

## PHY 335, Unit 4 Simulation and PCB desing

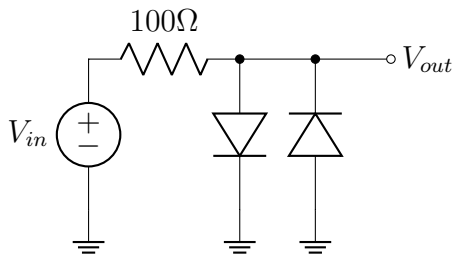
### Mini Lecture topics:

- Schematic capture, netlists
- Intro to SPICE, LTSPICE
- Simulation modes: Transient, AC analysis
- Basic PCB design and manufacturing
- Intro to KICAD

For this unit, please install the following software before the first day. Bring your laptop with you. I also recommend you bring a mouse! Please install these software packages:

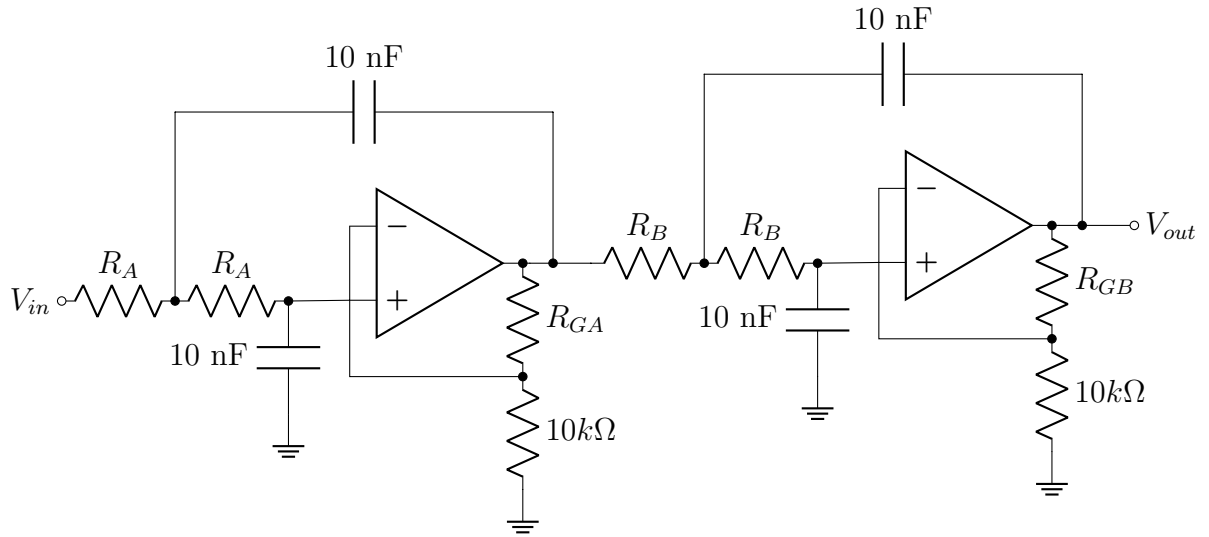
- **LTSPICE:** <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>  
For Windows, install the version on the linked page. For Linux, you have to run into via WINE (install via your package manager). There is a native MacOS version, but I recommend the Windows version run via WINE even for MacOS.
- **KICAD:** <http://kicad.org/download/> On the webpage you will find installers for Windows, MacOS and the most common linux distributions.

1. Build the following schematic in LTSpice.  $V_{in}$  should be an AC sine wave with 5Hz and an amplitude of 1V. Do a transient analysis (1s). What is the maximum current through the resistor. Reproduce a plot of  $V_{out}$  vs time and  $I_R$  vs time. Include a picture of the schematic in LTSpice into your report.



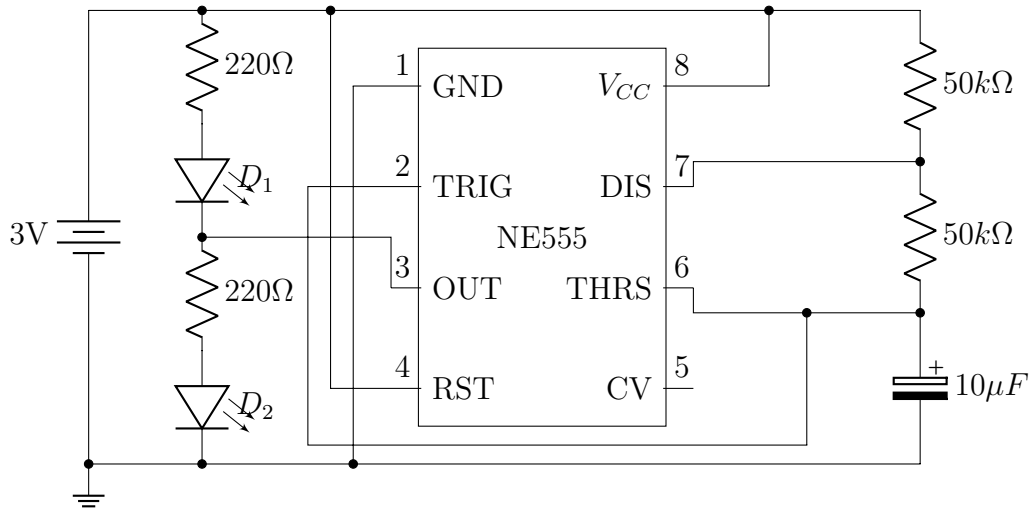
2. Build the following schematic in LTSpice three times (build it once and then copy), with the values listed below. Do an AC analysis from 1 Hz to 10 kHz. Reproduce the result for the gain and phase (Bode plots) for the three variants. How do these three filters differ? (Include a picture of the schematic in LTSpice

into your report.) Use OP37 as the opamp model.



- (a)  $R_A = 110k\Omega$ ,  $R_{GA} = 845\Omega$ ,  $R_B = 100k\Omega$ ,  $R_{GB} = 7.68k\Omega$
- (b)  $R_A = 158k\Omega$ ,  $R_{GA} = 1.54k\Omega$ ,  $R_B = 158k\Omega$ ,  $R_{GB} = 12.4k\Omega$
- (c)  $R_A = 267k\Omega$ ,  $R_{GA} = 5.76k\Omega$ ,  $R_B = 154k\Omega$ ,  $R_{GB} = 16.5k\Omega$

3. The following schematic is a led blinker. Simulate it in LTspice (transient analysis). Try to understand how the circuit works by reading the datasheet and compare with the simulation. Describe the function in your report.



4. Schematic capture the NE555 design above in KICAD and design a PCB for it. (Alternative, construct a circuit of your liking with at least comparable complexity, and design a PCB for it). Put a screenshot of the schematic and of the pcb layout in your lab report.

5. Bonus task (10 points). Will be discussed in class.