# Simple Circuit Simulator
# Project 1 – Milestone 5 Documentation

## 1. Project Overview

The Simple Circuit Simulator is a Python-based educational software tool developed to model and solve a simple DC electrical circuit. The simulator focuses on a circuit composed of a voltage source, a series resistor, and a load resistor connected between two buses. The purpose of this project is to demonstrate how fundamental circuit theory can be implemented using object-oriented programming techniques.

The simulator solves the problem of determining nodal voltages and circuit current when circuit parameters such as resistance, power consumption, and source voltage are known. By organizing the simulator into separate classes for circuit elements, circuit management, and solution logic, the program provides a clear and modular structure consistent with standard engineering practice.

Although the simulator is limited to a simple circuit topology, it reflects concepts used in real-world electrical engineering applications such as steady-state circuit analysis, power system modeling, and engineering education.

## 2. Class Diagrams

The Simple Circuit Simulator is structured using object-oriented design principles. Class diagrams are used to illustrate the relationships between classes, their attributes, and their methods.

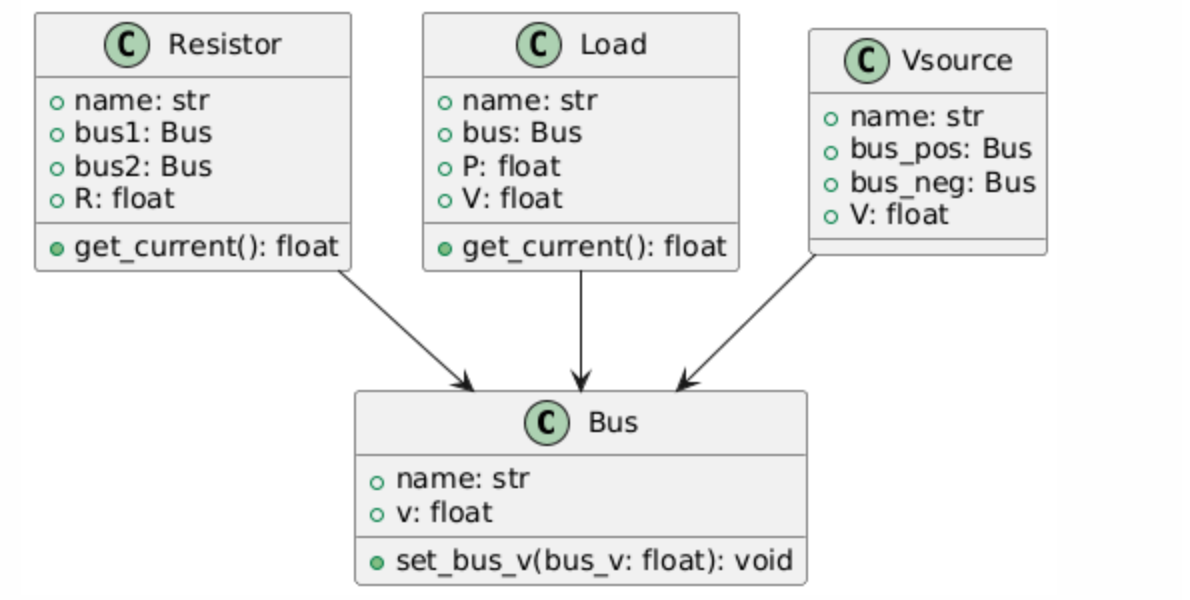


Figure 1. Element class diagram showing Bus, Resistor, Load, and Vsource.

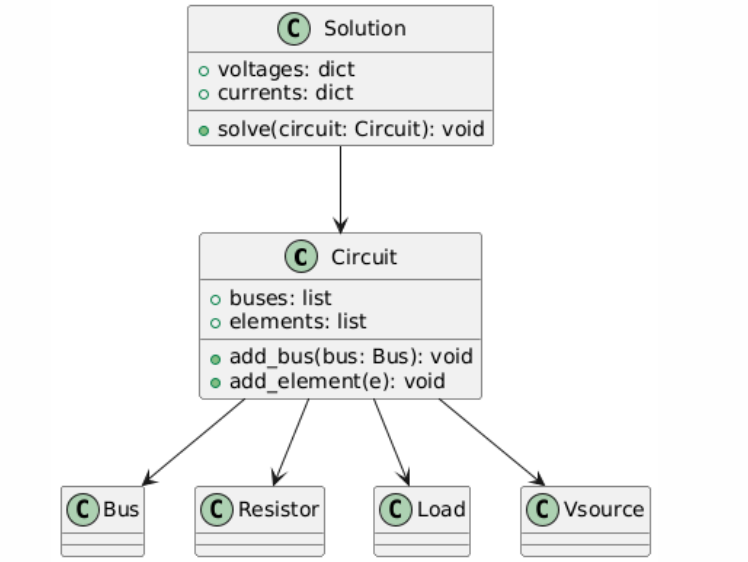


Figure 2. The class diagrams illustrate the object-oriented structure of the Simple Circuit Simulator. They show how circuit elements, buses, the circuit container, and the solution class interact, as well as the key attributes and methods used to model and solve a DC circuit.

### 3. Relevant Equations

The simulator is based on standard DC circuit equations, which are implemented directly in the solution algorithm.

Ohm's Law describes the relationship between voltage, current, and resistance:

$$V = IR$$

The relationship between power, voltage, and resistance for a resistive load is given by:

$$P = \frac{V^2}{R}$$

This expression is rearranged to represent the load as a constant impedance using conductance:

$$G = \frac{P}{V_{nom}^2}$$

Conductance is used throughout the simulator to simplify circuit calculations. The conductance of a resistor is defined as:

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

The conductance of the load is defined as:

$$G_{load} = \frac{P}{V_{nom}^2}$$

Kirchhoff's Voltage Law is applied around the circuit loop to determine the system current and nodal voltages:

$$V_A - IR - V_B = 0$$

Using conductance values, the circuit current is calculated as:

$$I = V_A \cdot \frac{G_r G_{load}}{G_r + G_{load}}$$

Once the circuit current is known, the voltage at bus B is calculated using:

$$V_B = V_A - IR$$

## 4. Example Case

This example demonstrates the operation of the Simple Circuit Simulator using a test case different from the validation case.

The circuit consists of a voltage source connected to bus A with a voltage of 80 V. A series resistor of 8 Ω connects bus A to bus B. A load is connected at bus B with a rated power of 800 W and a nominal voltage of 80 V. The load is modeled as a constant impedance.

The resistor conductance is calculated as $G_r = 1/8 = 0.125$ S. The load conductance is calculated as $G_{load} = 800/80^2 = 0.125$ S.

The circuit current is calculated using the conductance-based equation, resulting in a current of 5 A. The voltage at bus B is then calculated by subtracting the voltage drop across the resistor from the source voltage, resulting in a bus B voltage of 40 V.

The expected output for this example is a voltage of 80 V at bus A, a voltage of 40 V at bus B, and a circuit current of 5 A.