

# KiCad: Schematic Capture & SPICE Simulation

BME253L - Fall 2025 - Palmeri

Duke Biomedical Engineering

2025-09-23

## Table of contents

Learning Objectives . . . . .	1
Voltage Divider Circuit . . . . .	1
Capture the Schematic . . . . .	1
Simulate Maximum Power Transfer . . . . .	2
Multi-Source Circuit . . . . .	3
Capture the Schematic . . . . .	3
Simulate the Operating Point (OP) Analysis . . . . .	4
Simulate DC Sweeps . . . . .	4

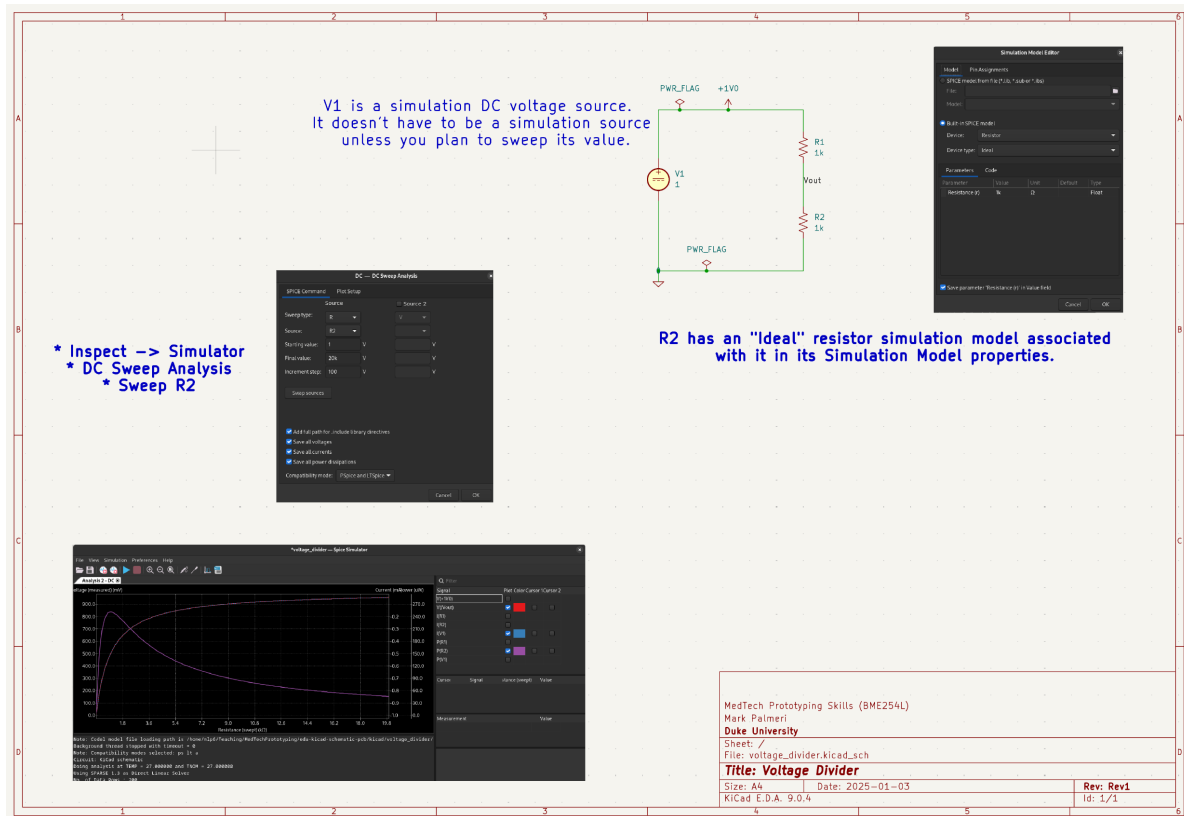
## Learning Objectives

- Create a schematic in KiCad
- Annotate and assign component values
- Set up and run a SPICE simulation
- Analyze simulation results

## Voltage Divider Circuit

### Capture the Schematic

- Create a new KiCad project.
- Fill in the schematic metadata that is populated in the right foot of the sheet.
- Create the circuit shown below using the schematic editor. Note that all of the sources chosen are SPICE DC sources.



**i** Gradescope!

Upload a screenshot of your complete schematic window to Gradescope.

## Simulate Maximum Power Transfer

In a recent problem set, you analytically solved for the load resistance that maximizes power transfer in a voltage divider circuit. In this section, you will simulate the same circuit in KiCad and verify your results.

- Sweep the value of  $R_2$  (representing the load resistance) from  $0.1\text{ k}\Omega$  to  $10\text{ k}\Omega$  in increments of  $100\Omega$ , for a fixed  $R_1$  of  $1\text{ k}\Omega$  (representing the source resistance).
- Plot the power dissipated in  $R_2$  as a function of  $R_2$  and identify the value that maximizes power transfer.

