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

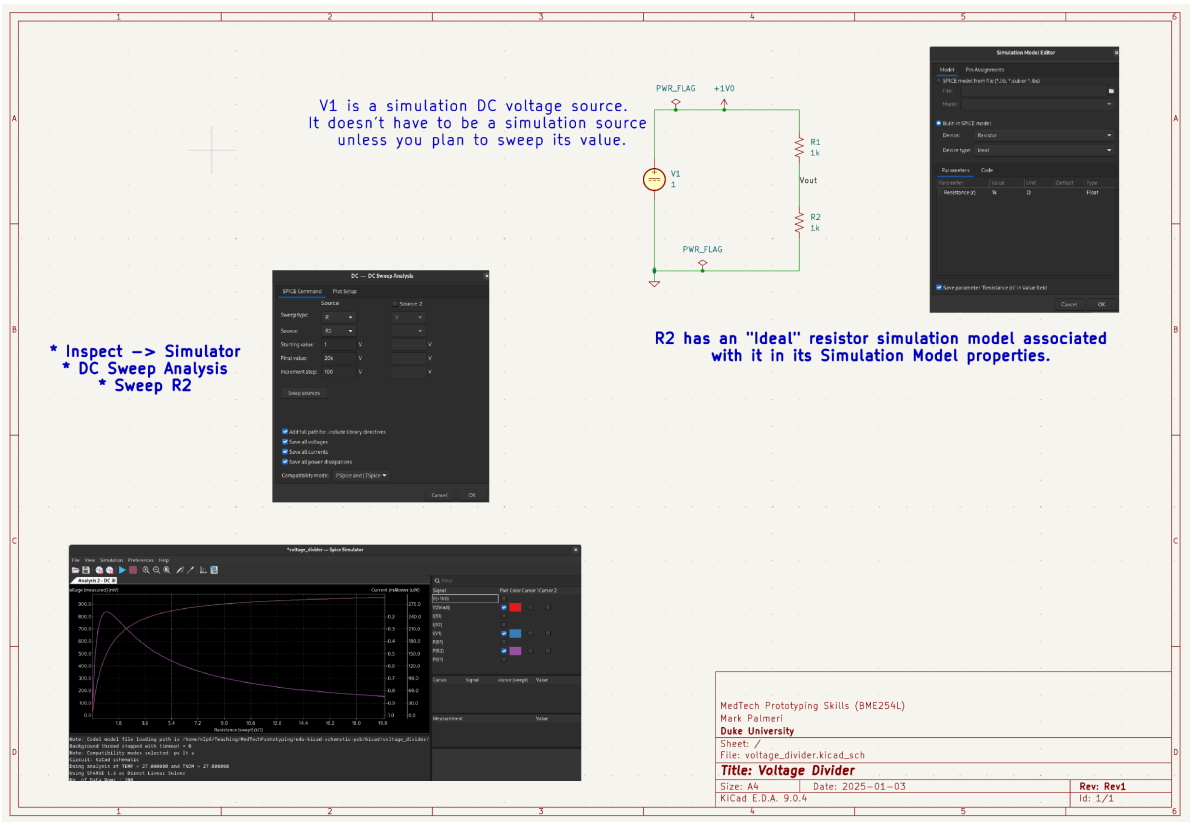
Learning Objectives

- 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.



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 to 10kΩ in increments of 100Ω, for a fixed R_1 of 1kΩ (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.

