:

To analysis node voltage, we use KVL law. There are several benefits of Kirchhoff's voltage law are given below:

- To Know about the Kirchhoff Voltage Law.
- Application of Kirchhoff voltage law in circuits.
- To analyze the circuit and circuit elements.
- To know about nodal voltages.
- To know about PSPICE & its uses.

2. NODE VOLTAGE ANALYSIS

DEFINITION:

In electric circuits analysis, **nodal analysis**, **node-voltage analysis**, or the **branch current method** is a method of determining the voltage between "nodes" (points where elements or branches connect) in an electrical circuit in term of branch currents.

There are two basic methods that are used for solving any electrical network: **Nodal analysis** and **Mesh analysis**. In this lab, let us discuss about the **Nodal analysis** method.

In Nodal analysis, we will consider the node voltages with respect to Ground. Hence, Nodal analysis is also called as **Node-voltage method**.

Procedure of Nodal Analysis

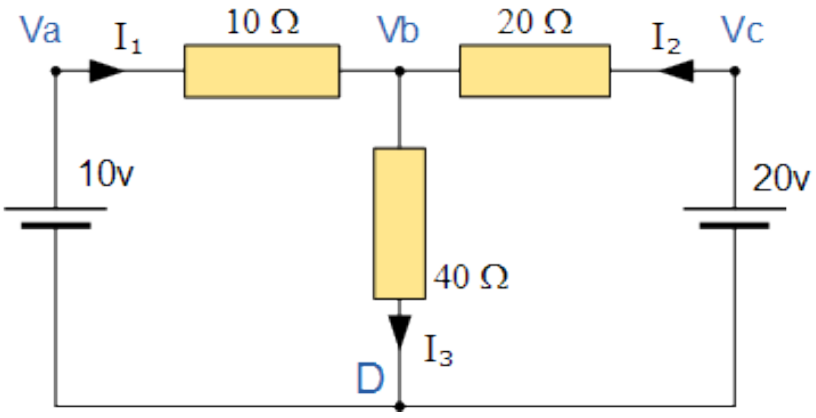
Follow these steps while solving any electrical network or circuit using Nodal analysis.

- **Step 1** − Identify the **principal nodes** and choose one of them as **reference node**. We will treat that reference node as the Ground.

  (A principal node or junction is a point where 3 or more branches join.)

- **Step 2** − Label the **node voltages** with respect to Ground from all the principal nodes except the reference node.
- **Step 3** − Write **nodal equations** at all the principal nodes except the reference node. Nodal equation is obtained by applying KCL first and then Ohm's law.
- **Step 4** − Solve the nodal equations obtained in Step 3 in order to get the node voltages.

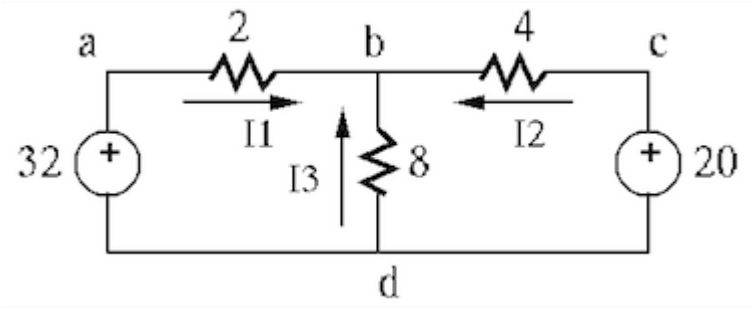
EXAMPLE:



HOW TO APPLY KCL ON EACH NODE:

- Assume there are "n" nodes in the circuit. Select one of them as the ground, the reference point for all voltages of the circuit. The node voltage at each of the remaining "n-1" nodes is an unknown to be obtained.
- Express each current into a node in terms of the two associated node voltages.
- Apply KCL to each of the "n-1" nodes to set the sum of all currents into the node to zero, and get "n-1" equations.

