

Project Documentation: Simple Circuit Simulator

Student : Trinh Truong
Student ID : 4888769
 Dr. Robert Kerestes
Instructor : Mr. Collin Griffin
 Mr. Paulo Radatz De Freitas

I. Project Overview

The **Simple Circuit Simulator** is a Python-based application designed to model and solve fundamental Direct Current (DC) electrical circuits. The simulator is designed to analyze a specific DC network topology consisting of a single-loop system. The configuration includes:

- **Voltage Source:** A DC power supply connected at the reference bus (Bus A) that provides the primary electromotive force for the circuit.
- **Series Resistor:** An element representing the line impedance or a discrete resistor connected between the source bus (Bus A) and the load bus (Bus B).
- **Constant Impedance Load:** A load component connected at Bus B, modeled as a fixed resistance derived from its nominal power and voltage ratings.

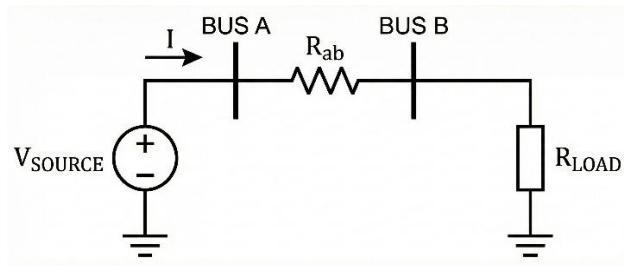


Fig.1. Circuit Configuration

Purpose: The primary goal of this simulator is to provide a computational framework for analyzing basic DC networks. It serves as a bridge between theoretical electrical engineering concepts and object-oriented programming (OOP) implementation, allowing users to define circuit topologies and calculate steady-state parameters.

Key Features and Functionality:

- **Component Modeling:** Object-oriented representation of Busses, Resistors, Voltage Sources, and Loads.
- **Constant Impedance Loads:** Implementation of load models that derive resistance from nominal power and voltage ratings.
- **Automated Power Flow:** A dedicated solver that calculates nodal voltages and branch currents.

- **Result Reporting:** Built-in methods to display the state of the circuit clearly.

Real-World Applications: While simplified, this simulator mimics the core logic used in industrial power system analysis software. It solves the problem of manual calculation errors in series-parallel DC networks and provides a scalable foundation for more complex grid simulations.

II. Class Diagram

The diagrams for each class can be represented as below:

