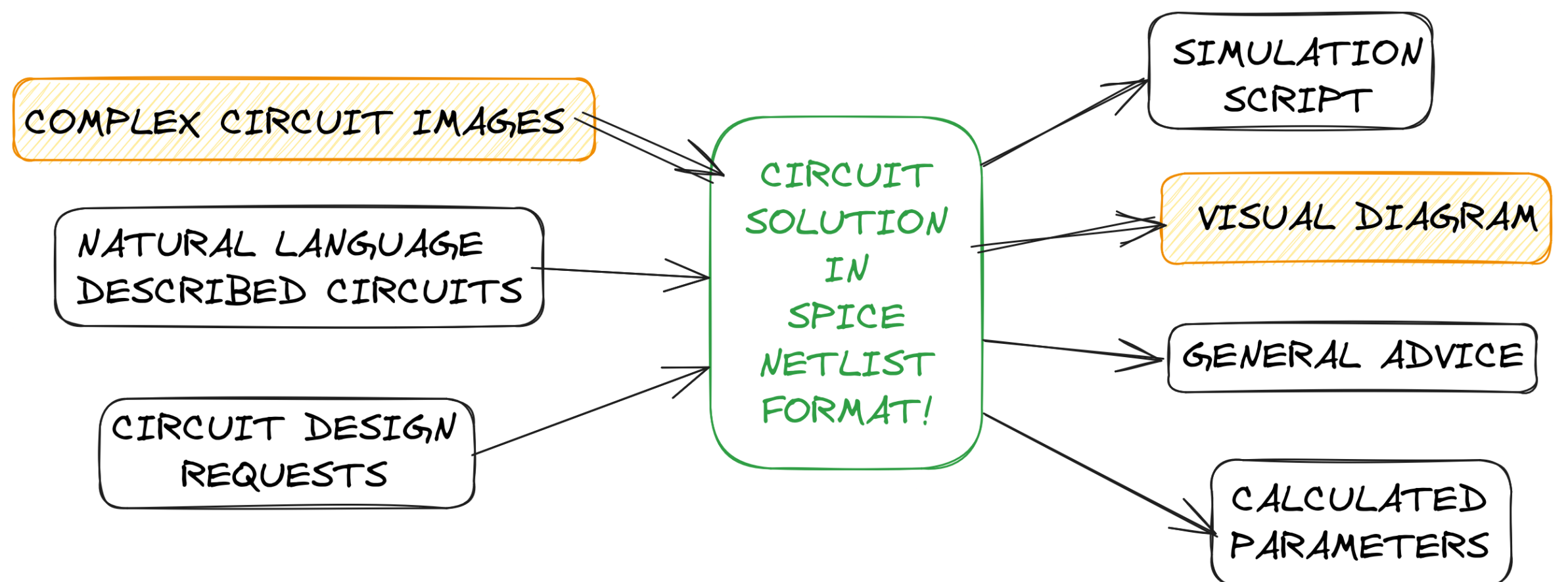


# Vision-Text Integrated Generative AI for Accurate Circuit Solutions

11360109 何承軒 111360128 林晏宇 11360137 許鎧晏 11360138 蔣安聖

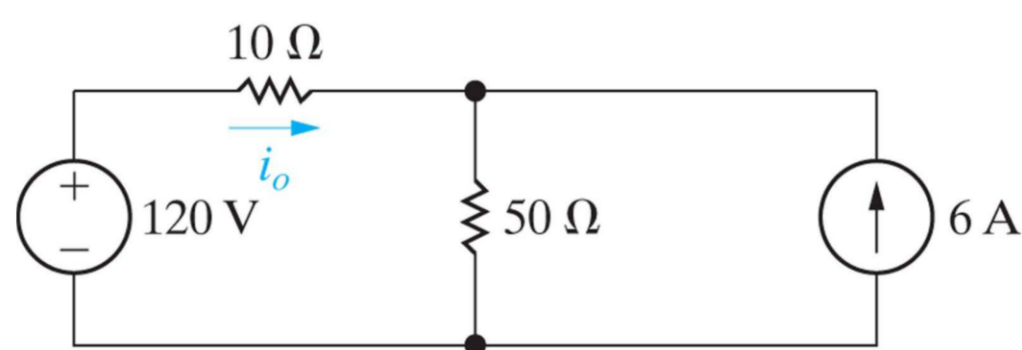
## Versatile Usage with Reliable Outputs

We specifically trained the model to interpret input using a unified format (SPICE) and to compute the answer, rather than merely performing calculations.



