

KiCad: Schematic Capture & SPICE Simulation

BME253L - Fall 2025 - Palmeri

Duke BME

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.

V1 is a simulation DC voltage source. It doesn't have to be a simulation source unless you plan to sweep its value.

* Inspect -> Simulator
* DC Sweep Analysis
* Sweep R2

R2 has an "Ideal" resistor simulation model associated with it in its Simulation Model properties.

MedTech Prototyping Skills (BME254L)
Mark Palmeri
Duke University
Sheet: /
File: voltage_divider.kicad_sch
Title: Voltage Divider
Size: A4 Date: 2025-01-03 Rev: Rev1
KiCad E.D.A. 9.0.4 Id: 1/1

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 100Ω , 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: Code1 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.