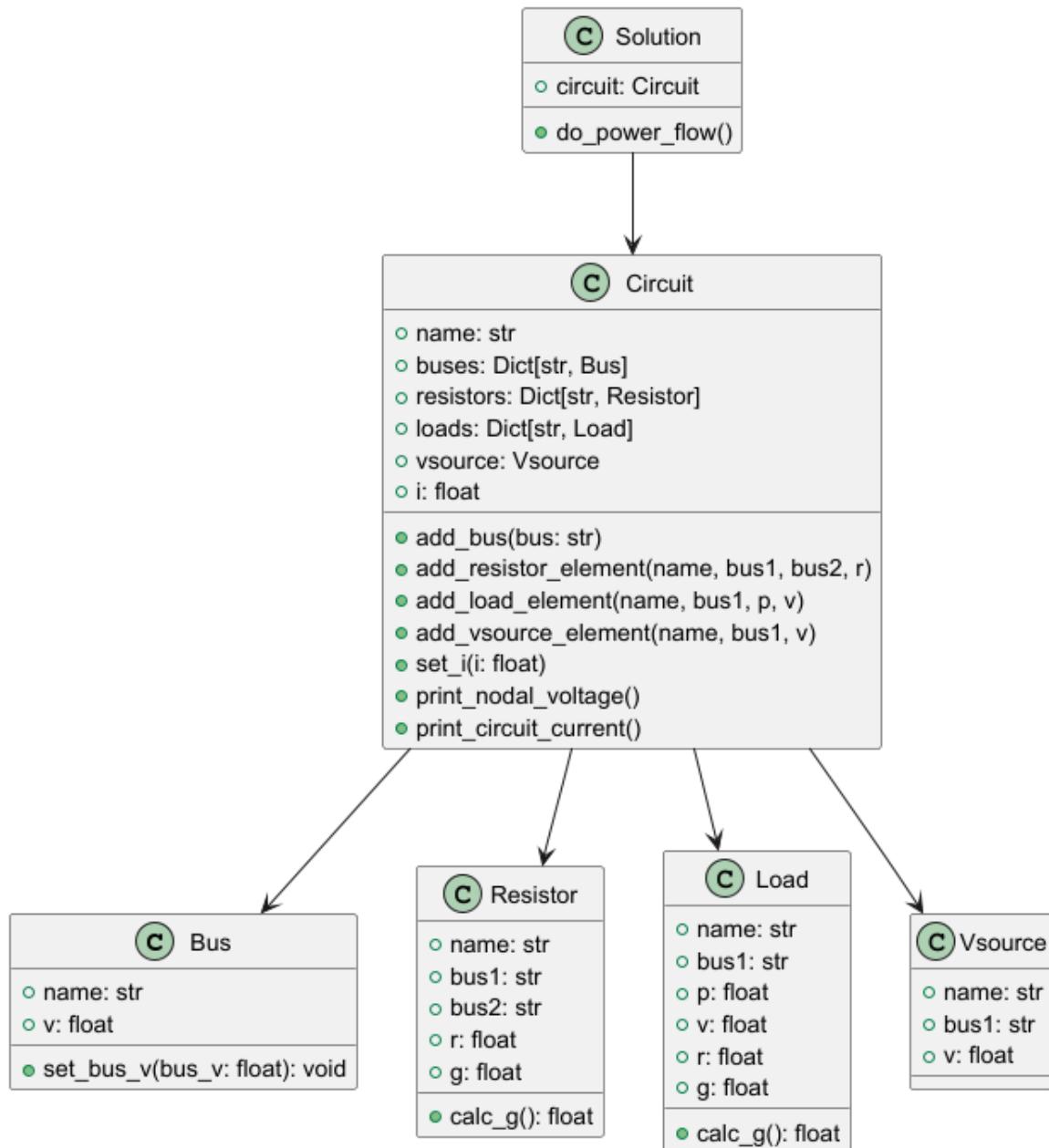


Fig.2. Classes diagram

III. Relevant Equations

The simulator performs calculations based on the following fundamental electrical laws:

- Ohm's Law:

$$V = I \times R$$

- Power-Voltage Relationship:

$$R_{load} = \frac{V_{load}^2}{P_{load}}$$

- Conductance Calculation:

$$G = \frac{1}{R}$$

- Kirchhoff's Voltage Law (KVL) & Series Solver:

$$I = \frac{V_{source}}{\sum R_{resistors} + R_{load}}$$

IV. Example Case

This section demonstrates the simulator's capability using a specific test scenario.

1. Problem Definition

A DC circuit is defined based on Fig.1 with the following parameters:

- Voltage Source (V_{source}): 240V;
- Line Resistor (R_{ab}): 10Ω;
- Load (R_{load}): Rated at 2400W and 240V.

2. Solution Process

Execution steps:

- Calculate load resistance:

$$R_{load} = \frac{240^2}{2400} = 24\Omega$$

- Total resistance:

$$R_{total} = R_{ab} + R_{load} = 10 + 24 = 34\Omega$$

- Circuit current:

$$I = \frac{V_a}{R_{total}} = \frac{240}{34} = 7.0588A$$

- Nodal voltages:

$$V_a = V_{source} = 240V$$

$$V_b = I \times R_{load} = 7.0588 \times 24 = 169.411V$$

3. Expected Output

When executed, the program generates the following results:

```
Bus a: V = 240.0 V
Bus b: V = 169.41176470588235 V
Circuit current I = 7.0588235294117645 A

Process finished with exit code 0
```

Fig.3. Output from the simulator