## Why SPICE?

You



Solve the circuit for  $i_o$ .

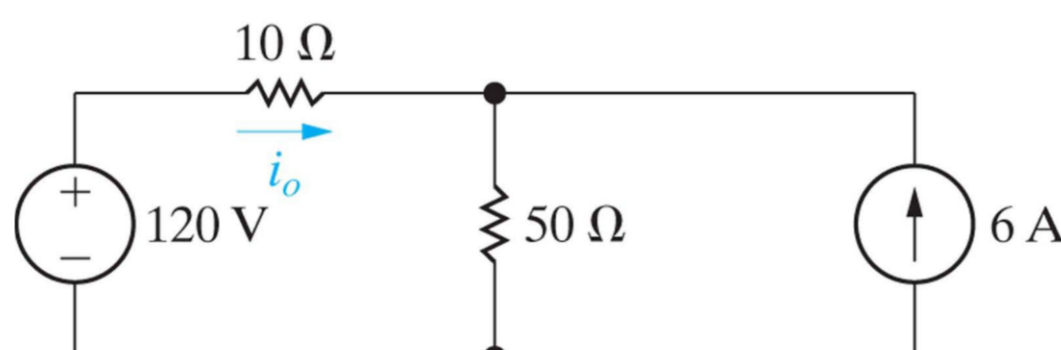
ChatGPT

Finished analyzing ✓

The current  $i_o$  in the circuit is  $-18$  A. This negative sign indicates that the direction of the current  $i_o$  is opposite to the assumed direction in the diagram. [↩]



You



Solve the circuit for  $i_o$ .

Circuit Theory Master 2

To solve for the current  $i_o$  in the provided circuit, we will first represent the circuit in SPICE syntax to create a netlist. Then, we can write a simulation program using PySpice to find the value of  $i_o$ .

The circuit consists of a voltage source of 120 V, a current source of 6 A, and two resistors with resistances of 10 Ω and 50 Ω.

```

from PySpice.Spice.Parser import SpiceParser

netlist_content = """
* Circuit Description
V1 1 0 DC 120
R1 1 2 10
R2 2 0 50
I1 2 3 DC -6
R3 3 0 1e-12 ; Small resistor to represent the current source's connection
"""

# We will write the netlist content to a file to be read by the SpiceParser
with open('circuit.netlist', 'w') as netlist_file:
    netlist_file.write(netlist_content)

# Parse the netlist and build the circuit
parser = SpiceParser(path='circuit.netlist')
circuit = parser.build_circuit()

# Run the operating point analysis
simulator = circuit.simulator(temperature=25, nominal_temperature=25)
analysis = simulator.operating_point()

# Calculate and print the output current i_o (through R1)
voltage_across_r1 = float(analysis['V1'])
current_through_r1 = voltage_across_r1 / 10 # Ohm's Law: V = IR, so I = V/R
print(f"Current i_o through the resistor R1: {current_through_r1:4f} A")
  
```

Current  $i_o$  through the resistor R1:  $-3.0000$  A



- ✓ Simple syntax for LLMs to write in
- ✓ Easy verification of model comprehension
- ✓ Get accurate answers with simulations
- ✓ Possibility for further modifications

## Circuit Diagrams On-The-Fly with chain-of-thought instructions

You

Design a circuit where a voltage source is connected to a resistor and a capacitor in series, and they are parallel to another resistor and a inductor in series.