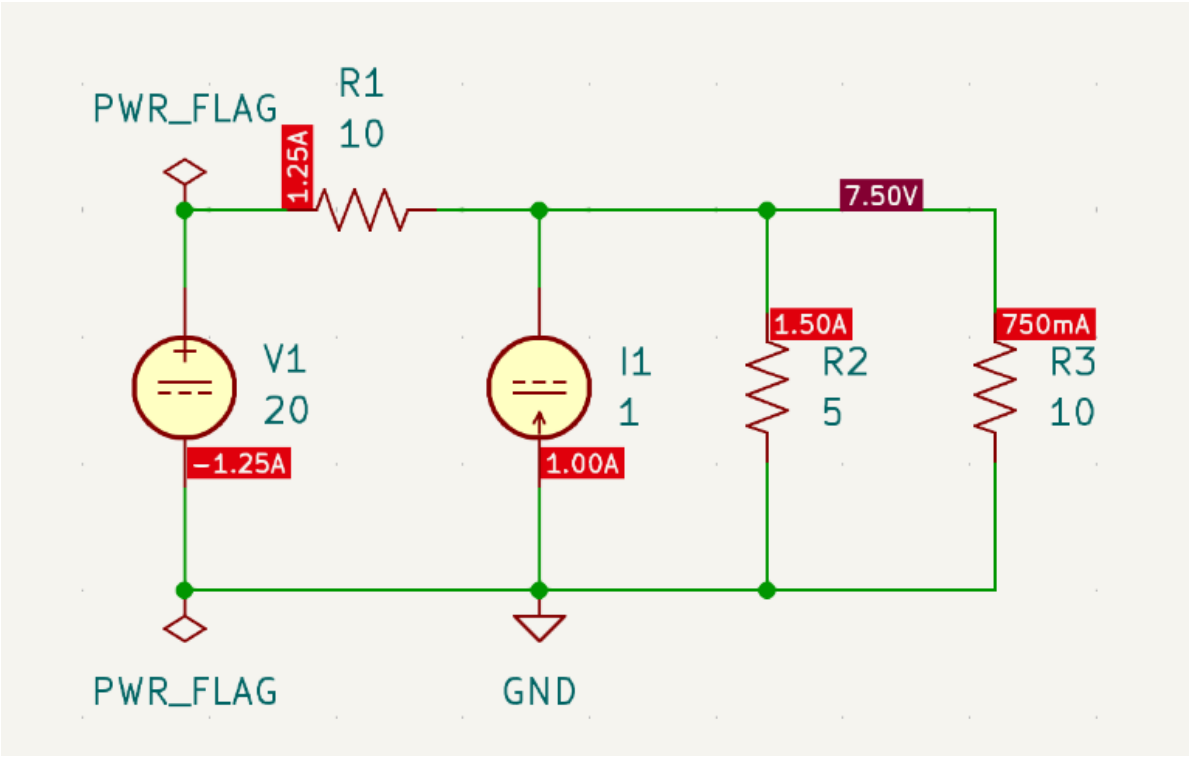
> **Gradescope!**
>
> Upload a screenshot of your swept resistance plot demonstrating maximum power transfer to Gradescope.

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: Codel 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
```

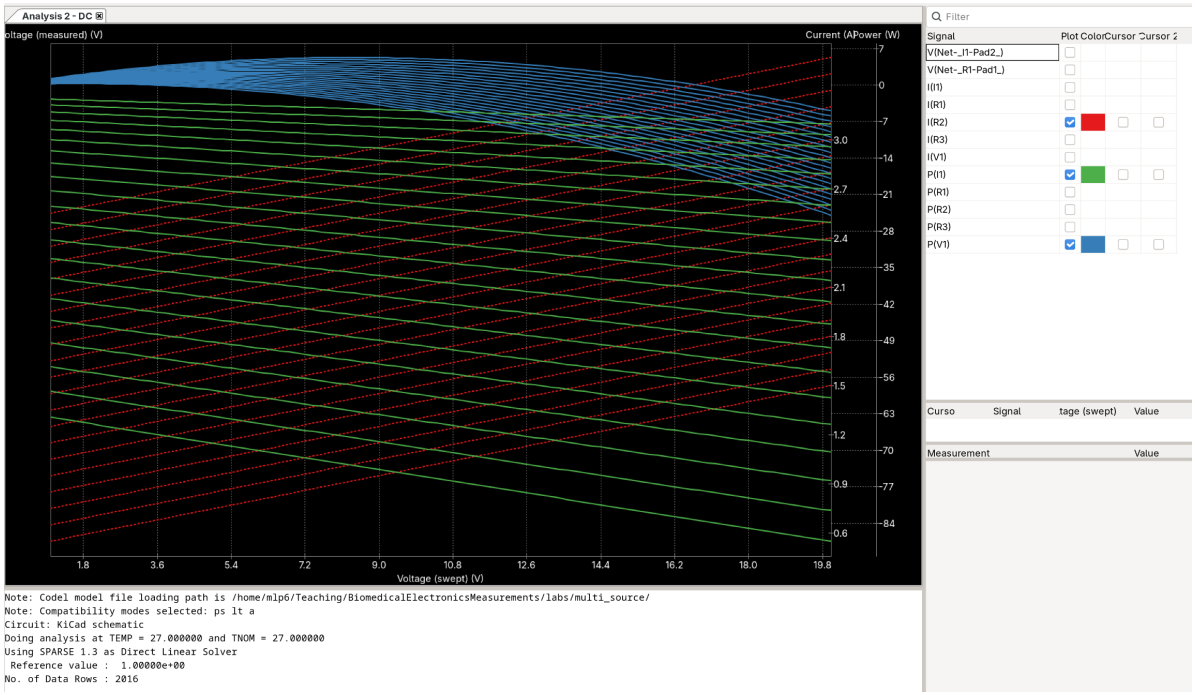
> **Gradescope!**
>
> Upload a screenshot of your operating point (OP) simulation results to Gradescope.

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.



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