CENG 215 Circuits and Electronics

LAB #3 Feuille

Place: PC Lab

Aim

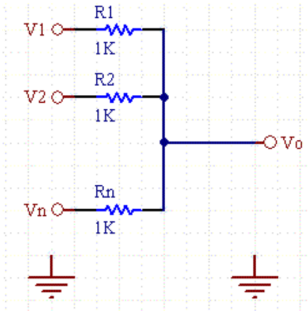
To build and analyze various resistive networks in PySpice and to compare the analysis results with the theoretical analysis results.

Materials/Devices:

PySpice

Work to be done:

The following circuit is a "resistive adder" used to find the sum of *n* different signals.



1. We want to find the sum of three signals like $V_i(t) = A_i \sin(2\pi f_1 t)$, i=1,2,3. Taking A_1=2 volt, A_1=5 volt, A_3=8 volt, $f_1 = f_2 = f_3$=50 Hz, simulate and plot $V_o(t)$.

2. We want to find the sum of 50 signals such that:

   $V_i(t) = A_i \sin(2\pi f_i t)$, $i \in \{1,2,3,\ldots,50\}$
   where $A_i = (i+1)$ volt and $f_i = 50(i+1)$ Hz

   Simulate the circuit and plot $V_o(t)$.

Final Remarks

Which of the following simulation methods do you prefer and why?

```
analysis = simulator.operating_point()
analysis = simulator.transient(step_time=100@u_us, end_time=100@u_ms)
```

Can you use operating_point() method in the above simulations? Discuss.