Assignment 2 - SPICE Simulation

M.Saketh Sai Ram - EE22B022

Introduction

The Python script `evalSpice.py` presented here is designed to perform circuit analysis on linear resistive circuits using a SPICE file parser. SPICE is a widely-used tool for simulating and analyzing electronic circuits. This script allows you to read a *.ckt* file, extract information about circuit components, and compute essential parameters such as node voltages and currents through the voltages.

Note : Only Resistors with non-zero resistance, Independent DC Volatge and DC Current Sources are used!

It is completely based on Kirchoff's Voltage Law and Current Law.

KVL : Algebraic Sum of all voltage differences of different circuit elements around any closed loop is zero.

KCL : Algebraic Sum of all currents entering and exiting a node must be equal to zero.

Code Description

The python script reads each line of the input file and parses the circuit description. It identifies circuit components such as resistors, voltage sources, and current sources and stores the data related to the components in dictionaries. The lines which are in between the terminators *.circuit* and *.end* will be considered. The text outside the terminators will not be considered. There must be reference node (GND) in the circuit. To make the construction of matrices easier, all the nodes were labelled with the integers `0,1,...`.

```
>>> V1 n1 GND dc 2e1
```

The dictionary of the above component will be stored in the form of `{'C':"V1",'N1':"n1",'N2':"n2",'M':20.0}`

Matrix Setup and Equation Formulation

To solve the circuit, the script sets up matrices ($A^{n\times n}$ and $B^{n\times 1}$) with all entries as zeros with n is equal to sum of number of nodes and number of voltage sources. Those matrices represent the system of linear equations based on Kirchhoff's laws. It accounts for resistor equations, voltage source equations, and current source equations.

Assume there are n nodes(indices ranges from 0 to $n-1$) and currents through k voltage sources(indices ranges from n to $n+k-1$).

- Assume GND has index g, while solving for the node voltages, the script excludes the g^{th}row and g^{th} column since the ground node has zero voltage.

System of Linear equations:

- n KCL equations at n nodes.
- k equations of the type $V_{N1} - V_{N2} = V_M$ for the nodes connected by voltage sources.

The node quations take the form

$$\sum \frac{V_{N1} - V_{N2}}{R_{N1,N2}} + \sum I_{N1,N2} = -\sum \xi_{N1,N2} \text{ —— } (\mathbf{1})$$

where the first sum is over the passive branches, the second sum over voltage sources and the third sum is over current sources.

The auxilary equations take the form

$$V_{N1} - V_{N2} = \delta_M \text{ —— } (\mathbf{2})$$

Using the equations (1) and (2), construct an equation of the following form:

$Ax = B$; where x contains the node voltages and the voltage currents.

Matrix A: It represents the co-efficients of the variables in the system of linear equations. It has two parts, one for the node voltage equations and one for the voltage source equations.

- For the node voltage equations(KCL equations), the elements of the diagonal a_{ii} for $i = 0$ to $n-1$ are positive,and the off-diagonal elements corresponding to the passive branches are negative.

The value of the element a_{ii} becomes $\sum \frac{1}{R_k}$, R_k is the value of the resistance which is connected from the node which is labelled as i to the other nodes.

Similarly, a_{ij} is equal to $-\sum \frac{1}{R_{ij}}$, where R_{ij} is the value of resistance connected between the nodes with labels i and j.

For the voltage source equations, there will be a row for each voltage source connection. The entries corresponding to the positive node will have the entry 1.0 in A, and the entries corresponding to the negative node will have entry -1.0 in A.

Matrix B: It represents the RHS of the system of linear equations. It includes two parts:

For the node voltage equations, the values on the RHS side depend on the sum of external currents connected to that node.

For the voltage source equations, the values on the RHS side represents the voltage connected between the nodes N1 and N2.

- List `x` contains the node voltages and the current through voltage sources. The first 'n' values in `x` represent the node voltages which are added to `node_voltage`, and the remaining `k` values represent the current through voltage sources which are added to `nodes_current`.

Two dictionaries returned by `evalSpice.py` for a general circuit are as follows:

```
({'1': -10.0, '2': -5.029239766081872, 'GND': 0.0}, {'V1': -0.004970760233918128})
```

Error Handling

1. **FileNotFoundError** : If the input file doesn't exists
2. **ValueError**: It occurs if any component other than V,R,I are present, Malformed circuits, Parallel Voltage Sources, Series Current Sources, circuit with no GND node, for repeated component names and if matrix A is singular i.e., when it is an invalid circuit.

Note: Two more test cases have been included in this python script.

File Names : `test_malformed_circuit_same_name_comp.ckt, test_no_GND.ckt`