

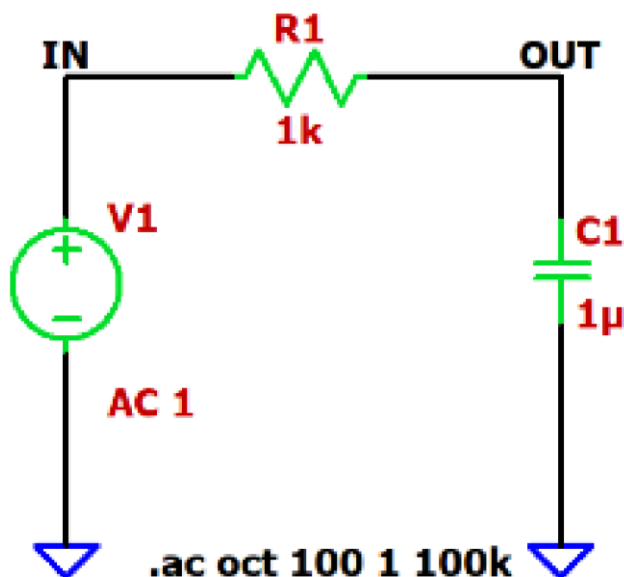
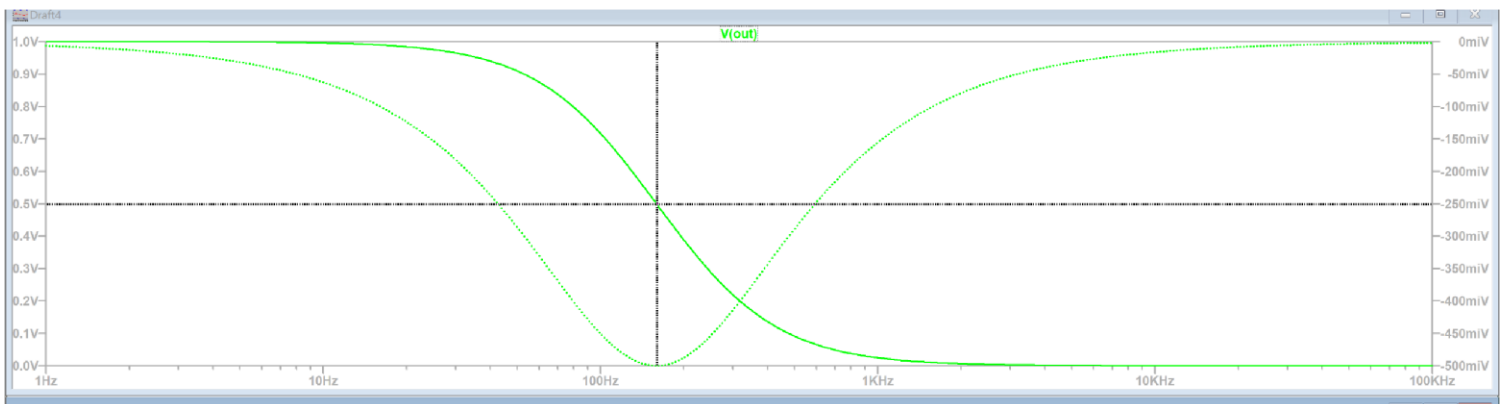
# Circuit Simulation

## Q1 Circuit Simulation

Prove your skill set using LTSpice (preferred Circuits simulation program) or equivalent simulation program (i.e. PSpice or MultiSim..)

### Q1.5 AC analysis

I can use AC analysis to find the frequency response of an RC or RL filter. In this circuit I used an AC power supply that was in series with a  $1\text{k}\Omega$  resistor and  $1\mu\text{F}$  capacitor. The circuit has a resistor and capacitor, so it is an RC circuit. When running the simulation we can see that two sinusoidal graphs are created.



## Grading Rubric for Q1.5 from F22 Intro to ECSE

1.5 **AC analysis** 5 / 5 pts

- ✓ **+ 1 pt** Proficient
- + 0 pts Learning (need one or more goals to be met under this objective)
- + 0 pts Need significant improvement

✓ **+ 1 pt** I can change my schematic and plot background to white and cut and paste on an external document

✓ **+ 1 pt** I can change the line thickness and color of my schematic and simulation output

✓ **+ 1 pt** I can label the simulation output clearly with the circuit schematic component names

✓ **+ 1 pt** I can intentionally show the most relevant part of a simulation by changing the simulation output window

- + 0 pts Incomplete
- + 0 pts change plot background to white