

Phase 1

Objective

Develop your static IR drop solver using modified nodal analysis in Python.

Your solver

Your IR drop solver must calculate voltages at each node of the power grid by solving a system of linear equations represented by the equation $GV = J$. You may use any Python libraries to solve $GV = J$, but you must develop your own code to build the G matrix and the J vector.

Inputs

Your code must read a SPICE netlist representation of the power grid as described below:

The SPICE netlist is represented with the following format:

`<electrical_component> <node1> <node2> <value>`

SPICE format description: The first letter of the term `<electrical_component>` denotes the component type. The component can be a resistor, voltage source, or a current source denoted by the letters R, V and I, respectively. If the node is **0** the node is a reference/ground node.

The node names have location encoding of the node including the x and y coordinate and the metal layer as described below

Node name format description:

`<netname>_<layer-idx>_<x-coordinate>_<y-coordinate>`

Outputs

Your IR drop solver must generate one output file the node name and the corresponding voltage at that node. The filename must have the .voltage filename extension. (Example: circuit.voltage)

The values must be of the format,

`<node_name> <voltage_value>`

Benchmarks

Benchmarking circuits for the IR drop solver are available in

The two simple benchmarks can be verified by hand calculations for the purposes of debugging. The rest of the benchmarks are bigger and cannot be solved by hand.

Validation of your solver

Your solver can be validated by running a SPICE simulation of the netlist and comparing the values of voltages at every node. To run SPICE, please use chipshub. The explanation on accessing chipshub for SPICE is available on [MiniProject2-Phase1](#).

A script is provided on on Cavas here to read your generated output file and spice generated output file and plot scatter plots to visualize discrepancies.

Ideally the voltages obtained from your ir solver and spice simulation must be equal for each node, forming a 45 degree line with the x axis.

Please include the generated scatter plots in your report.

Submission process:

Your submission must include two things:

- 1) Script: Your Python script. The top level file must be named ir_solver.py and must read in one input spice file and must generate one output file. The following is the command that must run your solver which takes two arguments. The input spice netlist and the path to the generated output voltage file.

```
python3 ir_solver.py --input_file <spice_netlist_name> --output_file <voltage_file_name>
```

- 2) Report: The report should include half to one page of text describing the challenges you faced while developing code. Include snippets and explanations for what different parts of the code does. Include scatter plots for testcase17 and 18.

Please submit your code and report via Canvas as one .zip file containing all the Python scripts/source code and the report. The zip file should also contain the requirements.txt with packages that are required to run the script as described in Mini Project 1.