SPICE simulation in Python

Introduction

This program implements a basic SPICE simulation. The inputs are in the form of a .ckt fle containing
a netlist with resistances, current sources, and voltage sources as circuit elements. The current
is assumed to flow from the first specified node to the second. The code outputs the voltages at
each node as well as the current through each voltage source in 2 dictionaries.

Implementation

1. Parsing:- The .ckt file is read line by line and a dictionary is made to store values and location
    of each voltage source. Another default dictionary stores the equivalent current source and
    equivalent admittance between 2 nodes. If a short wire is connected, it is treated as an admittance of
    value 2^{50} Siemens. All the nodes are mapped to consecutive integers.
2. Nodal Analysis- Is used to solve the circuits. Some modifications done are; GND is treated as another
    node while creating the matrix with the equation 1*VGND=0 added. Also, Current through voltage
    sources
    are considered as variables when writing KCL, and constraint equations of the form Vi-Vj=Voltage
    source
    connecting i and j are added.
3. Solving the linear system- Is done with np.linalg.solve()

Complexity

If m is the number of voltage sources and n is the number of nodes,
Time- np.linalg.solve() is $O(x^3)$ and all other operations are $<=O(x^2)$, hence $T=O((m+n)^3)$
Space- $O((m+n)^2)$