Circuit Theory Master 2

To design the requested circuit, I'll start by creating a SPICE netlist that describes the circuit's components and their connections. After that, I'll provide you with Python code to generate a schematic diagram of the circuit using the `schemdraw` library.

```

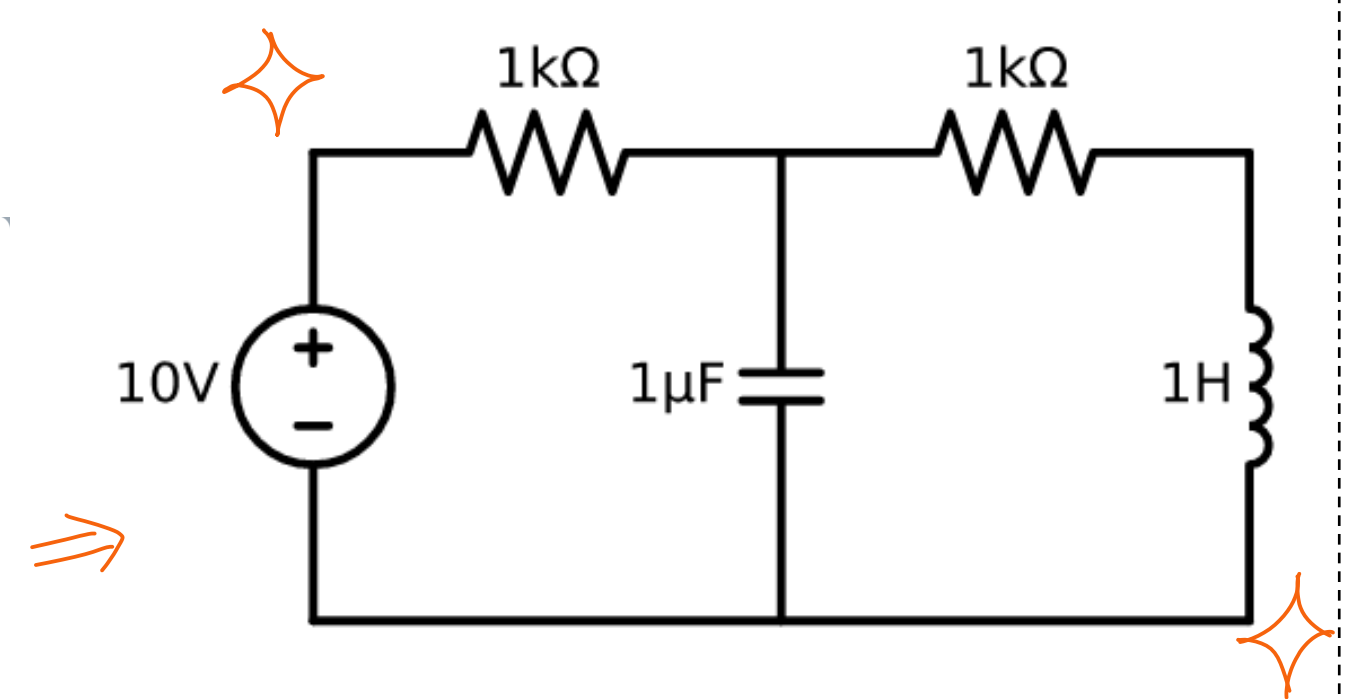
* Circuit Netlist
V1 1 0 DC 10V
R1 1 2 1k
C1 2 0 1u
R2 1 3 1k
L1 3 0 1H
  
