EE230- Analog lab (Labwork-3)
Spring Semester: Year 2021-22

January 22, 2022

Instructions:

- Write your netlists and show the simulation results of each question to the evaluating TA during the lab session on Jan 27, 2022.
- **No Additional time will be given.**
- **You can refer:** NGSPICE tutorial, model files of op-amp "ua741.txt" and diode "1n914.txt" uploaded on the course moodle / MS Teams channel and your written netlists of previous experiments / homeworks.

---

1. **Half-wave Precision Rectifier**

   (a) Simulate the circuit "Improved half-wave rectifier-A" shown in figure [1] using Ngspice. Apply a sinusoidal input signal, V_i of $10V_{pp}$ and $1kHz$. Plot V_o , V_{o1} and V_i waveform on the same plot.

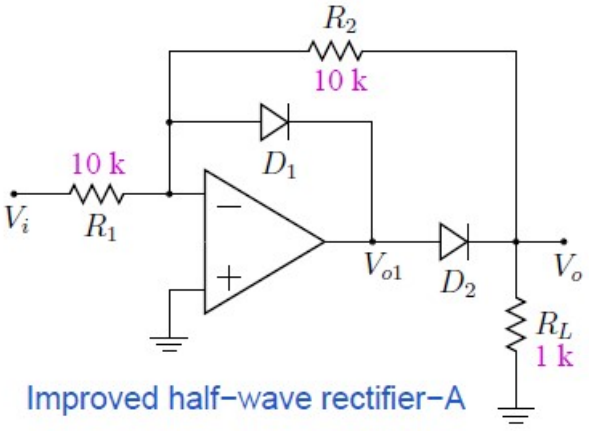

Figure 1: Improved Half Wave Rectifier-A Circuit

(b) Simulate the circuit "Improved half-wave rectifier-B" shown in figure [2] using Ngspice. Apply a sinusoidal input signal, V_i of $10V_{pp}$ and $1kHz$. Plot V_o , V_{o1} and V_i waveform on the same plot.

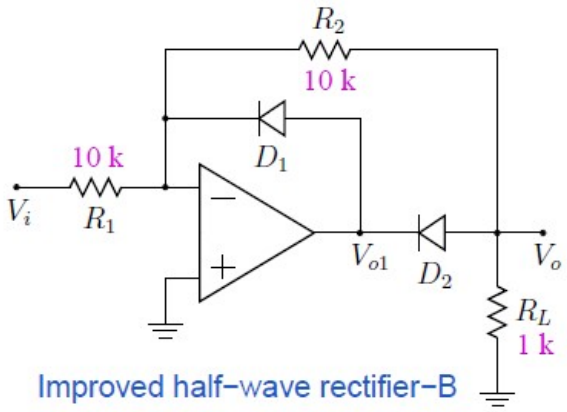

Figure 2: Improved Half Wave Rectifier-B Circuit

2. **Full-wave Precision Rectifier**

(a) Use "Improved half-wave rectifier-B" as a sub-circuit and simulate the "Full-wave rectifier" circuit shown in figure [3] using Ngspice. Apply a sinusoidal input signal, V_i of $10V_{pp}$ and $1kHz$. Plot V_o and V_i waveform on the same plot.

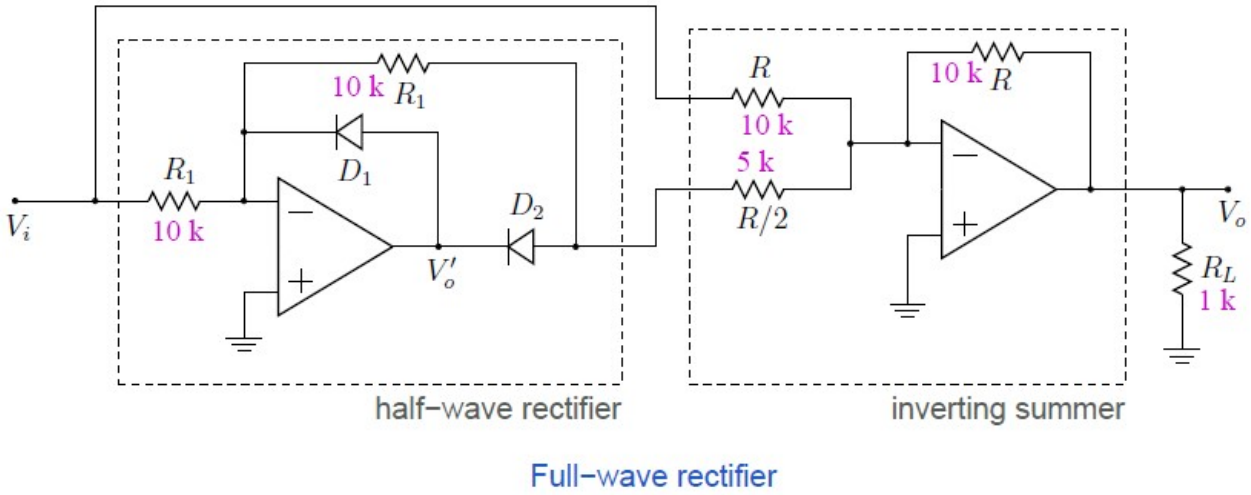

Figure 3: Full Wave Rectifier Circuit