1. Solve the equation system with "n-1" equations for the "n-1" unknown node voltages.



- In the same circuit considered previously, there are only 2 nodes b and d (a & c are not nodes) . We assume node d is the ground, and consider just voltage $V_b=V_{bd}$ _ at node v is the ground, and consider just voltage b, we have

PSPICE SIMULATOR:

SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose, open source analog electronic circuit simulator. It is a program used in integrated circuit and board-level design to check the integrity of circuit designs and to predict circuit behavior.

PSPICE is a computer-aided simulation program that enables you to design a circuit and then simulate the design on a computer.As this is one of its main purposes, it is used extensively by electronic design engineers for building a circuit and then testing out how that circuit will simulate.

PSPICE stands for Program Simulation with Integrated Circuit Emphasis.The Electronics Research Laboratory of the University of California developed it and made it available to the public in 1975.PSPICE is largely popular because of its user-friendly interface, extensions that support modeling of digital circuits, and its no-cost basic version.

PSPICE is a general purpose program designed for a wide range of circuit simulation including the simulation of nonlinear circuits, transmission lines, noise and distortion, digital circuits, mixed digital and analog circuits. It can perform dc analysis, steady-state sinusoidal (AC) analysis, transient analysis, and Fourier seriesanalysis

PROCEDURE:

Procedure to analyze complex circuitusing PSPICE as follows.

- First of all open the simulation software (PSPICE).
- Open the workspace.
- Then open the schematic diagram.
- In the above figures we can see that we have found the node voltages and mesh current, so put the different components in the schematic.
- Then place DC voltage source and resisters of specific value with desired design making loops and nodes.
- Ground earth is used to complete the circuit and provide a ground to the circuit.
- Then apply mesh current law and node voltage law we will find the value of each and every point.
- Connect all these components by means of wire and check marks the voltage and current show button.
- For confirmation new window will open which will show the message with zero error.
- After successful compilation of program it shows the desired currents passing through different resistors.

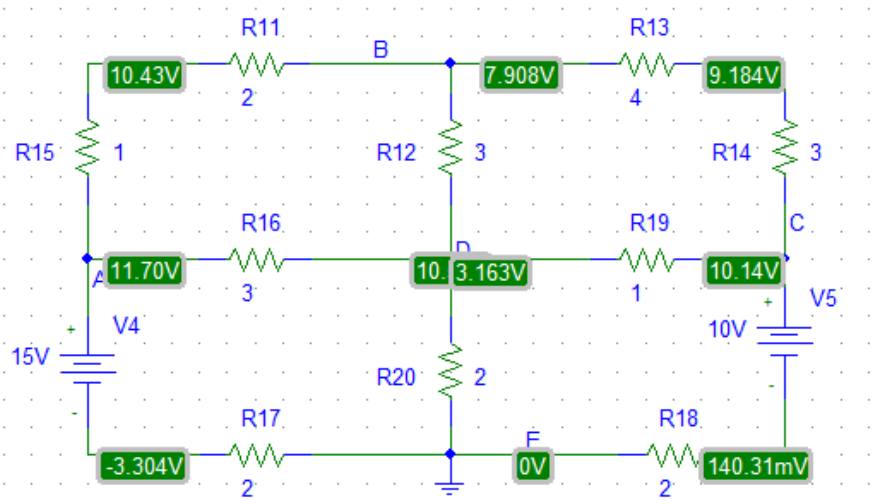
EXPERIMENTAL STEPS:

1. Draw your circuit in PSPICE.
2. Simulate the circuit.
3. Note down on the voltages between nodes.

OBSERVATIONS AND CALCULATIONS:

V_A	V_B	V_C	V_D
11.70V	7.908V	10.14V	10.5V

CIRCUIT DIAGRAM;



CONCLUSION:

The Node Voltage Method is one of two well-ordered methods of solving a circuit. ... Assign node voltage names to the remaining nodes. Solve the easy nodes first, the ones with a voltage source connected to the reference node. Write Kirchhoff's Current Law for each node.

After attending this lab, I am able to solve the circuits using node voltage analysis and clearly understood the concept of node, reference node and node voltage.

……T H E EN D……