### Gradescope!

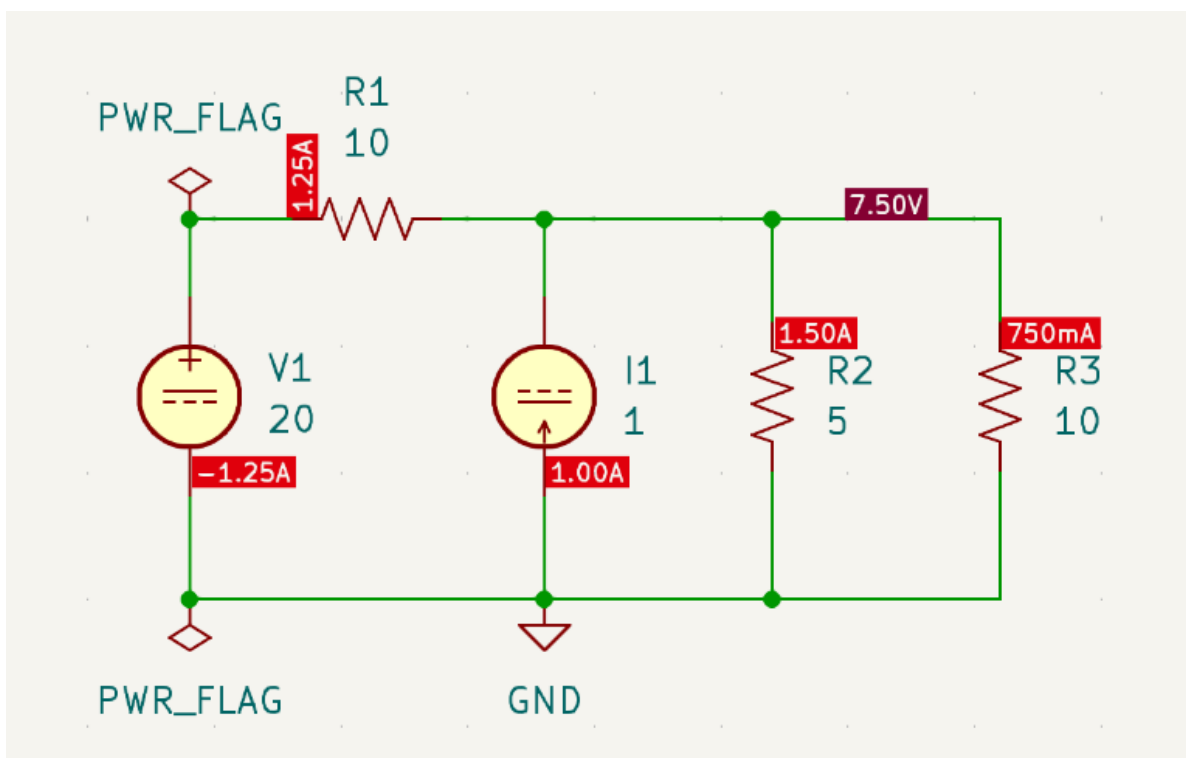
Upload a screenshot of your swept resistance plot demonstrating maximum power transfer to Gradescope.

## Multi-Source Circuit

The circuit shown below was one of the problems in your recent exam.

### Capture the Schematic

- Create a new KiCad project.
- Fill in the schematic metadata that is populated in the right foot of the sheet.
- Create the circuit shown below using the schematic editor. Note that all of the sources chosen are SPICE DC sources.



 Gradescope!

Upload a screenshot of your complete schematic window to Gradescope.

### Simulate the Operating Point (OP) Analysis

- Operating Point simulations are appropriate for circuits that have DC sources and single-valued element current / voltage values.
- Run an OP simulation for this circuit and verify your results, as shown annotated in the circuit above and summarized below.

```
Note: Code model file loading path is /home/mlp6/Teaching/BiomedicalElectronicsMeasurements/labs/multi_source/
Note: Compatibility modes selected: ps lt a
Circuit: KiCad schematic
Doing analysis at TEMP = 27.000000 and TNOM = 27.000000
Using SPARSE 1.3 as Direct Linear Solver
Reference value : 0.00000e+00
No. of Data Rows : 1
```

Simulation results:

I(r1):	1.25A
I(v1):	-1.25A
I(r3):	750mA
I(r2):	1.5A
I(i1):	1A
P(i1):	-7.5W
P(r2):	11.25W
P(r3):	5.625W
P(v1):	-25W
V(net_i1-pad2_):	7.5V
V(net_r1-pad1_):	20V
P(r1):	15.625W

 Gradescope!

Upload a screenshot of your operating point (OP) simulation results to Gradescope.

### Simulate DC Sweeps

- Perform a new analysis that sweeps several components over a range of values:
  - Sweep  $V_1$  from 1 to 10 V in increments of 0.2 V
  - Sweep  $I_1$  from 1 to 5 A in increments of 0.2 A
- Plot:
  - $I_{R_2}$

$$-P_{V_1}$$

$$-P_{I_1}$$

You should see results similar to those shown below.



**i** Gradescope!

Upload a screenshot of your DC sweep simulation results to Gradescope.