```

```

import schemdraw
import schemdraw.elements as elm

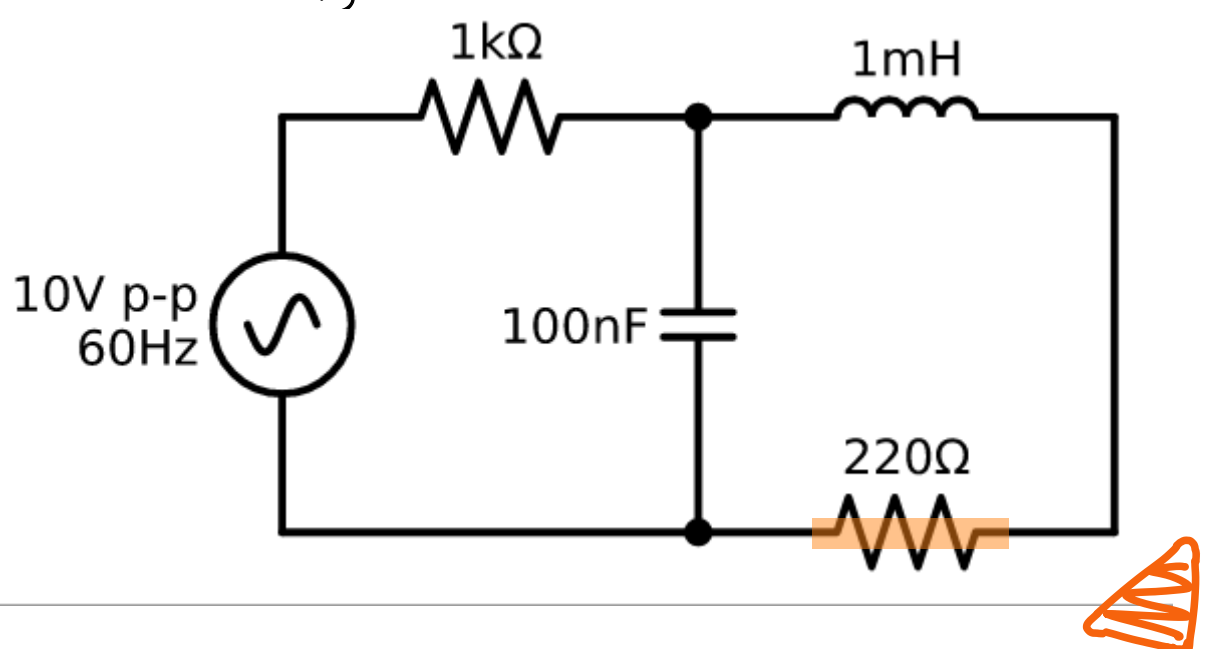
with schemdraw.Drawing() as d:
    d += (V1 := elm.SourceV().label('10V'))
    d += (R1 := elm.Resistor().right().at(V1.end).label('1kΩ'))
    d += (C1 := elm.Capacitor().down().at(R1.end).label('1μF'))
    d += elm.Line().to(V1.start)

    d += (R2 := elm.Resistor().right().at(R1.end).label('1kΩ'))
    d += (L1 := elm.Inductor().down().at(R2.end).label('1H'))
    d += elm.Line().to(C1.end)
  
```



## Solve for Complex Circuit Parameters while maintaining precision

Q:  $I_{avg}$  of The  $200\Omega$  Resistor



```

V1 n1 0 SIN(0 5 60)
R1 n1 n2 1k
C1 n2 n3 100n
L1 n3 0 1m
R2 n2 0 220
  
```

Simulate the circuit with PySpice

```

from PySpice.Spice.Netlist import Circuit
from PySpice.Unit import *
import numpy as np

circuit = Circuit('RLC Circuit')

circuit.SinusoidalVoltageSource('1', 'n1', circuit.gnd, amplitude=5@u_V, frequency=60@u_Hz)
circuit.R('1', 'n1', 'n2', 1@u_kΩ)
circuit.C('1', 'n2', circuit.gnd, 100@u_nF)
circuit.L('1', 'n2', 'n3', 1@u_mH)
circuit.R('2', 'n3', circuit.gnd, 220@u_Ω)

simulator = circuit.simulator(temperature=25, nominal_temperature=25)
analysis = simulator.transient(step_time=1@u_us, end_time=1@u_ms)

voltage_through_R2 = analysis['n3']

# Calculate the average current over one cycle
average_current_R2 = np.mean(voltage_through_R2) / 220
print(f"Average current of the 220Ω resistor: {average_current_R2 * 1_000_000}uA")
  
```

Average current of the 220Ω resistor:  $729.1792\mu\text{A}$

Break down the circuit by writing a SPICE netlist

Get accurate answers!